As originally published in the IPC APEX EXPO Conference Proceedings.

Study on Solder Joint Reliability of Fine Pitch CSP

Yong (Hill) Liang, Hank Mao, YongGang Yan, Jindong (King) Lee.
AEG, Flextronics International, Inc.,
Xin Qing Science & Technology Industrial Park, Zhuhai, 519180 China
Email: Hill.Liang@flextronics.com

Abstract

Today's consumer electronic products are characterized by miniaturization, portability and light weight with high performance, especially for 3G mobile products. In the future more fine pitch CSPs (0.4mm) component will be required. However, the product reliability has been a big challenge with the fine pitch CSP. Firstly, the fine pitch CSPs are with smaller solder balls of 0.25mm diameter or even smaller. The small solder ball and pad size do weaken the solder connection and the adhesion of the pad and substrate, thus the pad will peel off easily from the PCB substrate. In addition, miniature solder joints reduce the strength during mechanical vibration, thermal shock, fatigue failure, etc. Secondly, applying sufficient solder paste evenly on the small pad of the CSP is difficult because stencil opening is only 0.25mm or less. This issue can be solved using the high end type of stencil such as Electroforming, which will increase the cost.

For this analysis, we focus on the following areas:

- Reliability performance of different SMD or NSMD pads
- Bigger pad size with higher adhesive strength
- IMC thickness analysis and the results(such as Ni crack or thick IMC)
- Improvement with underfill on CSP
- Improvement of solder paste deposition on pad with vacuum support
- Optimized reflow profile and reliable solder joint relationship

Key words: Fine pitch；CSP, Reliability, IMC, Surface finish, SMD, NSMD, Stencil Opening, Fixture, Underfill.

Introduction

Mobile products are getting smaller, lighter, faster and most importantly have enhanced functionality. More and more fine-pitch (pitch≤0.4mm or 16mils) CSPs are being used in mobile products to meet the requirement. Fine pitch implies small termination, low mechanical strength and low reliability. Mobile phone will experience various surrounding such as hot, cold, moisture, chemical or other exposures. More challenges are impacting the reliability of the mobile phone. The failures due to weak connections have caused huge losses to the mobile phone industries in the past few years. So, higher reliability needs to be considered during the design and manufacturing process. For example, pad design consideration, surface finish selection, process optimization etc.

Most experts in the industry have completed many research activities and acquired significant achievements on the failure and reliability analysis. This paper was completed based on the industry expert information and the authors experiences. It describes how to improve the reliability of fine pitch CSP solder joints based on pad design, surface finish, stencil design, underfill considerations, fixture design and reflow profile consideration. The critical factors must be considered and optimized before the mass production.

Pad Design

Two types of solder lands design are being used for CSP package; they are Solder Mask Defined (SMD) and Non-solder Mask Defined (NSMD) [2]. SMD lands has solder mask overlapping on the copper land (Fig 1); NSMD lands are copper defined as there is solder mask clearance around the copper lands (Fig 2). The two types of lands design have advantages and disadvantages respectively.

Figure 1 – Solder Mask Defined Pad

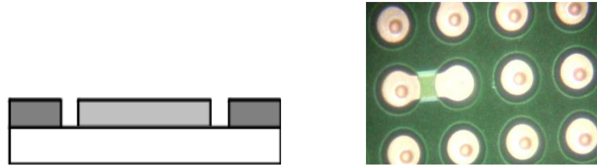


Figure 2 – Non-Solder Mask Defined Pad

SMD lands require larger diameter metal land to achieve the same diameter of solderable land as NSMD because of the overlapped solder mask; the metal land can stick on the substrate more than NSMD; secondly, the solder joint of SMD has a higher standoff and better reliability. But, because the solder cannot wet to the solder mask, it will affect the geometry of the solder joint, with high stress created at the solder mask opening edge which decreases the fatigue life compared to NSMD lands [2] (Fig 3).

