# LEAD-FREE DEFECT GUIDE 2

*A Guiding Influence in the Electronics Industry*

SMART GROUP

Published June 2008.

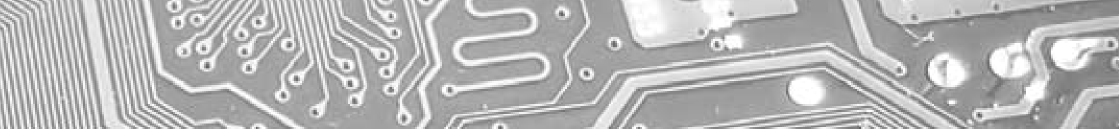
# customer care

Artetch are not only highly focused on customer care and product quality but are also committed to product and process development within the electronics industry. They support a number of industry bodies including The SMART Group and The National Physical Laboratory (NPL), by either providing technical input or supplying evaluation test vehicles to be used within their Research and Development programmes.

The knowledge gained from this involvement aids Artetch's customer base and the industry as a whole. Much of the knowledge being gathered on Lead Free processing has been done with the support of Artetch.

artetch circuits Ltd.


Artetch Circuits Limited<br>
Riverside Industrial Estate<br>
Littlehampton<br>
West Sussex<br>
BN17 5DF<br>
+44 (0)1903 725365<br>
sales@artetchcircuits.co.uk



# Welcome to the SMART Group Lead-Free Defect Guide2

**We hope that you find this second guide a useful and informative reference for recognising and solving problems within your manufacturing and design process.**

The SMART Group has been at the heart of disseminating information regarding the diverse research and development needed to implement the change to lead-free soldering. It is due to the knowledge and feedback we have built up, through seminars, the European LEADOUT project, correspondence on our unique Email Forum *smart-e-link*, that we have been able to confidently put this defect guide together. We would like to thank all of those SMART Group members and speakers at events who have given freely of their time and experience. We hope you find this a useful and informative reference guide.

As we in the industry have seen, the defects encountered during and after implementation of lead-free soldering, have brought us a variety of new (and in some cases old) challenges. This lead-free guide is a compilation of defects designed to help managers, designers, engineers and operatives to solve these everyday problems and to let them troubleshoot effectively and efficiently.

Amongst all of this change there has also been reassuring constants, good manufacturing still requires intelligent designs, a capable process, reliable manufacturing controls and the people to design, implement and drive them through production with minimum disruption.

We hope you find this a useful guide and that it helps you achieve your objective.

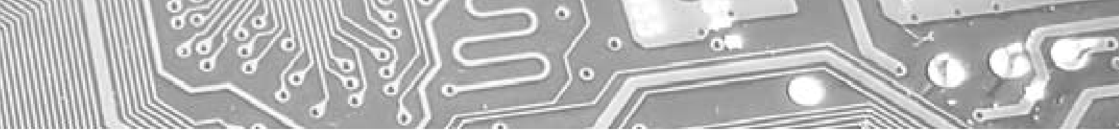
**Naim Kapadia B Eng DMS IET**
SMART Group Technical Committee Member
Manufacturing Engineering Manager, JJS Electronics UK

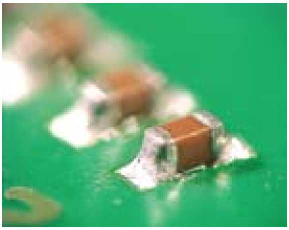
# Introduction

Before we look at some of the common and less common lead-free process defects let's first provide readers with examples of lead-free joints that most engineers would accept as being typical of joints found in industry. The examples have all been taken from mainstream manufacturing processes like reflow, wave and selective soldering.

During the early introduction of lead-free there was no inspection criteria, hence SMART Group were one of the first organisations to make criteria available at hands on workshops, seminars and through its web site. In recent times the IPC 610D "Acceptability of Electronic Assemblies" has been updated and is the first version of the standard to provide lead-free criteria and examples of typical process faults or indicators. It is inevitable that the document will be continually updated to include more lead-free examples in the future.

