AN-1448
APPLICATION NOTE

One Technology Way • P.O. Box 9106 • Norwood, MA 02062-9106, U.S.A. • Tel: 781.329.4700 • Fax: 781.461.3113 • www.analog.com

Recommended Rework Procedure for PBGA Packages

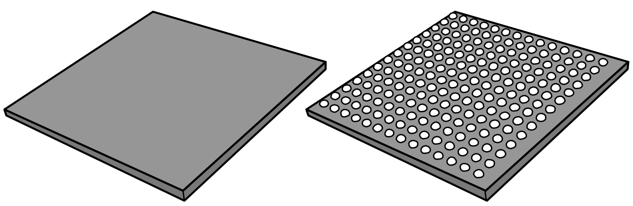
By Ramon Navarro

INTRODUCTION

This application note provides a recommended procedure for removing a plastic ball grid array package (PBGA) from a printed circuit board (PCB).

PACKAGE DESCRIPTION

The PBGA is a package format with a main distinguishing characteristic of an array of solder balls as a mode of interacting with the substrate (such as a PCB). This feature gives the PBGA an advantage over other package formats with different pin configurations (such as single, dual in line, and quad types) in terms of achieving higher pin count density. Interconnection inside the PBGA package is implemented through wire bonding or flip chip technology. The die for the PBGA, which contains the integrated circuit, is encapsulated in mold compound.



15521-001

Figure 1. Illustration of a PBGA Component

REWORK FOR PGBA COMPONENTS

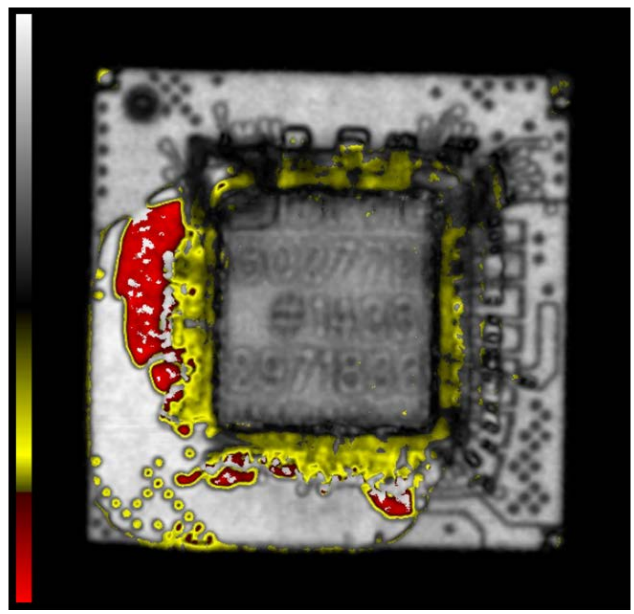
If defects are present after attaching the PBGA component to the PCB, rework to remove the defective component and replace it with a functioning device. Before removing the device, apply heat on the defective component until the solder joints are liquefied, facilitating the removal of the component from the board.

The conventional rework procedure is as follows:

1. Board preparation.
2. Component removal.
3. PCB land clean up.
4. Application of solder paste.
5. Component alignment and placement.
6. Component attachment.
7. Inspection of rework.

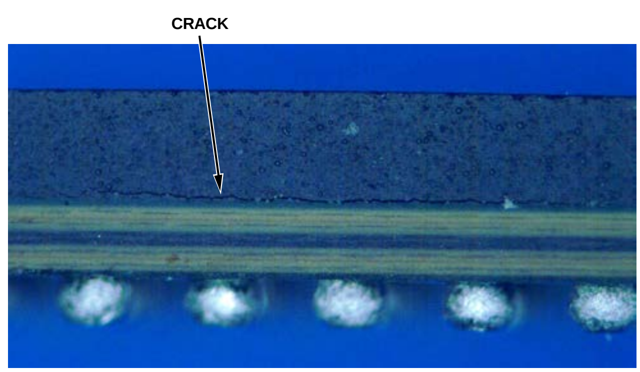
COMPONENT REMOVAL AND DELAMINATION

The manner of component removal can impart mechanical stress on the PBGA and/or the PCB. Remove the defective device carefully to avoid damage to the PCB, nearby components, and to the device itself, particularly if a user plans to subject the device to failure analysis. Excessive stresses on the PBGA component, such as heating the component beyond the specified peak temperature or overexposure to heat, can lead to package delamination or external physical damage (see Figure 2 and Figure 3). For a component that must undergo further analysis, delamination caused by improper component removal makes identifying the true failure mechanism more difficult. Consequently, proper removal of the defective component is necessary to conduct effective failure analysis.