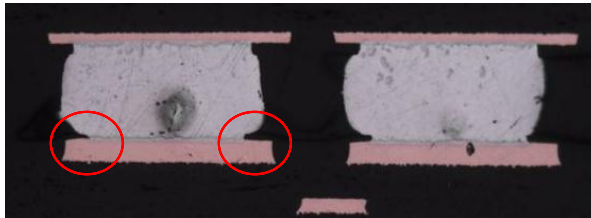


Figure 3 – Cross Section of SMD pad

NSMD require a smaller diameter copper land; the absence of solder mask around the solder land allows the solder to flow around the edges of the land, eliminating any areas of stress concentration [2]. This makes the solder joint wider, potentially giving it longer fatigue life. On the other hand, the wider solder joint will decrease standoff height and possibly decrease the solder joint reliability. Secondly, the smaller copper area adhering to the substrate will cause easy board land peel off during rework [1].

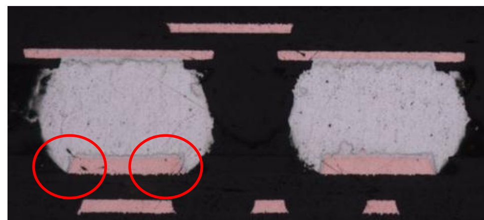
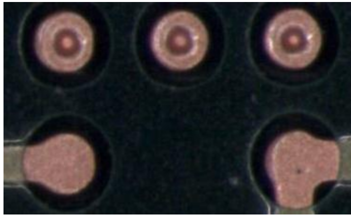


Figure 4 – Cross Section of NSMD

Via in land is generally not recommended. Currently, microvias are becoming more common in CSP land designs. Most of the CSP solder joints will have voids whenever there is a microvia on the pad (Fig 5). It causes a reliability risk to the solder joint.

Studies show that most voids are not a reliability risk in causing a crack, however, they reduce the joint area and in the short term will lead to failure when a crack is propagated [2]. It is recommended to have the via away from the pad (Fig 6).



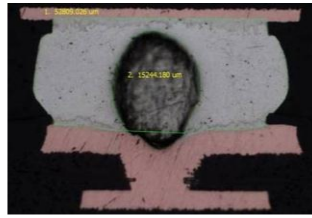


Figure 5 – Via in Pad causing voids

Figure 6 – Via out of Pad

Pad Design Conclusion

1. NSMD design has shown a higher reliability solder joint connection than an equivalent SMD pad.
2. CSP below 0.8mm pitch and a part exposed to high mechanical stress should use SMD, because the pad is held down and helps to prevent pad lifting during mechanical shock or rework.
3. If underfill rework is required, it is recommended to use SMD for a higher rework yield.
4. The vias should be placed out of the pad to avoid voids.

Surface Finish Selection

Surface finish of the PCB land plays an important role in CSP solder joint reliability. The purpose of surface finish is for: solderability protection, a conductive surface for contacts/switches, wire bonding surface, and solder joint interface. Some of the most common surface finishes are Hot Air Solder Leveling (HASL), Organic Surface Protection (OSP), Immersion Tin (I-Sn), Electroless Nickel/Immersion Gold (ENIG), electrolytic nickel/electroplated gold and Immersion Silver (I-Ag).

For HASL (Fig 7**)**, the thickness of the surface finish on the PCB changes by the pad location, the uneven surface is not advisable for fine pitch component assembly.

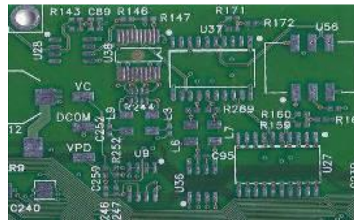


Figure 7 – HASL Surface Finish

For ENIG (Fig 8**)**, a thin layer of immersion gold, 0.05-0.15um, is plated on the top of electroless nickel, the nickel plating thickness is 2.54-8um [1]. It has a planar surface, long shelf life and excellent wettability. But the biggest risk is potential black pad and brittle joint issues for the CSP component [2]. The nickel surfaces under the gold layer tend to be effected during immersion gold plating. Two failure modes were found in recent years. The first failure mode was a non-wetting or de-wetting condition referred to as black pad. The second failure mode was an interfacial fracture that is associated with mechanical stress, because the affected solder joints can not form a robust mechanical bond with the PWB. As a result, the solder joints fail with a relatively small applied force [3], revealing lands with little or no solder left on them. The exposed nickel surface on the land is smooth with an appearance varying from grey to black in color (Fig 9).

