TECHtips

Electronics Manufacturing Insights from ACI Technologies, Inc.

Pad Cratering

A customer contacted the Helpline to perform analysis on a lead-free assembly which exhibited intermittent functionality. The lead-free assembly exhibiting intermittent functionality when pressure was applied to the ball grid array (BGA) packages. Industrial adaptation of a Restriction of Hazardous Substances (RoHS) compliant solder standard has created a new host of failure modes observed in lead-free assemblies. Pad cratering occurs when fractures propagate along the epoxy resin layer on the underside of the BGA connecting pads. While originating from process, design, and end use conditions, it is the combination of a rigid lead-free solder with inflexible printed circuit board (PCB) laminates that has advanced the prevalence of this condition. Pad cratering is simply the result of mechanical stress exceeding material limitations.

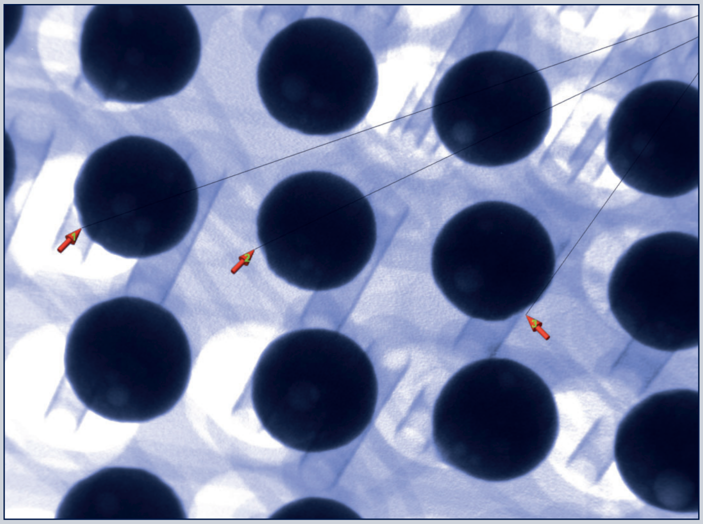

Figure 1: X-ray inspection of BGA showing flattened appearance of several solder balls.

An X-ray inspection revealed subtle anomalies at the interface where a ragged and flattened appearance of several solder balls was observed (Figure 1). An endoscopic examination further confirmed that gapping at the interface had indeed occurred (Figure 2). To confirm this condition, microsection failure analysis was performed using enhanced scanning electron microscopy (SEM) magnification. Pad cratering was observed throughout the BGAs. In addition, fractures at the solder joint/gold pad interface and within the PCB were observed indicating an overstress condition (Figure 3).

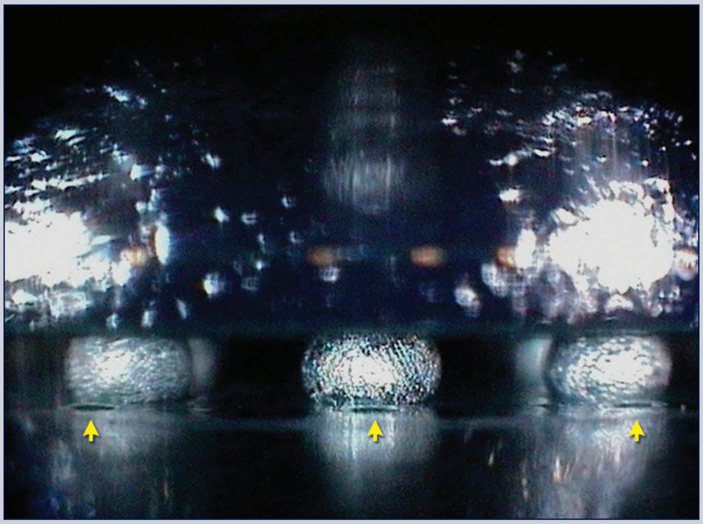

Figure 2: Endoscopic examination confirms gaps at the interface as indicated by the yellow arrows.

While it is unlikely that this specific occurrence was the root cause of the assembly failure, it may be indicative that a complete separation has occurred elsewhere; creating intermittent failures. Solder fracturing along the path of these “craters” can compromise connecting traces and vias resulting in an electrical failure. An examination of the metallurgical structure of the fractured solder joint was found to be consistent with a well-controlled lead-free soldering process. This further strengthened the conclusion that the solder fracture resulted from pad cratering.