15521-002

Figure 2. Delamination Between Substrate and Encapsulating Mold Compound for PBGA Component Due to Excessive Heating (Observed Through Scanning Acoustic Microscopy)



15521-003

Figure 3. Low Magnification Image of Crack on PBGA Due to Excessive Heating (Side View)

BOARD PREPARATION

It is strongly recommended to dry bake the PCB assembly prior to the rework procedure to eliminate residual moisture. If not removed, this residual moisture can cause damage to the component during reflow due to popcorn cracking. Bake the PCB assembly for a minimum of 4 hours at 125°C, as long as these conditions do not exceed those specified for other components on the PCB. When the conditions exceed other component specifications, use alternative baking conditions detailed in the Joint Industry Standard IPC/JEDEC J-STD-033.

COMPONENT REMOVAL

Different tools are available for component removal. Component removal can involve heating the component until solder reflow occurs, then mechanically removing the component while the solder is still in a liquid state. A programmable hot air rework system can provide controlled temperature and time settings.

Follow the temperature profile used for component attachment when performing the rework. The rework temperature must not exceed the peak temperature specified on the moisture sensitivity level (MSL) label. It is acceptable to shorten the duration of heating (for the liquidus region, for example) as long as full solder reflow is attained. Keep the time at which the package temperature is at the solder reflow region to less than 60 sec. Maintain vacuum pressure for the pick-up tool less than 0.5 kg/cm^2 to prevent the component from lifting out before full reflow is reached and to avoid pad lift. Do not reuse any components that are removed from the PCB.

Control the rework temperature to avoid damage to the PBGA component and the PCB. Take into consideration that the PBGA may heat faster compared to lead frame-based packages, such as standard outline integrated circuits (SOIC) and lead frame chip scale packages (LFCSP), due to its thermal mass. Note that covering the area immediately surrounding the component with thermal tape provides further protection. In addition, it is recommended to heat the PCB underneath to reduce the temperature difference between both PCB sides and to minimize board warping.

When defining the rework tool settings, perform characterization of the temperature profile. This characterization is particularly important when reworking a specific component for the first time. It is also necessary to perform characterization on PBGA components with different body sizes, solder composition, or different PCB materials, configurations, size and thicknesses, because these may have different thermal mass values.

Characterization must include monitoring of the temperature, time, and other equipment tool settings (see Figure 4). Attach thermocouples to different parts of the board assembly, such as the top of the PBGA component and the top of the PCB (see Figure 5). Analyze the time and temperature profile data and obtain working parameters for the component removal from the evaluation.

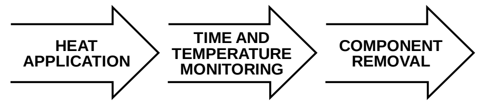


Figure 4. Simplified Flowchart for Component Removal Evaluation

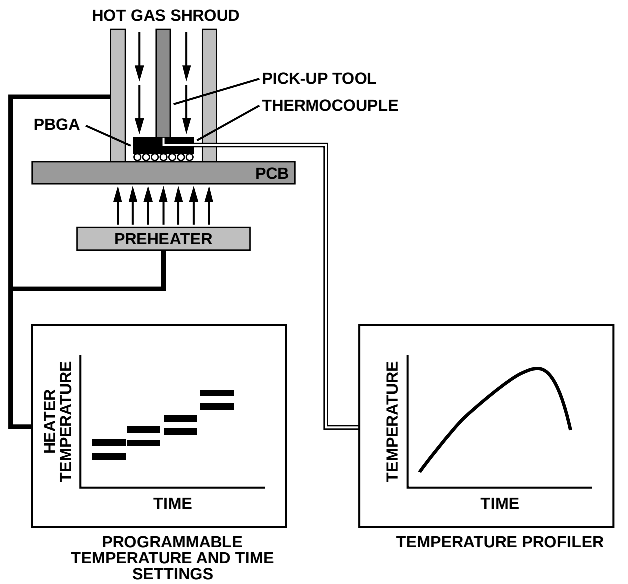


Figure 5. Illustration of a Component Removal Characterization Setup

PCB LAND CLEAN UP

The lands on the PCB exhibit excess solder after the removal of the PBGA component, and must be prepared prior to attachment of a new replacement PBGA component. The lands may be prepared in two steps:

1. Desoldering. Remove the excess solder from the lands through the use of desoldering braid in conjunction with a blade type soldering iron (see Figure 6). Choose a blade width that matches the maximum width of the component footprint. The blade temperature must be low enough to avoid damaging the circuit board. Solder flux can be applied on the lands before using the desoldering braid and soldering iron to remove the excess solder.

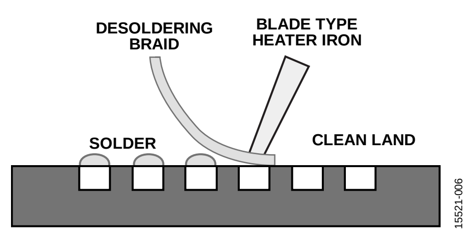


Figure 6. Desoldering the PCB Lands

2. Cleaning. Apply cleaning solvent on the site and wipe clean using a lint free cloth. Use a solvent that is specific to the type of paste used in the original assembly.

