INDIUM CORPORATION®

Indium Corporation Tech Paper

How to Reduce Voiding in Components with Large Pads

Authored by: Wisdom Qu.

Introduction

As the electronics industry moves toward miniaturization and multifunction, smaller components with more functions, such as quad-flat no-leads (QFNs) and LGAs, are increasingly used in a variety of products.

QFNs, being a near chip-scale package, have perimeter contacts on the package bottom providing electrical connections to the PCB. An exposed large area thermal pad, located on the center of the bottom, improves heat transfer out of the chip. QFNs have been increasingly used in the electronics industry, because they: show no visible lead protruding from the bottom; possess a smaller footprint and smaller profile than that of QFPs; and they provide excellent electrical and thermal performance.

To leverage the advantage of the exposed large area thermal pad effectively dissipating heat to a PCB board from the chip, it is necessary to design a corresponding thermal pad, as well as vias at the location of the PCB board where the QFN is mounted. The big thermal pad on the board provides reliable soldering, as well as heat transfer while the vias provide heat dissipating paths. However, during reflow, when the exposed thermal pad on the QFN bottom is soldered onto the corresponding thermal pad on the PCB board, flux outgassing from solder paste on the thermal pad with vias forms bubbles. With the SMT process, difficulty in venting the outgas causes large voids to form. We can only try to find solutions to minimize voiding although it is almost impossible to prevent it from happening.

The land grid array (LGA) package looks very similar to a BGA package. It has flat contacts arranged in array on the bottom side of the package. It is very difficult to control voiding because the contacts are two to three times larger than BGA balls. Moreover, for LGA and QFN packages, no relevant technology standards have ever been developed by the industry and, to a certain extent, plague the electronic processing industry.

This paper uses a series of experiments, including optimization of stencil apertures and reflow thermal profiles, and the use of solder performs, to attempt to find solutions to voiding issues.

Experiment Design

The PCB board used in the experiment is a nickel-gold board with a thickness of 1.6 mm, and the heat dissipating pad on the PCB board has 22 vias (Figure 1). Each of the QFNs 48 flat contacts has a width of 0.28 mm and a pitch of 0.5 mm. The exposed pad on the QFN package is 4.1 × 4.1 mm (Figure 2). All the pads are finished with Sn.

WISDOM QU

Wisdom Qu, Assistant Technical Manager, Eastern China, Indium Corporation. Wisdom provides technical support for Indium Corporation's electronics assembly materials, semiconductor and advanced assembly materials, and epoxy flux products. Wisdom has over nine years of experience in surface-mount technology and has presented at technical conferences and customer sites.She is an SMTA Certified Process Engineer.

email: wqu@indium.com

Full biography: www.indium.com/corporate/bio/

A **QR** (quick response) **code** contains encoded data.

When scanned with a smart phone's camera (via a QR reader application), it will take you to a specific URL or text message.

- *Download article*
- *Share with a friend*

http://indium.us/D194

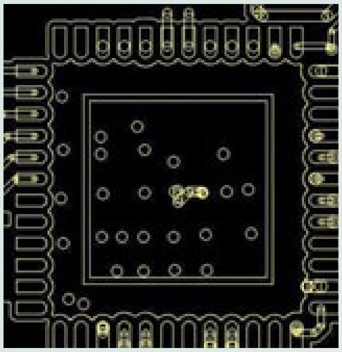


Figure 1. The heat-dissipating pad on the PCB board.

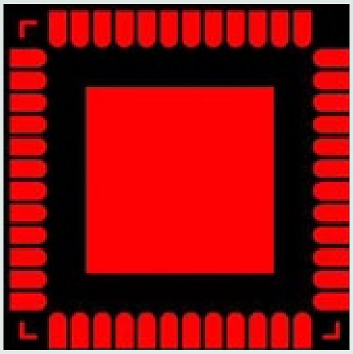
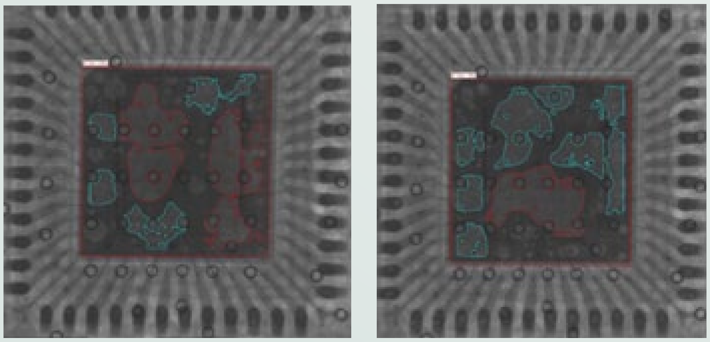


Figure 2. QFN component upside down to show its contact and exposed die attach pad.

