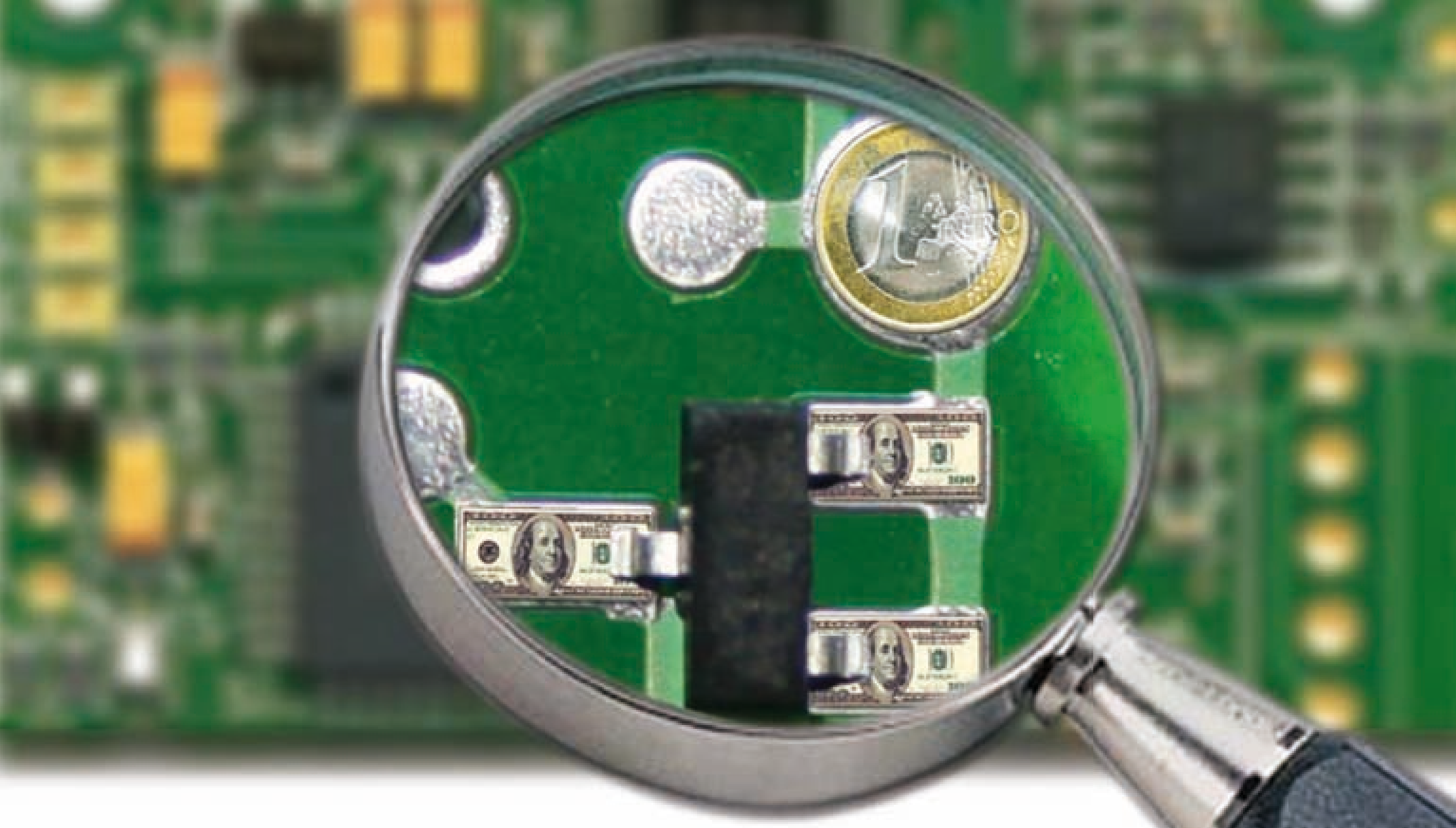


PCB inspection is more important today than ever before!

Industry experts continue to stress the need to inspect hidden solder joints!

Figure 1. The BGA package has not been placed into the paste deposit.

Figure 6. Lead-free joint on a SOIC after reflow soldering in a convection process.

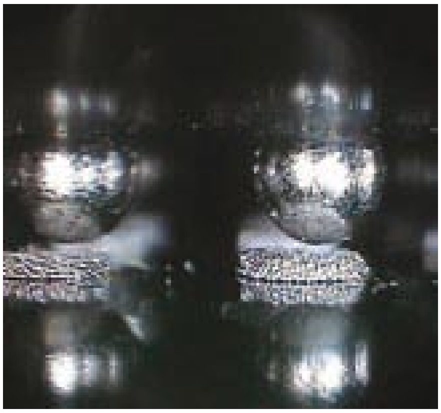


Figure 9. Another BGA package that has not been placed into the paste deposit.