The following are all typical lead-free joints that would be considered satisfactory by most engineers working in manufacture today. We have included the alloy, printed board surface finish, process and the type of termination for easy reference.



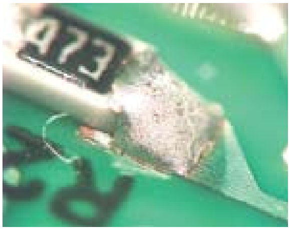


**Chip Component (Reflow)**
Solder: Tin/Silver/Copper (SnAgCu)
PCB: Gold over Nickel

**Gull Wing Termination (Reflow)**
Solder: Tin/Silver/Copper
PCB: Silver

**Through Hole Termination (Selective Solder)**
Solder: Tin/Silver/Copper
PCB: Gold over Nickel

**J Lead Termination (Reflow)**
Solder: Tin/Silver/Copper
PCB: Silver

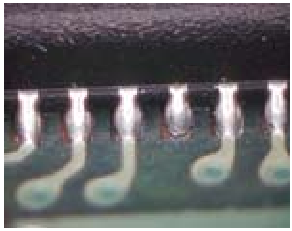



**Chip Component (Wave)**
Solder: Tin/Copper/Bismuth
PCB: Copper OSP

**Through Hole Termination (Intrusive Reflow Solder)**
Solder: Tin/Silver/Copper (SnAgCu)
PCB: Gold over Nickel

**Gull Wing Termination (Wave)**
Solder: Tin/Silver/Copper
PCB: HASL Lead-Free

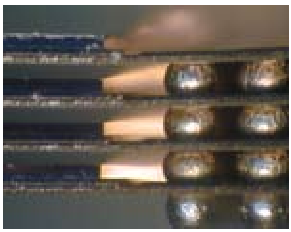
**Ball Grid Array (Reflow)**
Solder: Tin/Silver/Copper
PCB: Silver



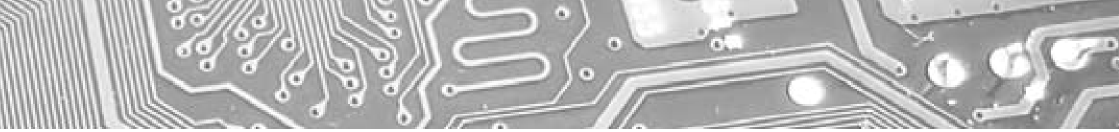
**Land Grid Array Termination (Reflow)**
Solder: Tin/Silver/Copper
PCB: Gold over Nickel

**Through Hole Termination (Hand Solder)**
Solder: Tin/Silver/Copper
PCB: Tin

**Through Hole Termination (Wave Solder)**
Solder: Tin/Copper/Bismuth
PCB: Copper OSP



**STACK Package (Reflow)**
Solder: Tin/Silver/Copper
PCB: Gold over Nickel



# Lead-Free Defect Guide

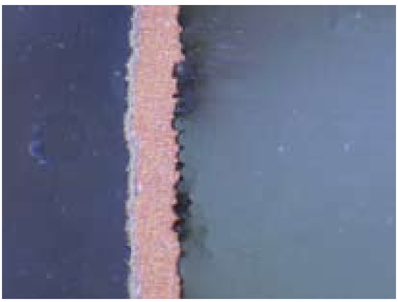
The collection of photographs on the following pages should help to illustrate some of the most common assembly and soldering defects associated with the introduction of lead-free technology. In certain markets these may not be considered defects, only examples of variations in the process; however each example should be investigated to confirm its possible impact on the acceptability or reliability of the product to the end customer.