Experiment 1:

The Effect of Different Solder Pastes on Voiding

In order to compare the effect of different solder pastes on voiding, two brands of solder pastes are chosen for the experiment. Solder paste A is from a Japanese company and solder paste B is from a U.S. company. Both companies are well-known in the industry. The solder alloy used is SAC305, powder size number 4; the stencil employed is 4 mils in thickness; and the aperture on the stencil is 1:1 to match the exposed thermal pad on the QFN. The experimental result with the two different solder pastes is shown in Figure 3. Large voids appear in QFN solder joints with both solder pastes. This is possibly due to the large size of the thermal pad and the full coverage of solder paste on it. Therefore, the flux outgassing within the solder paste is affected when the solder is at molten state and the air in the vias has no channels to escape. Thus, larger voids are formed and even the best solder paste will not help.



Voiding rate is 35.7% with solder paste A.

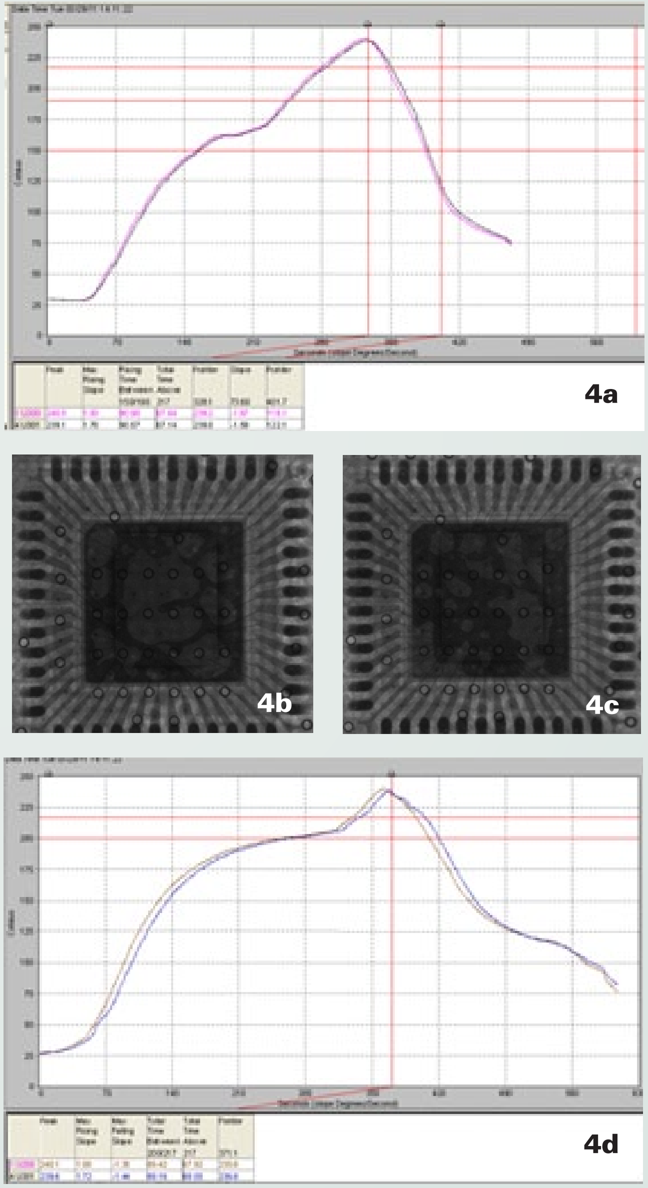
Voiding rate is 37.2% with solder paste B..

Figure 3. The voiding in the QFN solder joint using two different solder pastes.

Experiment 2:

The Effect of Different Reflow Profiles on Voiding

Taking into account that the volatile substance in the flux will produce a large amount of gas during reflow, two typical reflow profiles are chosen in the experiment. One reflow profile is the typical linear reflow profile as shown in Figure 4a, and the other is the saddle-shaped platform reflow profile as shown in Figure 4d. After reflow, it is found that the



Figures 4a, 4b, 4c, and 4d. The voids in the solder joint with two different reflow profiles being used. Figure 4a is the linear profile used in the experiment, and Figure 4b shows the voids. Figure 4d is the saddle-shaped platform profile used in the experiment, and Figure 4c shows the voids.