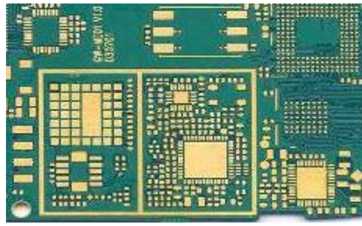


Figure 8 – ENIG Surface Finish

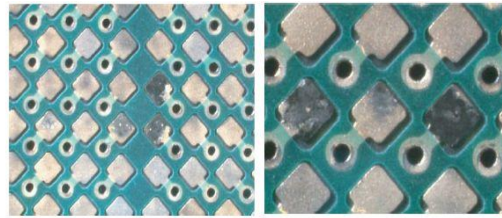


Figure 9 – Black Pad

Brittle joints or brittle failure on the ENIG were induced by weak joints between the IMC and the Ni layers and the brittleness of the IMC itself (Fig 10). It is observed that the brittle failure occurs in the Ni- Cu-Sn IMC or at the interface between the IMC and the Ni layers. Young's modulus data for the IMC indicates relative brittleness of materials (Fig 11); a higher value indicates a more brittle nature of the IMC.

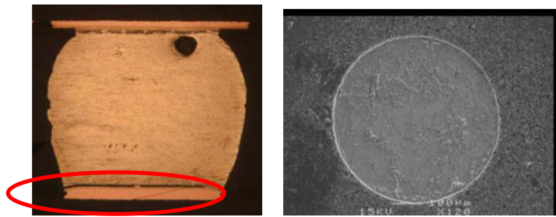


Figure 10 – Brittle Failure on ENIG [4]

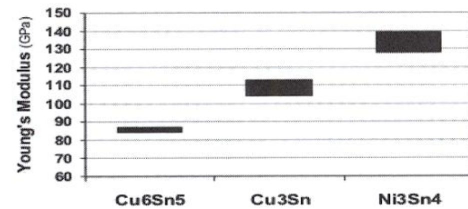


Figure 11 – Young's Modulus of IMC [5]

OSP (Fig 12), OSP is a thin layer, 0.1-0.4um; with the organic coating directly on the copper. OSP has a planar surface which can be used for fine pitch CSP assembly [1]; which is good for mechanical reliability (see Fig 11) and recommended for CSP, BGA and LGA type components. OSP does not have black pad or brittle joint issues, but the IMC growth rate in solder on OSP surface finish is higher than that on ENIG, where excessive growth of IMC potentially promoting brittle failure, weakening the solder joint strength and affect the long term reliability. OSP pads may generate more brittle failure and have much lower solder ball attachment strength after thermal aging (Fig 13).

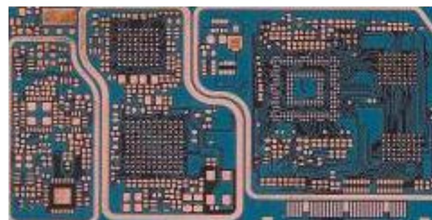
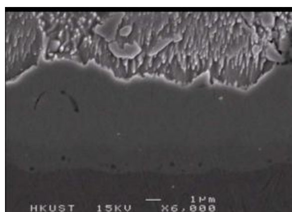


Figure 12 – OSP Surface Finish



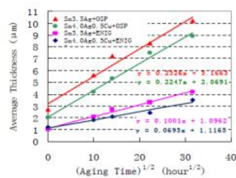


Figure 13 – IMC Growth with Temperature & Time [4]

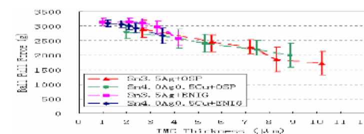


Figure 14 – Bonding Strength Decreasing by IMC Thickness [4]

OSP with selective ENIG (Fig 15) is a selective process that is used to get the best properties from ENIG and OSP. OSP is the surface finish for the pads that need to be soldered; ENIG surface finish on the rest of the PCB pad which will be used for keypad, spring connectors and heat-seal/anisotropic conductive adhesives.