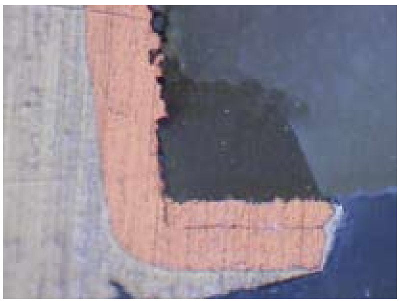
If you experience other lead-free process issues not illustrated in this guide, SMART Group would be happy to include them in future editions of the Lead-Free Defect Guide. Just send examples by email to technical@smartgroup.org

## Soldering Iron Corrosion

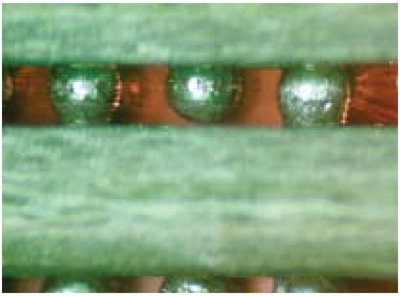
During the early introduction of lead-free, solder iron tip damage was very common due to the variability of the plating on some tips. It has also been caused by poor control and limited training of the impact of the solder alloy and the reluctance for staff to switch off irons when not being used. The example shows erosion of the copper core due to lack of protective plating which may have become damaged due to incorrect use of the iron tip.

## Copper Hole Wall Pullaway





Copper plating separating from the plated through hole wall can lead to problems, this should not occur even with the elevated temperatures of lead-free assembly. Many companies have changed the laminate materials for lead-free assembly. With a harder epoxy surface, cracking has been seen in the epoxy to glass interfaces. Changes are often needed in the plating process to improve or promote adhesion and eliminate hole wall pull away. This type of problem has been fairly uncommon prior to the introduction of lead-free laminates. Similar adhesion problems have been experienced on surface mount pads separating for the surface of the board. Often some of the newer high Tg laminates have a weaker copper foil adhesion to the surface of the board. This is commonly seen on area array packages where the surface pad is particularly small.

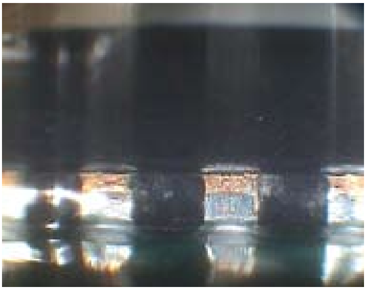


## Open Joint on POP Package

POP Package refers to package on package assembly or stack where two or more Ball Grid Array BGA or Chip Scale Packages CSP components are placed on top of each other and reflow soldered. One of the main reasons for open solder connections is package to package warping. This can lead to one or multiple open connections, these are very difficult to detect during component manufacture or assembly. The example photograph shows two joints on a demonstration package with one open connection. Normally this type of defect is easily detected during x-ray but with multiple balls on top of each other it is still challenging.

## Incomplete solder coverage

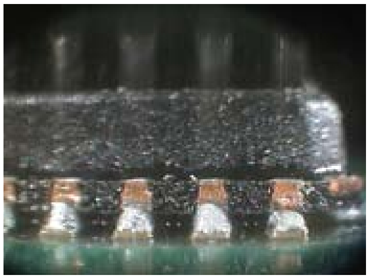
Lead-free wave soldering often results in solder joints with reduced hole file. This can be related to the process parameters, the solder finish on the surface of the printed board or the flux type. Each step needs to be examined to pin point the actual cause. Due to the increased temperatures of lead-free processes the design on multilayer boards can have a great influence on the solder solidify prematurely due to the heat loss on the inner layer connections. Wherever possible the inner layer ground or earth plane connection should be modified, increasing the clearance between the holes and the inner layer, reducing the number of connections or reducing the track connection width. The two example joints show incomplete solder coverage on the top surface of the board, however both easily exceed the minimum requirements of the IPC610 level 3 and should not be reworked.