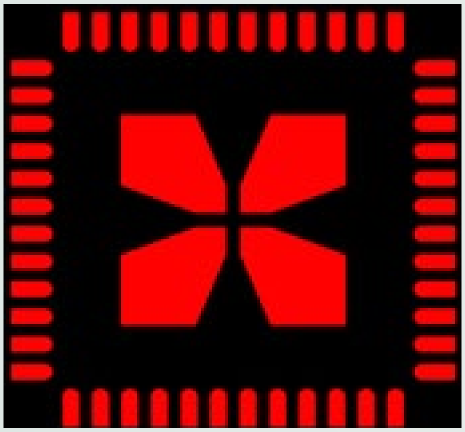
voiding rate is between 35-45% in both cases. However, with the linear thermal profile, as compared to the saddle-shaped platform thermal profile, there are more obvious big, yet less small voids in the solder joint, as shown in Figure 4b. With the saddle-shaped platform thermal profile, there are many small voids, as shown in Figure 4c. The solder joint has no big voids simply because the saddle-shaped platform thermal profile contributes to the full evaporation of volatile substance in the flux as a result of high-temperature baking before the temperature reaches the solder paste melting point. In the linear thermal profile, where the time from warm-up to the melting point is shorter, most of the volatile substances cannot timely evaporate before the temperature reaches the melting point. When the temperature reaches the melting point, the high-surface tension of molten solder prevents the outgas from escaping, resulting in a small number of large voids.

Experiment 3:

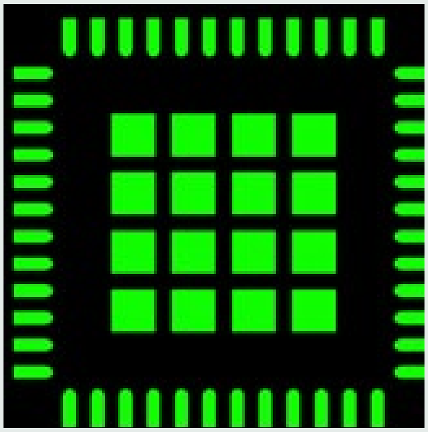
The Effect of Different Stencil Aperture Patterns on Voiding

Due to the large-size thermal pad with vias, the flux outgas within the molten solder has no channels to vent during reflow and is prone to form large voids. A most effective way of venting is by dividing the large pad into multiple small quadrants. Alternatively, modifying stencil aperture so that the solder paste will be printed as multiple small quadrants can also be helpful. In the experiment, stencil aperture is modified and three stencil aperture patterns are used for this purpose, as shown in Figures 5a, 5b, and 5c.

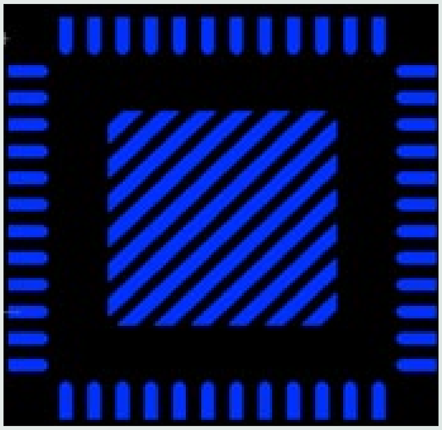
Using three different stencil aperture patterns, the solder joints are examined with an X-ray inspection instrument for voids after reflow (Figure 6). They have similar voiding rates



5a. Stencil aperture pattern 1.



5b. Stencil aperture pattern 2.



5c. Stencil aperture pattern 3.