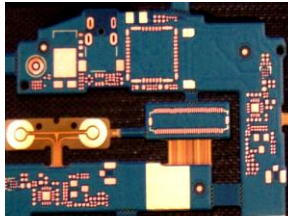


Figure 15 – OSP with Selective ENIG

I-Sn (Fig 16), a thin layer of immersion tin, with 1-1.5um thickness is plated on top of the copper pad, with less brittle CuSn intermetallic layers occurring compared to NiSn intermetallic layers since the solder joint is directly connected to the copper pad.

The planar surface is good for fine pitch CSP assembly and the cost is effective compared to ENIG. But the shelf life and heating cycles for immersion tin board finish is limited. Tin whiskers are another issue that affect the reliability because of residue stress occurring in the coating (Fig 17).

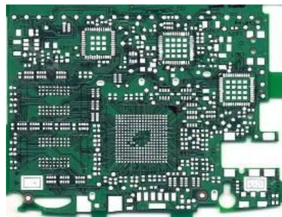


Figure 16 – I-Sn Surface Finish

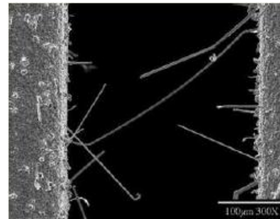


Figure 17 – Tin Whiskers [6]

For Electrolytic Nickel/Gold, a thin layer of electrolytic gold, with 0.15-0.75um thickness, is plated on top of electrolytic nickel, the nickel plating thickness being about 3-5um. This process is more expensive than the ENIG process. The advantage of this process is that it does not have the black pad issue and has a good electro-mechanical connection, but it is difficult to control the gold thickness. A special design has to be considered for meeting the plating process; it is very difficult for HDI PCB with CSP.

I-Ag (Fig 18) is a thin layer of immersion silver, with 0.1-0.4um thickness which is plated on top of the copper pad, and no less brittle intermetallic layers will occur compared with NiSn intermetallics layers since the solder joint is directly connected to the copper pad [1]. The planar surface is good for fine pitch CSP assembly and it is cost effective compare to ENIG. But special handling is required. Plating chemistry, process control and thickness are critical. There are concerns for corrosion, mechanical durability and voids (Fig 19).

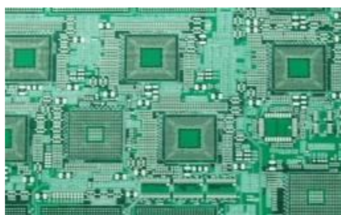
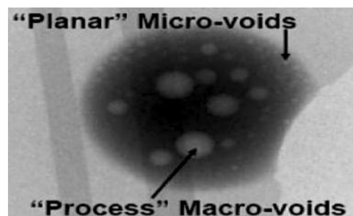


Figure 18 – I-Ag Surface Finish



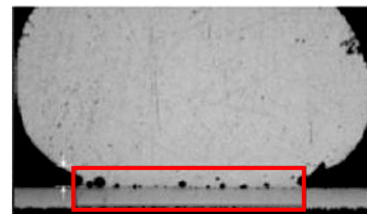


Figure 19 – Void in Solder Joint [7]

UnderFill

Underfill is usually required when the CTE of the chip and the CTE of the organic multilayer board (Fig 20) is inconsistent. The main reason for CSP underfill is to overcome the mechanical stress induced during drop-testing or from keypad pressure. Underfill encapsulants provide an excellent solution for solder joint mechanical stresses, such as shock, drop, and vibration in

hand-held devices (Fig 21). In today's electronics the trend is to apply underfill for CSP especially for the handheld products which might come across mechanical stress due to drop during usage especially in automotive and military electronics. Products are expected to be in good working condition for years in vibration and/or severe shock environments.