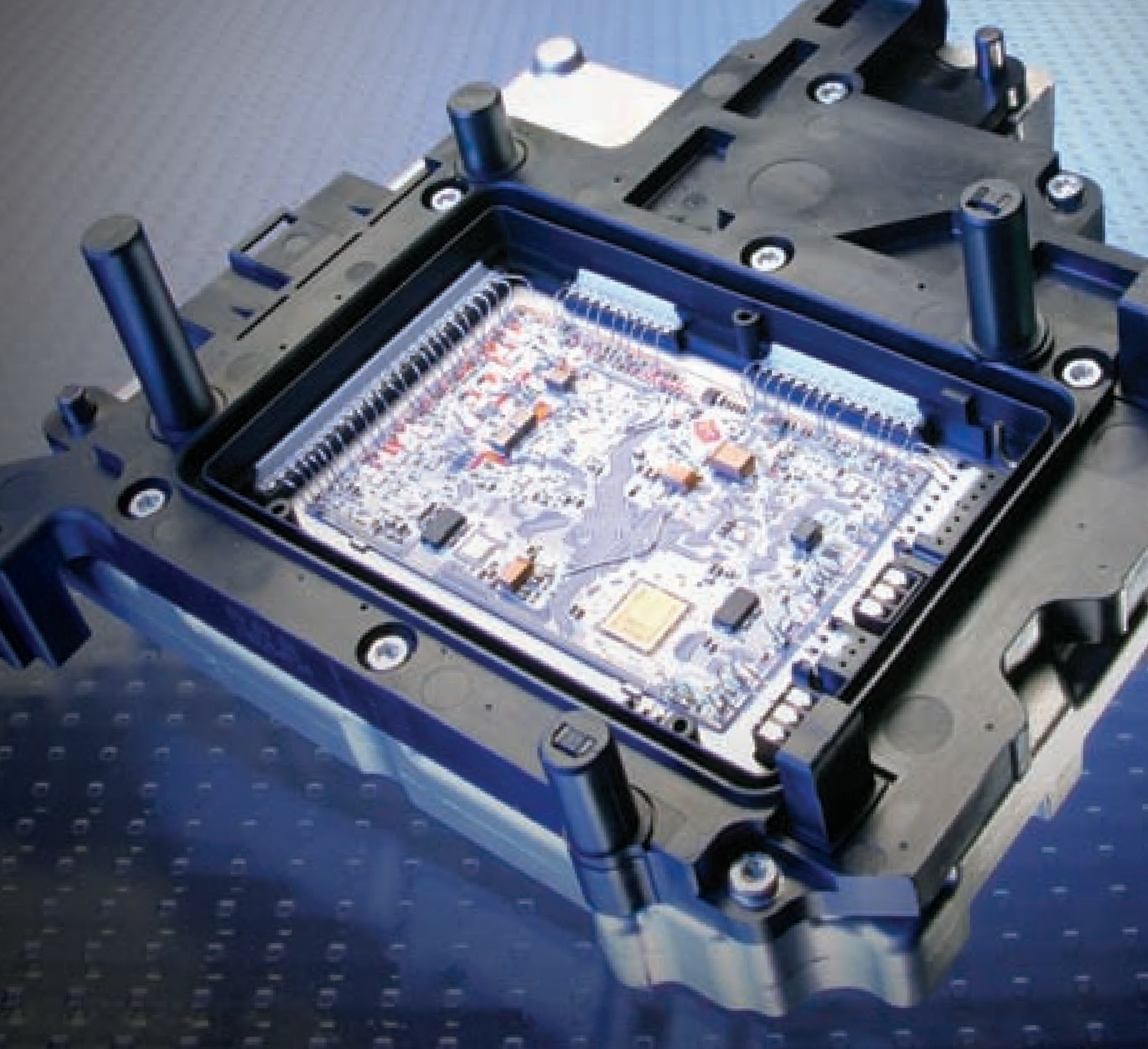
www.globalsmt.net

GLOBAL SMT & PACKAGING

The Global Assembly Journal for SMT and Advanced Packaging Professionals

Volume 8 Number 5 May 2008

ISSN 1474 - 0893



Tetsuro Nishimura
Interview Inside

IMPROVING PRINT PERFORMANCE USING AREA RATIO SENSITIVITY ANALYSIS

WAFER-LEVEL CAVITY PACKAGE WITH VIA-THROUGH-PAD INTERCONNECTS

NANOTECHNOLOGY AND MATHEMATICAL METHODS FOR HIGH-PERFORMANCE THERMAL INTERFACE MATERIALS

TOOLS & METHODS FOR LEAN PRODUCTION MANAGEMENT IN EA

NEW PRODUCTS

INDUSTRY NEWS

INTERNATIONAL DIARY

Bob Willis

Ball grid array & lead-free assembly defects, part 1

Global SMT & Packaging and Mesago Messe will once again be running the "Process Advice and Defect Clinic" on stand 205, hall 9 at SMT Nuremberg 3-5th June where visitors can bring process defects for discussion and optical and x-ray inspection. It's an ideal opportunity to get free advice.

Over the last few years I have been asked to examine many different ball grid array (BGA) defects from assembly and on field failures here are a selection of the defects and some of the popular causes. We also feature some other lead-free defects in this two-part column leading up to the Process Advice & Defect Clinic in Nuremberg.

Figure 1. The BGA package has not been placed into the paste deposit.

Poor Placement Force Control

In *Figure 1*, the BGA package has not been placed into the paste deposit; the ball terminations are sitting proud of the surface of the paste. This may be due to uneven paste deposits, a warped BGA or a random chip component under the BGA, which does happen from time to time.

There is also some evidence of print misalignment on the board. Provided the ball was placed into the paste, and at this pitch, a satisfactory joint should be achieved during reflow. Unfortunately the lifted termination may not allow this to happen.

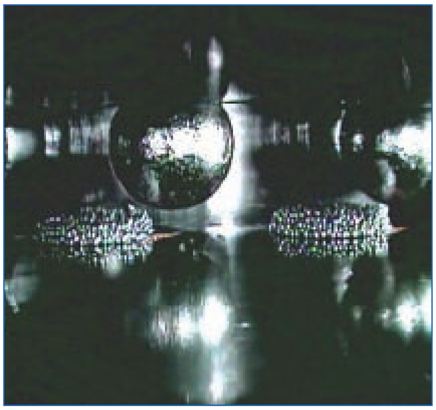


Figure 2. Solder paste despot is not correctly aligned with the pads.

Misalignment of solder paste deposit

Figure 2 clearly shows that the solder paste deposit is not correctly aligned with the pads on the surface of the board. The relative placement position of the BGA is within the limits of reflow to re-centre the part. However, correction should be made for placement error. In this situation, with error on placement and paste, the course-pitch BGA will probably still solder successfully.