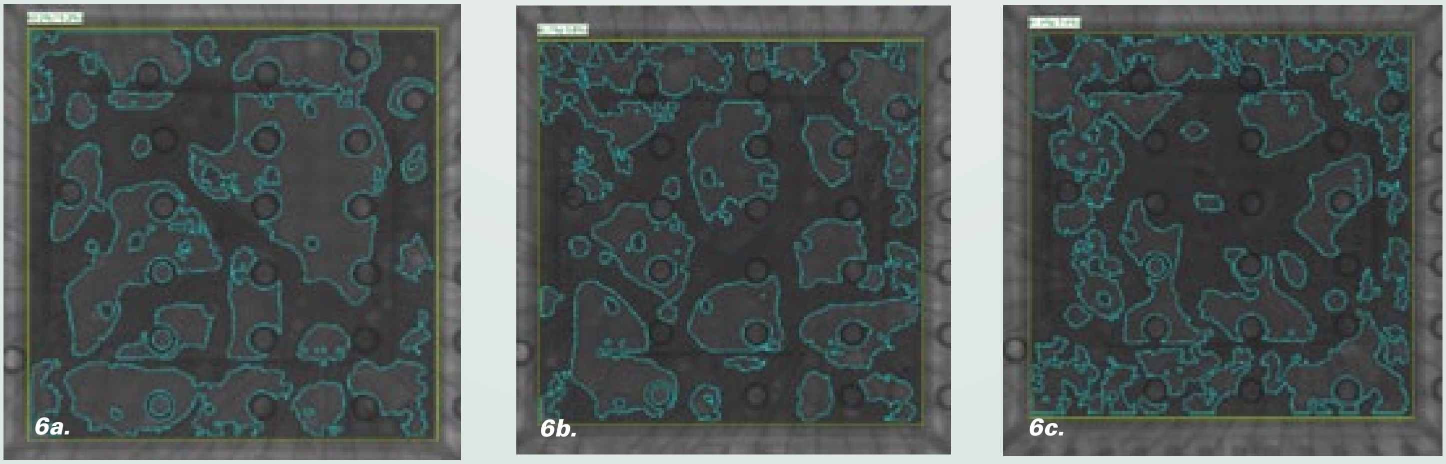


Figure 6. Three different stencil aperture designs are used to increase outgas venting channels. Figure 6a shows the voids with stencil aperture pattern 1, 6b shows the voids with stencil aperture pattern 2, and 6c the voids with stencil aperture pattern 3.

at about 35%. As shown from left to right in Figure 6, the size of voids reduces with the increase in number of the small quadrants of solder paste. With stencil aperture pattern 1, the largest void has an area of 15% of the whole solder joint (Figure 6a). With stencil aperture pattern 3, the largest void has an area of 5% (Figure 6c). With stencil aperture pattern 1, the voids are larger yet fewer. Comparing Figures 6b and 6c with Figure 6a, even though voiding rates with stencil aperture pattern 2 and 3 are similar to that with stencil aperture pattern 1, voids are larger than that with stencil aperture pattern 1. With stencil aperture pattern 3, there are no large voids but many small voids (Figure 6c).

The same three stencil aperture patterns for QFN components are employed for another product, and result in a voiding scenario different from that shown in Figure 6. The voiding rates decrease with the increase of the number of small quadrants of solder paste. This suggests that for some QFN components, modifying stencil aperture design to increase venting channels will help reduce voiding rate. But for some QFN components, especially those with a big pad that has many vias, modifying the stencil aperture pattern will not help reduce voiding.

Experiment 4:

The Effect of Solder with Low Flux on Voiding

Since the voiding mainly depends on the venting of flux outgas, is it possible to reduce voiding by use of solder with low flux? We experimented with solder preforms of SAC305, sized 3.67 x 3.67 x 0.05 mm. It contains 1% flux, compared with the solder pastes used in this experiment that contain 11.5% flux, and the area ratio of the solder preform to the pad is 0.89. With solder paste on the thermal pad being replaced with a solder preform, it is expected to reduce the amount of the flux outgassing, thus reducing voiding. There is no need to modify the stencil aperture design for the perimeter electrical contacts of a QFN component. The only change is the stencil aperture design for the thermal pad. As shown in Figure 7, four small apertures with a diameter of 0.015 inch are designed at the four corners of the thermal pad to hold the solder preform in place.

The reflow profile used in the production line is employed without any change. The solder joints are examined with X-ray inspection equipment for voids in the QFN component solder joint after reflow, and the result is shown in Figure 8. The voiding rates is 3-6%, with single largest void being just about 0.7%.

Experiment 5:

Can Solder Preforms Solve the Voiding Issue of LGA Components?

Since the LGA component has larger pads, it is also difficult in voiding control. Can solder preforms be exploited to reduce voiding in LGA component solder joints? A typical LGA with two different sizes of circular pads is shown in Figure 9. It has 58 pads with a diameter of 2 mm, 76 pads with a diameter of 1.6 mm, and there is a via with each pad.