APPLICATION OF SOLDER PASTE

Application of solder paste prior to attaching the replacement PBGA component to the board aims to replace the solder paste applied in the initial board assembly to maintain the reliability of the PBGA solder joint. It is important to apply a consistent volume of solder paste for each solder ball to avoid planarity issues when the PBGA is attached to the board.

A stencil can be used to apply solder paste to the PCB lands. Accuracy of the alignment of stencil alignment is crucial for uniform reflow solder processing. Use the same PBGA aperture geometry and stencil thickness used in the board assembly. Use trapezoidal apertures (see Figure 7) to ensure uniform paste release and reduce smearing.

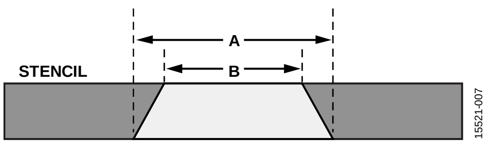


Figure 7. Stencil Aperture Geometry (A is Longer than B)

There are circumstances where the use of a stencil to apply solder paste on the PCB lands uniformly and accurately may not be feasible, particularly for boards with high component density, or with tight geometries. In this case, consider applying solder paste onto the solder balls at the bottom of the components. This can be achieved through the use of a stencil to apply solder paste over the tips of the solder balls, or through dispensing of the solder paste for all solder balls (see Figure 8 and Figure 9). A specially designed jig and/or rework equipment can be used for this purpose.

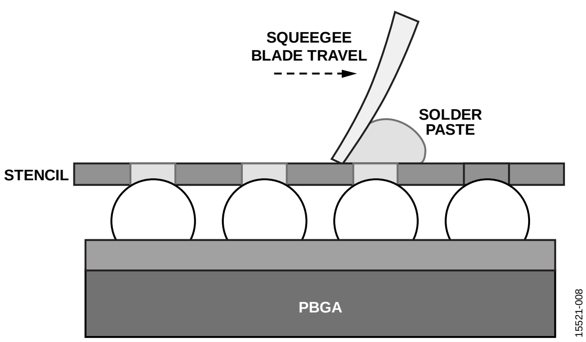


Figure 8. Solder Paste Stencil Printing on Solder Balls

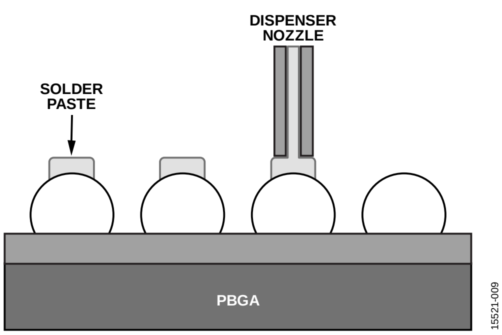


Figure 9. Solder Paste Dispensing on Solder Balls

COMPONENT ALIGNMENT AND PLACEMENT

Accurate placement of the component to the board is crucial. Pick and place equipment with a split beam optical system facilitates the alignment of the PBGA to the board. This type of imaging system involves the superimposition of the image of the PBGA solder balls on that for the PCB lands (see Figure 10). The placement machine must have the capability of allowing the user to make fine adjustments along the x- and y-axes, including rotation.

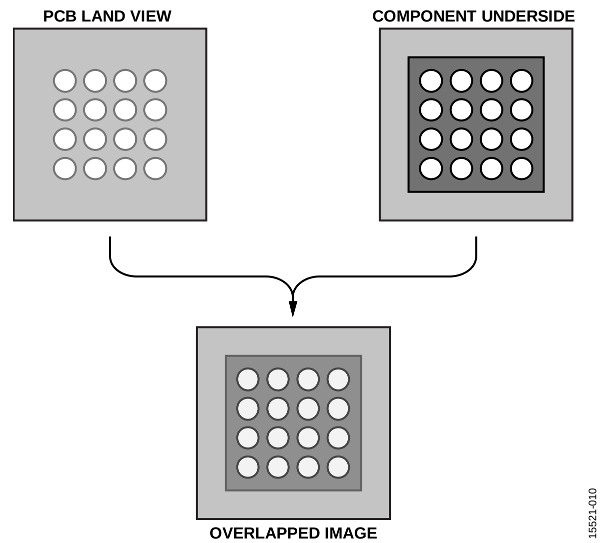


Figure 10. PCB and Component Alignment Using a Split Beam Optical System

The accuracy of placement of the component is dependent the equipment or the process used. Although PBGA packages tend to self align during reflow, ensure that the placement offset is less than 50% of the PCB land width. Gross misalignment may result in electrical short circuits due to solder bridging.

COMPONENT ATTACHMENT

Because all the reflow parameters are optimized, use the same thermal profile developed during the original attachment.

INSPECTION OF REWORK

Inspect the assembled PBGA for defects after reflow, such as misalignment or damage. Perform X-ray inspection to check for problems, such as solder bridging and solder balling. Perform electrical test verification if necessary to verify that the component functions properly.