Poor alignment of the printed image is fundamental; even with manual or semi automatic printing the operator should have seen the printing error. Error can occur with fine pitch design on automatic printers when the tolerance of the board can vary. In the future, with lead-free, we may see errors on printing between side one and side two of the board on small packages like 0201, due to thermal expansion and contraction of the laminate.

Figure 3. Contamination is not desirable in any production process.

Foreign debris

Any foreign debris on the printed board prior to printing will tend to be held on the surface of the board. If boards are left after printing, airborne contamination is often retained by the tacky paste deposits. BGA devices are sometimes found with debris on the balls or between the terminations when supplied. In most cases this is not conductive and even if it reflows in the joint it is burnt off.

It is certainly not desirable in any production process. Boards should be provided clean in suitable packaging, pasted boards should not be left to gather contamination, and contamination on BGAs should be highlighted to the supplier.

Figure 4. Chip capacitor with a tin termination soldered with tin/silver/copper lead-free paste.

Satisfactory lead-free joint

In *Figure 4*, a satisfactory joint has been produced, with the wetting line of the solder clearly visible around the joint. This is common with any termination or PCB finish that does not become liquidus during normal reflow soldering.

The board was reflowed with a peak temperature of 140°C and a temperature differential of 8°C with a seven-zone oven in air. It is important to remember when discussing reflow parameters to check on the oven in question as often engineers talk about the number of zones but fail to state the length of each zone.

Figure 5. Lead-free joint on a gull wing lead with a tin finish.

Insufficient solder fillet

The solder joint shown in *Figure 5* is outside the requirements of IPC 610 level 3 with very low heel and toe fillets. There is evidence of wetting all around the lead to pad interface.

There is a distinct line between the solder tin/silver/copper and the lead plating but this is common with lead-free alloys. The pull force required to remove this 0.020” lead from the pad would be in the order of 600-800 g. As an example, some companies have a minimum of 450 g for tin/lead products.