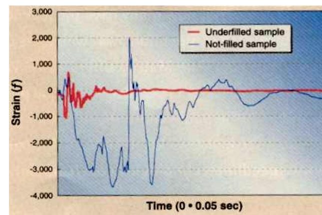


Figure 20 –Strain of motherboard with underfilled and without underfilled sample [10]

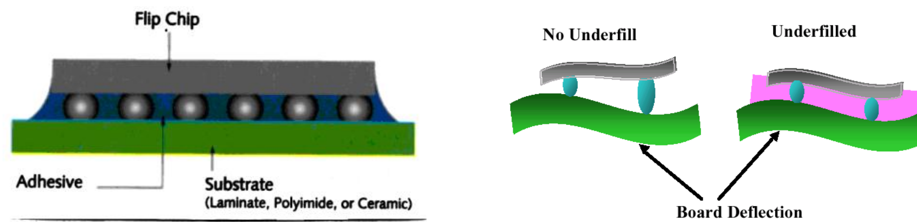


Figure 21 – Protection on Solder Joint

Challenges

Two types of failure mode occur during underfill for the CSP component (Fig 22). First, the coverage issue where the underfill material does not cover the bottom side of the component completely, it happens because of the viscosity and irregular ball layout. Another is the non-cured issue; the underfill material on the center portion of the component is not cured during the reflow process because the flux residue is not compatible with epoxy glue. These issues can be resolved by optimizing the dispensing path and curing profile.

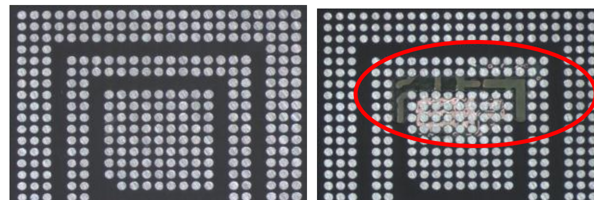


Figure 22 – Good Sample and Failure Sample

Solder Paste Printing

Vacuum support blocks and even solder joints can increase the fatigue life of connections [2]. Even and consistent solder volume deposited on the board pads guarantee a good solder joint. The factors affecting solder paste printing include stencil aperture, solder paste viscosity, printing parameters and PCB supporting tool. Generally, type 4 solder paste is used for fine pitch CSP assembly; the factors related to process can be optimized and should include stencil aperture and support tooling. Vacuum support blocks do improve the performance in getting a better solder paste print volume on the board pads. For the traditional solder paste printing process, support pins or support blocks are used (Fig 23), the PCB warps slightly during stencil separation. This will degrade the paste printing performance and might not produce a good solder paste print. The vacuum support block does help in getting a good solder paste print volume because the vacuum suction does hold the whole PCB on the block more tightly (Fig 24). A DOE has been done for 0.4mm pitch CSP, with the results shows significant differences between the vacuum and non-vacuum support method (Fig 25). Please note, that a flat PCB during solder paste printing will result in even printing.

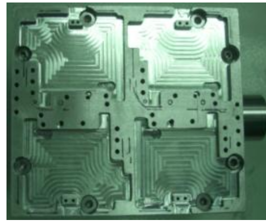


Figure 23 – Vacuum Support Block

Figure 24 – Illustration of Stencil Separation from PCB

A, Support Pin B, Vacuum Support Block

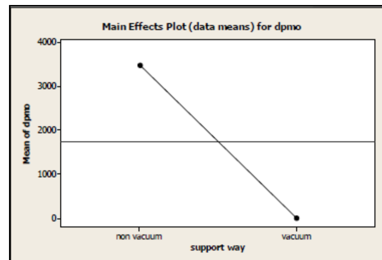


Figure 25 – Comparison on Non-vacuum versus Vacuum Support (8)

Stencil Design

Stencil thickness and aperture size will determine paste volume, which is very critical for fine pitch CSPs.