Optimizing both the reflow profile and the stencil apertures does not help improve voiding in the case of solder paste being used, and the voiding rate is 25- 45 % as shown in Figure 10.

How can solder preforms be exploited for LGA component soldering? An LGA component has round pads with diameters of 2 mm or 1.6 mm (Figure 11a).

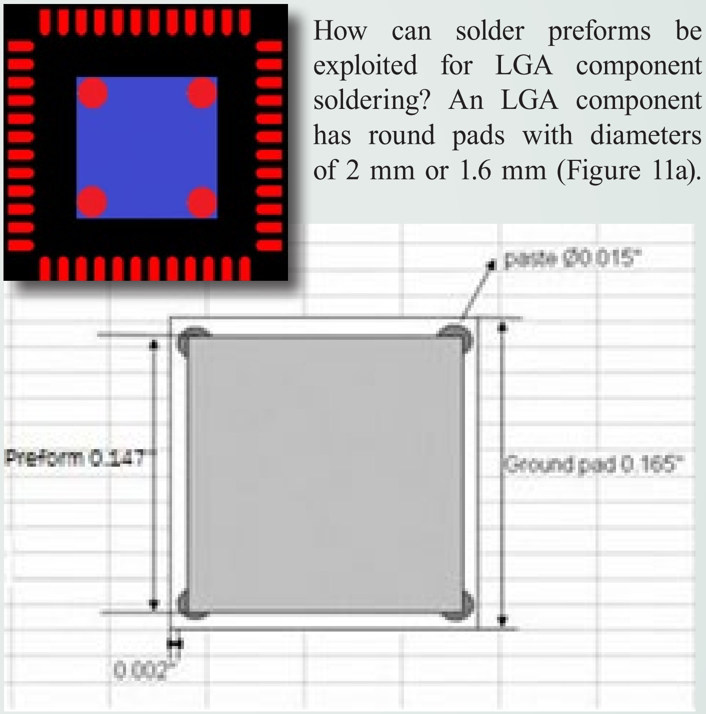


Figure 7. Four small stencil apertures with a diameter of 0.015 inch are designed at the four corners of the thermal pad to hold the perform in place.

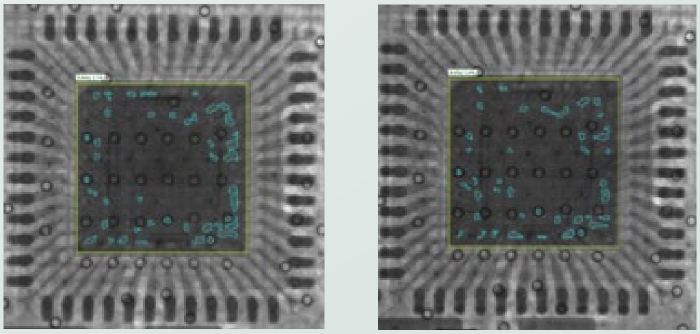


Figure 8. With solder preforms being used to replace solder paste, the voiding rate is 3-6%, with single largest void being just about 0.7%.

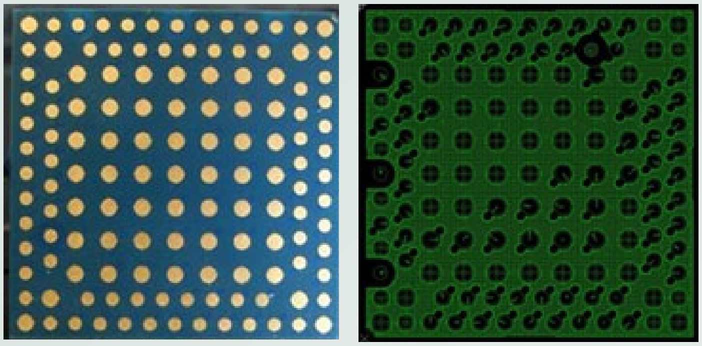


Figure 9. A LGA package has two different sizes of pads.