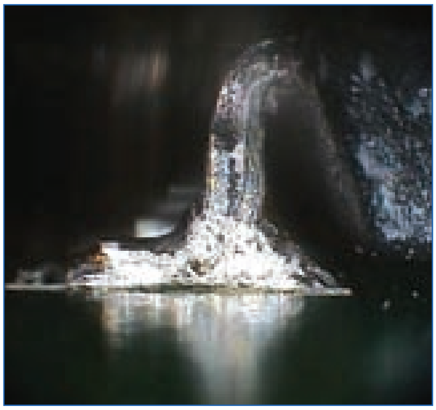


Figure 6. Lead-free joint on a SOIC after reflow soldering in a convection process.

Satisfactory gull wing joint

The joint in *Figure 6* would meet the size and shape joint requirements for IPC610 level 3. Some inspectors may consider the surface of the joint unacceptable but this slightly rough surface is not untypical of lead-free joints.

Figure 7. A MELF that has been reflow soldered using tin/silver/copper paste in a convection system.

Satisfactory lead-free joint on MELF termination

Reflow joints on MELF terminations often do not have the volume of solder to meet the minimum inspection criteria; however, they are seldom rejected. The MELF in *Figure 7* has been reflow soldered using tin/silver/copper paste in a convection system. A satisfactory joint has been produced and the wetting line can be seen along the width of the termination.

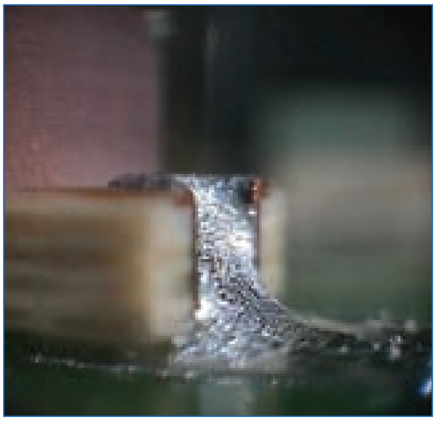


Figure 8. Example of a castellation termination on a lead-free light emitting diode (LED) soldered with tin/silver/copper paste.

Satisfactory castellation termination

The wetting to the pad and the surface of the termination in *Figure 8* is good and exceeds the soldering standards of the IPC610.

The surface of the joint looks rough, but this is not untypical of some joints as they solidify on cooling and may be expected. During shear force measurements the LED exceeded the push force requirement, higher than tin/lead terminations of the same size.

Figure 9. Another BGA package that has not been placed into the paste deposit.

Poor placement force control

Like the BGA package in *Figure 1*, the BGA package in *Figure 9* has not been placed into the paste deposit; the ball terminations are sitting proud of the surface of the paste. This may be due to uneven paste deposit, a warped BGA or a random chip component under the BGA, which does happen from time to time.

Many people do not inspect boards after placement, those that do often use automatic systems that would not detect

Continued on page 63

Continued from page 6

this type of error which may lead to an open connection after reflow. Random sample inspection should be at the heart of any good robust process control program.

Figure 10. These two BGA joints show reduced wetting.

BGA pad joint wetting

Two BGA joints in *Figure 10* show reduced wetting on an OSP board surface; you can easily see the edge of the pads after reflow. This device had been operating successfully for three years and was in fact mechanically sound.

The degree of wetting on some surfaces can affect all components and this example could be rejected simply due to the way it is viewed. This is probably an example where optical inspection would reject the soldering and x-ray accepts it. If this was found in current production, the quality of the PCB should be investigated to see if it is a batch solderability issue or a board that has been processed incorrectly.

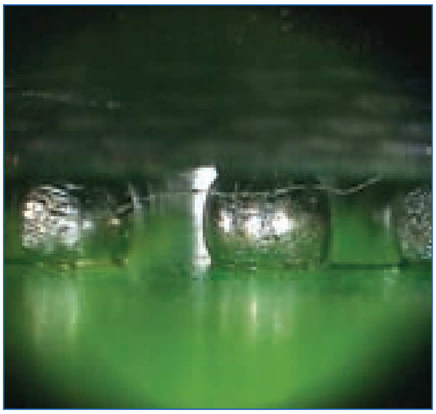


Figure 11. Foreign debris shown on a board.

More debris

Any foreign debris, hair or fibre on the printed board prior to printing will tend to be held on the surface of the board. It is not uncommon today for some sites to clean boards through two sticky rollers prior to printing. If boards are left after printing airborne contamination is often retained by the tacky paste deposits. BGA devices are sometimes found with debris on the balls or between the terminations when supplied. In most cases this is not conductive and even if it reflows in the joint it is burnt off.