For 0.4mm pitch CSP, a 0.25mm square aperture with round corners is recommended, although there is no significant difference between square and round type apertures, but the solder paste volume for square opening is much higher than round opening because of the bigger area (Fig 26) with more solder paste deposited on the pad. This will improve the solder joint reliability because of higher standoff height.

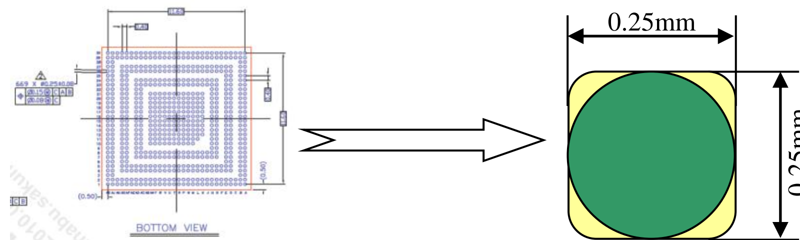


Figure 26 – Comparison of Square and Round Opening

Reflow Profile

Four different regions of the reflow profile are shown: the preheat region, the soak region, the reflow region and the cool down stage (Fig 27).

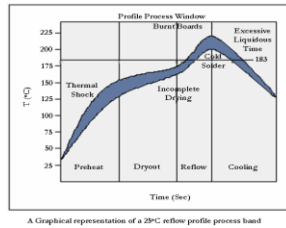


Figure 27 – Reflow Profile

Preheat

The goal is to preheat the entire PCB assembly to a temperature between 120 and 150 °C. The rate of heating is very critical; as it could cause thermal shock to the PCB and / or the components. Activators commence reducing oxidization and some solvents begin evaporating during the preheat. The rate of heat rise is controlled between 1-4°C/second.

Dryout (Soak) phase

The temperature is between 150 – 170°C over a period of 60-120 seconds. The purpose of this phase is to ensure that the solder paste is fully dried before the reflow zone. Activators react to clean components and pad interface surfaces. It also provides thermal stabilization for large and small components to ensure uniform heating before the PCB enters the reflow region.

Reflow

The peak temperature is 30-40°C above 217°C, the liquidus temperature for SnAgCu lead free solder paste. This stage will melt the solder alloy. During the reflow stage, all solderable portions of the PCB assembly should reach peak temperatures. During this stage, it will achieve full liquidus to assure good solder flow (wetting) and fillet formation. For most of the solder paste, wetting time (time above liquidus – TAL) is 30-60 seconds. If the TAL time is too long, the IMC layer will be thicker in the solder joint. Minimizing the duration (TAL) will result in: minimal intermetallic formation (Cu_6Sn_5, Cu_3Sn, etc.); generate a finer grain structure for the solder joint resulting in greater solder joint integrity; minimize exposure of the PCB assembly and components to high temperature.

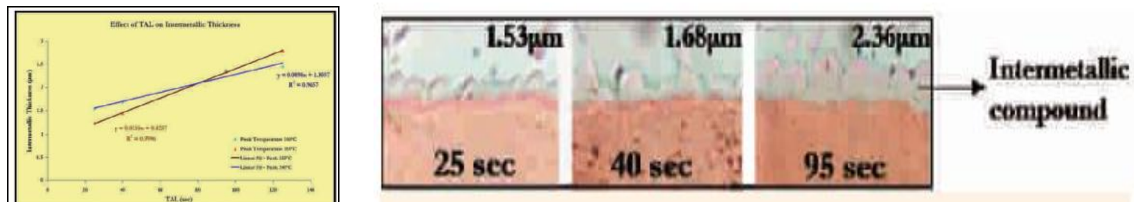


Figure 28 –TAL Effect on IMC Thickness [9]

Cool down

This zone cools the assembly after the reflow. Common cooling rates are controlled between 1-4 °C/second. Excessively slow cool down can create solder joints with large grain size; which could lead to a brittle construction and potentially weak solder connections.