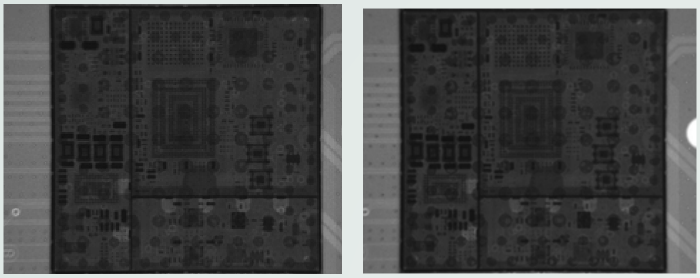
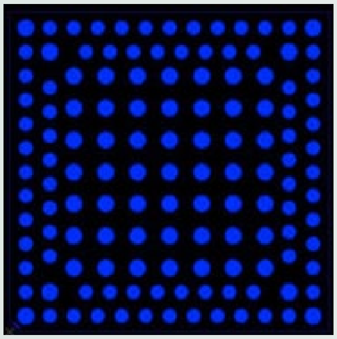
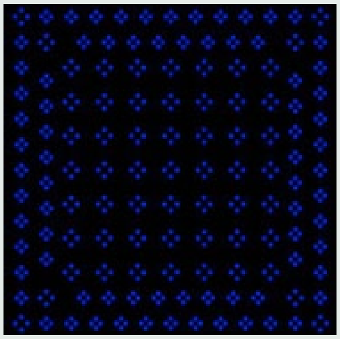


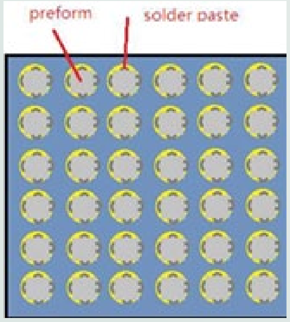
Figure 10. Optimizing the reflow profile and the stencil apertures does not help improve voiding.



11a. The pads of a LGA package.



11b. Four small stencil apertures are designed for each pad.



11c. The solder paste deposited on each pad through the four small stencil apertures will hold the solder preforms in place.

Taking into account the necessity of holding the solder preforms and the LGA component in place, four small stencil apertures instead of one are designed for each pad (Figure 11b). With the solder perform to pad area ratio being 0.8, the solder paste deposited on each pad through the four small stencil apertures will hold both the solder preforms (Figure 11c) as well as the component in place. After reflow, without changing the settings of the reflow profile, the solder joints are examined with X-ray inspection equipment and shows that the voiding rate is 6-14% (Figure 12).

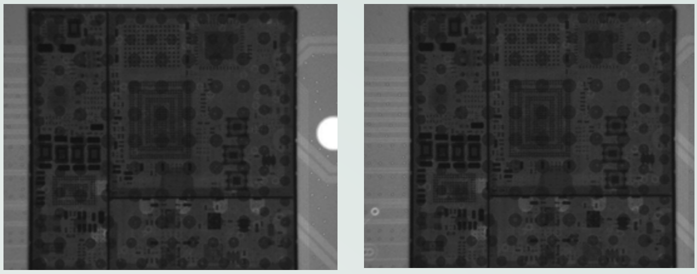


Figure 12. Voiding rate is 6-14 % with solder preforms replacing solder paste for LGA component soldering.

Solder preforms are available in standard shapes such as rectangles, squares, washers and discs, as well as irregular shapes. They come in a variety of packaging options, including tape and reel to facilitate fast accurate SMT placement (Figure 13). When reflow with solder preforms, there is no need to make any changes to reflow profiles. In addition to reducing voiding, solder preforms can be used to provide repeatable, accurate volume of solder when solder paste printing cannot provide sufficient volume of solder alloy.

Figure 13. Solder preforms are available in tape & reel packaging for precise SMT placement.

Summary

For components with large pads, such as QFNs and LGAs, modifying stencil aperture design so that the paste will be printed as multiple small quadrants can help reduce large-size voids because venting channels increases to facilitate the escaping of flux outgas. The flux volatilization temperature of different solder pastes should be taken into consideration so that most volatile substances will evaporate before reflow, helping to reduce large voids. Linear reflow profile helps to reduce the number of voids. When there are big pads with many vias, modifying stencil aperture and reflow profile will not help reduce voiding. Solder preforms will effectively reduce voiding, mainly because the flux content in solder preforms is five times less than the flux contained in solder paste. The flux in solder paste comprises solvent, rosin and thickener, containing large amounts of volatile substance, which leads to the formation of large voids at high temperature. Whereas the main ingredient of the flux in solder preforms is rosin with no solvent or other substances, solder preforms can effectively reduce voiding.

First presented at SMTA East China, May 2011.