There is no real criteria for this type of contamination but it should be considered a process indicator requiring process improvement.

It's important to remember that if you are going to conduct failure analysis some of the simple techniques should always be considered before more advanced evaluation techniques. Details on Failure Analysis Workshops for printed circuit boards and components run at ITRI are available at *www.ASKbobwillis.com/faworkshops.pdf*

Bob Willis is a process engineering consultant, well known in the industry as a provider of theory and hands on training world wide. Bob has also produced a comprehensive range of interactive training CD-ROM and quality control wall charts on electronic manufacture including lead-free. He may be contacted at www.ASKbobwillis.com where you have the opportunity for on-line consultancy.

Bob Willis

Ball grid array & lead-free assembly defects, part 2

Global SMT & Packaging and Mesago Messe will once again be running the “Process Advice and Defect Clinic” on stand 205, hall 9 at SMT Nuremberg 3-5th June where visitors can bring process defects for discussion and optical and x-ray inspection. It’s an ideal opportunity to get free advice.

Over the last few years I have been asked to examine many different ball grid array (BGA) defects from assembly and on field failures here are a selection of the defects and some of the popular causes. We also feature some other lead-free defects in this two-part column leading up to the Process Advice & Defect Clinic in Nuremberg.

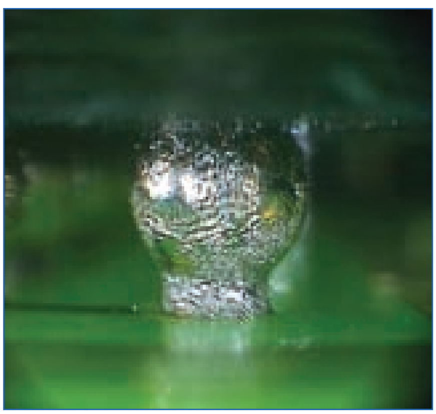


Figure 1. The ball in this figure has not started to reflow, indicating an incorrect profile.

Incomplete reflow of lead-free paste

The BGA in *Figure 1* has lead-free terminations and is being reflowed with lead-free solder paste tin/silver/copper. The paste has started to reflow, but it has not completely reflowed. The ball has not started to reflow, so clearly the profile used on this assembly was incorrect. Generally it would be the peak temperature or the time at peak temperature that was incorrect on this example. It is feasible that too long a period at high pre-heat prior to reflow could have exhausted the flux in the paste.

Certainly this would not be considered an acceptable joint.

Figure 2. Close-up view of a lead-free BGA and non-reflowed solder paste.

Clearly in *Figure 2*, the assembly has not achieved the required temperature during reflow. The size of the paste deposit for a termination of this size should also be questioned—either the incorrect stencil aperture has been used or the paste release from the stencil needs to be investigated. Either of these issues could lead to open circuits if warping of the device takes place.

Lead-free J lead joints

The two joints in *Figure 3* have soldered on the front face of the pins, which would normally be easily seen during inspection at low magnification, but the joints have not formed under the body of the part.

This is probably due to solderability problems with the printed board and are not related to the lead-free process.

Ceramic ball grid array joint

Figure 4 is fairly typical of a tin/lead joint formed on CBGA terminations. As the ball is a high temperature 90% lead/10% tin termination it does not become liquid during normal reflow. When the solder paste reflows with a peak temperature of between 210-220°C, the tin/lead paste will wet the pad and climb up the ball termination. This means there will always be a demarcation line between the solder and the ball. The line around the ball at the top of the joint is how the ball is initially soldered to the package.

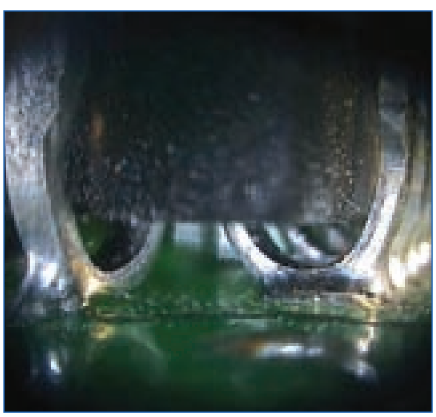


Figure 3. The joints have not formed under the body of this part.