Pad cratering is found to be more common in BGAs for a variety of reasons.

continued on next page

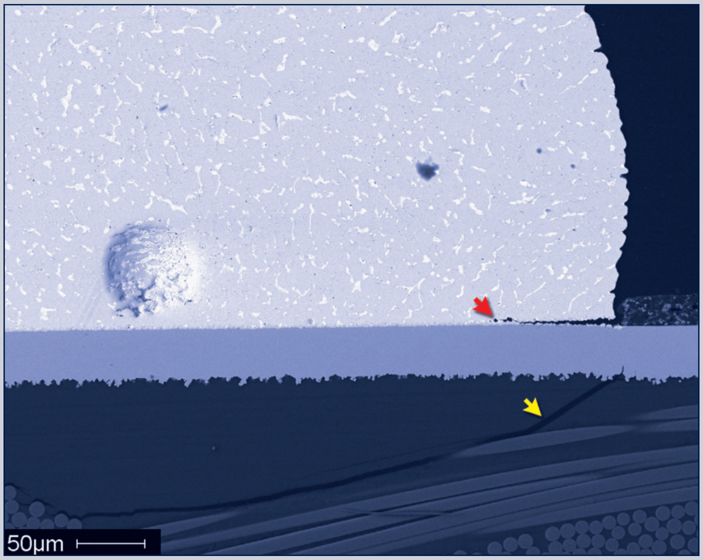

Figure 3: SEM micrograph showing fractures at the solder/gold pad interface (red arrow) and within the PCB (yellow arrow).

- A BGA solder joint does not have the inherent stress relief as would be found on a gull wing lead connection.
- BGA solder lands typically have a low individual surface area.
- BGAs are more susceptible to damage
    - as a result of vibration due to high mass
    - or deflection of the substrate due to high footprint area.

Prevention of pad cratering requires measures that either reduce the stress on the interconnect site or mitigate the effects of stress. Reducing stress may require layout modification of the PCB or redesign of the next higher assembly. Adhesives can be applied as underfills or as external package perimeter bonding to provide additional mechanical strength, but this application may not be practical and has not yet proven to be enough in certain situations.

The material composition (fillers) and fabrication methods (cure process) require changes to increase the thermal resistance and dimensional stability in laminates for lead-free processing. These changes also tend to increase the propensity for pad cratering, especially when coupled with the lead-free solders.

Changes to the material specification for the PCB laminate can also be an effective strategy to prevent pad cratering.

IPC-4101 (Specification for Base Materials for Rigid and Multilayer Printed Boards) requires a minimum trace peel strength but may not be enough for all designs and service environments. Peel strength testing per IPC-TM-650 Method 2.4.8 (Peel Strength of Metallic Clad Laminates) Condition A (as received) can be used as a comparative test between different laminate materials. Condition B (after thermal stress) can be used to compare laminate materials after exposure to soldering processes. Samples for testing should be sourced from qualified PCB suppliers and can be as provided per IPC-TM-650 Method 5.8.3 (Peel Strength Test Pattern) or per IPC-2221B (Generic Standard on Printed Board Design) specimen C or N.

Peel strength testing should also be performed periodically by PCB suppliers to ensure their "as received" laminate is in compliance with applicable specifications and to ensure the PCB fabrication process does not degrade the laminate and increase the likelihood of pad cratering on the completed assembly.

ACI Technologies offers a variety of analytical instrumentation and techniques for failure analysis and qualification testing of PCB suppliers to ensure compliance to all applicable IPC specifications. X-ray, endoscopy, SEM/EDS (Energy Dispersive X-ray Spectroscopy), and optical microscopy capabilities are available to investigate possible issues and determine root causes. Assistance can also be provided for peel strength testing as well as testing of various adhesives and underfills to mitigate pad cratering. Contact the Helpline at 610.362.1320, via email to helpline@aciusa.org or visit the website at www.aciusa.org for more information.

ACI Technologies, Inc.

ACI Technologies, Inc. 1 International Plaza, Suite 600 Philadelphia, PA 19113 *phone:* 610.362.1200 *web:* www.aciusa.org

Training Center *phone:* 610.362.1295 *email:* registrar@aciusa.org

Helpline *phone:* 610.362.1320 *email:* helpline@aciusa.org