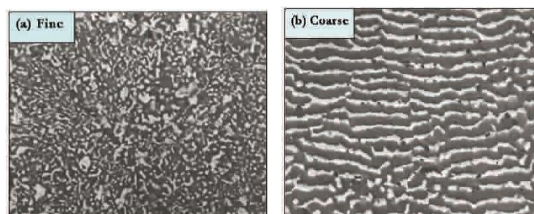


Figure 29 –Cooling Rate Effect on Grain Size [9]

Profile measurement

To acquire a good solder joint, sufficient heat of the component and solder paste is necessary. To avoid moisture and thermo-mechanical stress induced failures on plastic components, the temperature has to be measured on the component body and solder joint, and checked to ensure it does not exceed the specification, with excessive or insufficient heat to be avoided. Hence, thermocouples, which are generally used to measure the temperatures during reflow, should be attached at the solder joint as well as the body of the component.

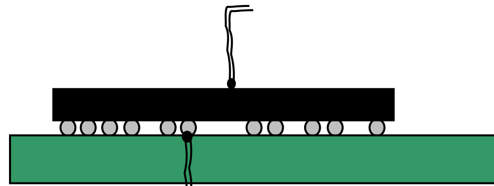


Figure 30 – Thermocouple Location

Summary

Based on the work the following are recommended:

1. NSMD type is recommended for the CSP with 0.8mm pitch and above; SMD type is recommended for CSP with less than 0.8mm pitch with underfill.
2. A via hole away from the solderable pad will reduce the voids occurring in the solder joint which will help to improve reliability.
3. OSP with selective ENIG surface finish is recommended for fine pitch CSP on mobile product to help acquire higher solder joint reliability.
4. Underfill can be used for fine pitch CSP in mobile product to acquire high reliability.
5. Vacuum support blocks are helpful for solder paste printing; it can improve the reliability of the solder joint with an even and more uniform solder paste volume deposited on the board lands.
6. Square with round corner stencil aperture openings are recommended for fine pitch CSP stencil opening designs to acquire more solder paste deposited on the board land pads and to get higher standoff height.
7. Reflow profile is very critical for solder joint reliability, the parameters have to be controlled within specification based on the solder paste supplier and CSP component supplier recommendation.
8. Thermocouples should be attached on the component body and the solder joint of fine pitch CSPs to ensure there is sufficient heat at the solder joint and to avoid excessive heat on the component body.

Acknowledgements

The authors would like to acknowledge and thank the many unnamed contributors to help make this paper possible.

Reference

1. David Geiger and Jonas Sjoberg, "Flextronics Design Guidelines PCBA", 2012.
2. IPC-7095C standard, "Design and Assembly Process Implementation for BGAs", 2013.
3. L. Ma, Discussion for Black Pad and Failure Analysis for Related Example..
4. Fubin Song and S. W. Ricky Lee, "Investigation of IMC Thickness Effect on the Lead-free Solder Ball Attachment Strength: Comparison between Ball Shear Test and Cold Bump Pull Test Results" , 2006 Electronic Components and Technology Conference, 2006.

5. H. Roberts and K. Johal, Lead-free Board Surface Finishes (Chapter 9), “Lead free soldering”, Editor Jasbir Bath, Springer, 2007.
6. Nhat Vo, et al, “NEMI Recommends Standard Test Methods to Assess Propensity for Tin Whisker Growth”.
7. Luhua Xu and John H.L. Pang, Effect of Intermetallic and Kirkendall Void Growth on Board Level Drop Reliability for SnAgCu Lead-free BGA Solder Joint”, smt.pennnet,.
8. Printing Process Improvement DOE in Flextronics, 2008.
9. Manian Ramkumar and Viswam Puligandla, “The Importance of Reflow Hidden Reactions and Profile”, Global SMT & Packaging, Sept. 2008.
10. SMT CAP program Flextronics.