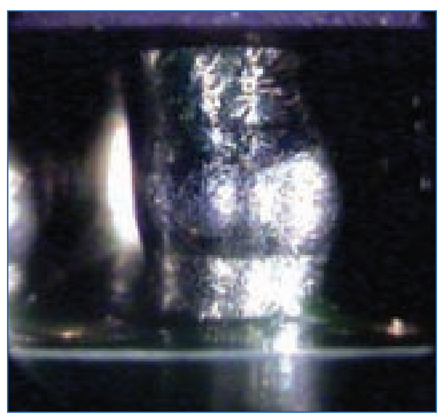


Figure 4. Typical tin/lead joint formed on CBGA terminations.

If you do a microsection of this joint you would still see the perfect ball sitting between the two joints.

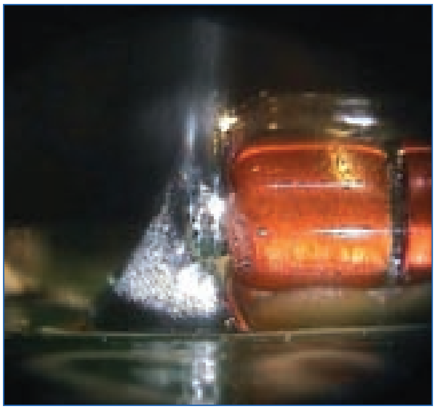


Figure 5. A perfectly satisfactory lead-free joint.

MELF Lead-free wave solder joint

The side view of a lead-free solder joint shown in *Figure 5* was produced with tin/silver/copper alloy and is a perfectly satisfactory lead-free joint. The solder surface shows evidence of the lines often seen in lead-free joints formed during solidification of the alloy and is a surface effect. This example was produced on a double wave with a solder temperature of 260°C and water-based, no-clean flux technology.

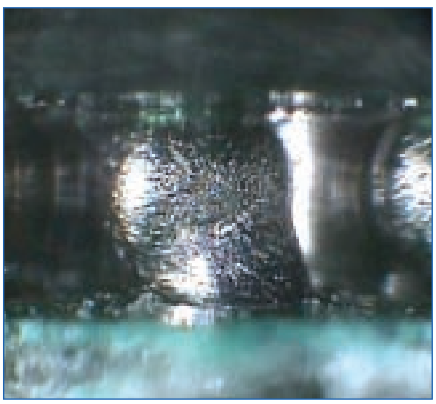


Figure 6. A casualty of the transition from a tin/lead process.

Lead-free BGA with tin/lead solder

Figure 6 is fairly typical of what can be seen during the transition from a tin/lead process and lead-free. More and more component suppliers are changing to lead-free terminations, which do not reflow at normal tin/lead solder paste process temperatures.

What is seen in this example is the result of reflowing tin/lead paste at 210-220°C which is normal. If the ball had been tin/lead, the termination would have reflowed and the solder and the ball would have coalesced together to form a perfect joint. In this case the reliability of the joint may be perfect, as the solder has wetted the ball; however, there will be a lot of debate on this issue for some time.

The key point is to be informed by suppliers when and on what batches changes to alloy will occur.

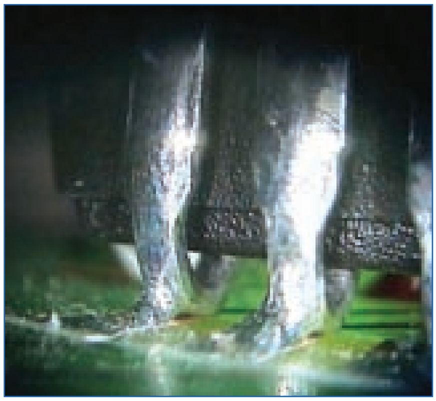


Figure 7. Joints that have not formed under the body of the part.

Lead-free J lead joints

The joints in *Figure 7* have soldered on the front face of the J leads. This would normally be easily seen during inspection at low magnification. The joints have not formed under the body of the part. In this case the base pad finish was copper OSP treated. This problem is probably due to OSP solderability problems with the printed board and not related to the lead-free process. These joints would be outside of the criteria for IPC 610.

Figure 8. These lead-free joints have failed to wet correctly.

Gull wing leads

Figure 8 shows an example of lead-free joints on gull wing terminations that have failed to wet correctly. The board had been wave soldered with tin/copper/nickel, and the OSP pad had not wetted fully, probably due to some problems with the surface of the copper. This would be outside of the requirements of IPC 610.

Figure 9. The balls in this part are plastic with a metallised solderable coating.

Correct BGA alignment

Clearly the solder paste has not reflowed in *Figure 9*, but the position of the balls in the paste is correct. The board assembly was being inspected prior to reflow for position of the terminations. This device is different than most as the balls in this example do not collapse during reflow and help improve joint reliability. The difference with this part is the balls are plastic with a metallised solderable coating.

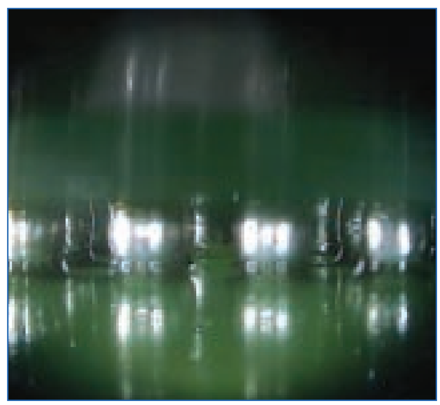


Figure 10. Solder short found during regular scanning of BGA terminations.

BGA solder short

Finding shorts with x-ray is easy, but it can be just as easy with optical inspection, as *Figure 10* shows, by scanning along one side and then the second side. If the light from the opposite side of the BGA is not visible along the two rows of balls, a short must be present and can be examined by focusing along the rows of balls.

Solder shorts are most often caused by warping of the BGA substrate during rework or initial reflow. The substrate warps at the edges or in the centre and squashes the solder balls.

Figure 11. An incorrect profile resulted in partial reflow on the pin.

Incomplete reflow of lead-free paste

Figure 11 shows an example of pin-in-hole (PIH) reflow of a through-hole connector with lead-free solder paste. The profile for the area of the board was not correct, resulting in partial reflow on the pin. The solder paste has been pushed out along and over the pin tip; it has started to reflow. Push out of paste is common with the process, but the time in a liquid state needs to be long enough for the solder to completely reflow, wet the pin and plated through-hole, and then coalesce back into the hole to form the through-hole fillet. If it does not, it gives a good coating on the pin but little value to the joint.

Through-hole parts are normally the most thermally demanding parts on the board and need to be correctly profiled particularly for lead-free assembly.

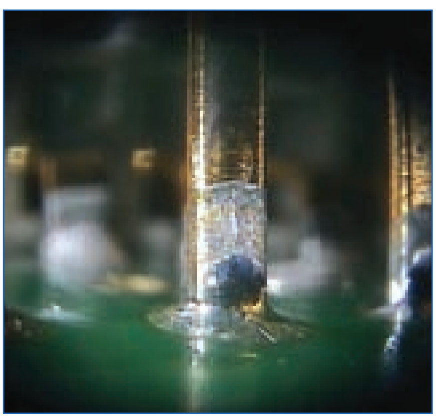


Figure 12. This though-hole connector has been satisfactorily soldered.

Modifying the temperature profile provided a perfect joint with 100% fill of the plated through hole, easily meeting IPC610 requirements.

Lead-free through hole reflow

Figure 12 shows a satisfactory topside view of a through-hole connector reflow-soldered using tin/silver/copper paste. The joints all had a 100% fill and positive fillets on both sides of the board assembly. The Harting connector pins are gold flash over nickel, provided as the company's lead-free option.

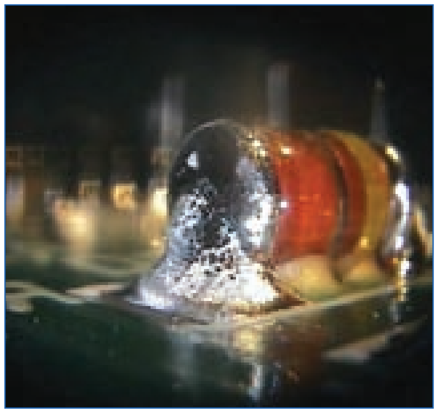


Figure 13. The lines on this joint are merely a surface effect.

MELF lead-free wave solder joints

The lead-free solder joint in *Figure 13* was produced with tin/silver/copper alloy and is a perfectly satisfactory lead-free joint. The solder surface shows evidence of the lines often seen in lead-free joint formed during solidification of the alloy and are a surface effect. This example was produced on a double wave with a solder temperature of 260°C and a water-based, no-clean flux.

Figure 14. Although this joint is bulbous, it still meets inspection criteria.

This side view of a lead-free solder joint in *Figure 14* was produced with tin/silver/copper alloy and is a perfectly satisfactory lead-free joint. The joint is bulbous in its appearance but still meets the inspection criteria. The solder surface shows evidence of the lines often seen in lead-free joints formed during solidification of the alloy and is a surface effect. This example was produced on a double wave with a solder temperature of 260°C and water-based, no-clean flux technology.

It is not uncommon for lead-free joints to be fuller in their appearance, due to the poor drainage of most alternative alloys.

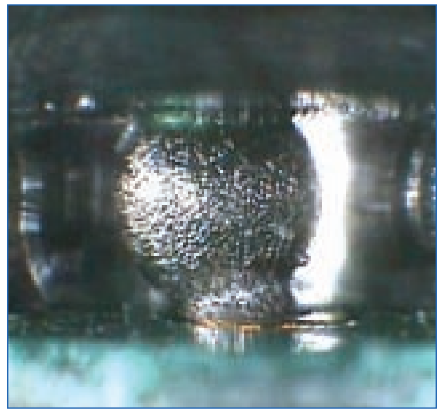


Figure 15. The paste has reflowed and wet the pad and the ball termination, but the ball has not reflowed.

Lead-free BGA joint

Figure 15 shows a lead-free termination reflowed with tin/lead paste. The paste has reflowed and wet the pad and the ball termination; the ball has not reflowed. There are a number of issues here: small volume of paste, limited wetting on the pad and lead-free termination with tin/lead paste.

The limited wetting to the pad may be as a result of the small volume of paste available rather than a specific issue with solderability. The process needs to be reviewed for print quality and solderability of the pad finishes. BGAs being supplied as lead-free without the supplier informing the vendor must be avoided.

It's important to remember that if you are going to conduct failure analysis some of the simple techniques should always be considered before more advanced evaluation techniques. Details on Failure Analysis Workshops for printed circuit boards and components run at ITRI are available at www.ASKbobwillis.com/faworkshops.pdf

Bob Willis is a process engineering consultant, well known in the industry as a provider of theory and hands on training world wide. Bob has also produced a comprehensive range of interactive training CD-ROM and quality control wall charts on electronic manufacture including lead-free. He may be contacted at www.ASKbo bwillis.com where you have the opportunity for on-line consultancy.

ERSASCOPE 1 – *The Original*

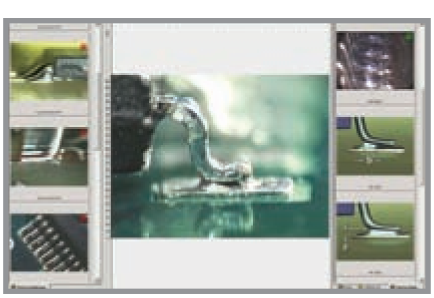
6.990,– €

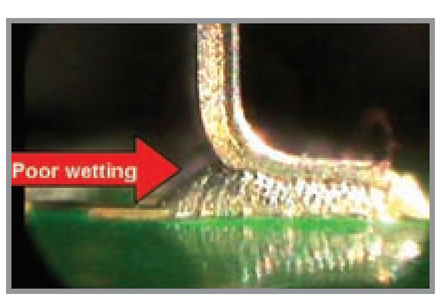
* Package price complete; ex works excl. VAT

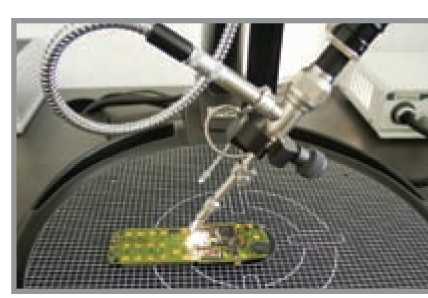
ERSASCOPE 1 – *The Original* offers the award winning, original ERSASCOPE inspection capability at the lowest price possible. The core component of the system, the ERSASCOPE optic, is the original, patented optic. A completely new USB 2.0 camera uses CMOS technology offering optimal light sensitivity and resolution. The new halogen light source supplies optimal light to both the ERSASCOPE 1 optic as well as the flexible gooseneck now included standard. The multi-functional stand and X-Y table offer a total of 7 axes of movement for the scope. The user friendly ImageDoc Basic software package is included standard and offers the following features: Live Image/Still picture; Reference Picture Groups/ Image Database; Limited ERSA/Fraunhofer Database with Soldering Problems/Solutions; Image processing/label-ling; Measurement & Automatic Measure Control function as well as basic Reporting & Emailing functions.

System includes:

- ERSASCOPE optical unit
- TV-adapter
- Digital USB Camera
- Halogen light source
- ERSASCOPE stand
- X-Y-Inspection table
- ImageDoc Basic 1.3 Software







Kurtz ELECTRONICS

ERSA GmbH • Leonhard-Karl-Str. 24 • 97877 Wertheim, Germany
Tel.: +49 (9342) 800-0 • Fax: +49 (9342) 800-100 • info@ersa.de • www.ersa.de