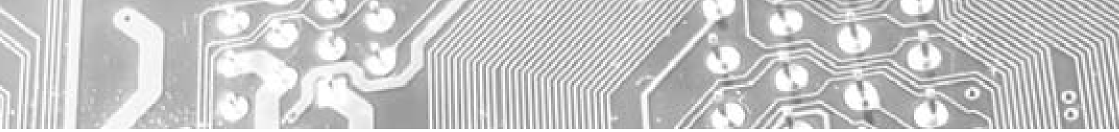
## Solderability Failure on LGA

Quad Flat No Lead QFN and Land Grid Array LGA packages often do not have solderable side terminations or they are poorly solderable due to the component manufacturing process. There is no requirement for the side terminations to be solderable based on IPC 610 requirements, however it is often the case that customers like to see a joint and it makes Automatic Optical Inspection AOI possible. Poor solderability of the edge termination is normally caused by a lack of a protective coating like tin on the surface of the copper lead frame or resin smearing during component singulation. Solderability is best tested using a wetting balance or in production with paste and a glass slide for reflow simulation.

## LGA Floating on Reflow



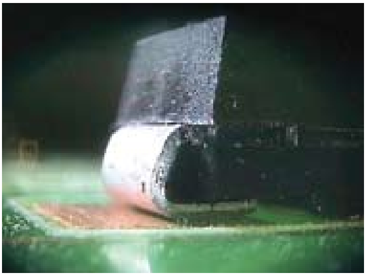
Open solder joints caused by lifting of a Land Grid Array LGA package during reflow is a very common problem. This is normally very easy to overcome with good stencil design. The most common reason for problems with these parts is excess solder paste under the centre pad of the package causing the part to float on the surface of the solder. As a basic guide the centre pad stencil aperture should be reduced to 50-60% of the area of the pad. Depending on the size of the pad the area should be split up with 4, 6 or even 9 equally spaced apertures. This modification reduces float on reflow and increases the chance of the volatile elements of the solder paste escaping reducing void formation in the joint interfaces.



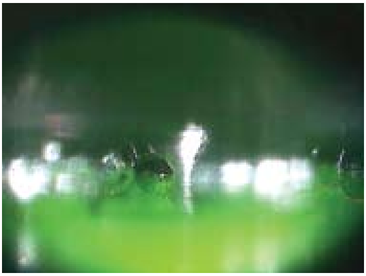
## Damage of Component

Moisture Sensitive Devices MSD's must be correctly packaged to prevent them being mechanically damaged during handling, or cracking, when they pass through the reflow process. The waffle trays of components have a thick plastic band around the trays which would not be adequate to maintain the trays position if dropped. This in the case of a fine pitch gull wing lead that would probably lead to bent leads. Although the package when opened does have a desiccant bag, there was no evidence of a relative humidity indicator strip, which should also be present to show. High levels of humidity can lead to pop corning/cracking of the component body during reflow. Examples of cracking are also provided in the defect section of this guide.

## Lead-Free Wave Solder Skip



Here is a solder skip mainly due to incorrect wave height as there is no evidence of solder on the pad or the termination. This may be a lead-free joint but the principle behind the defect is the same. If there had been solder on the termination the process parameters may have been correct, problems with the pad solderability may have existed. In the past skips have also been seen with different component suppliers and different lead forms on this type of capacitor. This type has the highest stand off of any component. When investigating skips look at the design, differences in component lead form and process issues like flux gassing and wave height.



## Solder Balls on BGA

Good example of solder balling under a BGA which is certainly outside of the IPC criteria. The ball will have undoubtedly reduced the insulation separation distance below the minimum for the product design, however reworking the component to remove it may be extreme. The solder ball is most likely as a result of using paste to rework the part and excessive paste resulted in a solder ball. In the case of rework its worth looking at the new dip paste for rework as it can eliminate rework stencils.

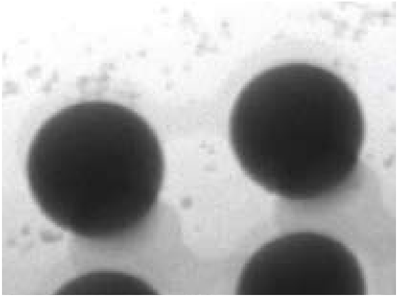
With care and a high pressure air line the ball could probably be removed rather than removing the component, it's worth a try and inspecting the component location after with optical or x-ray equipment.

## BGA Ball Damage

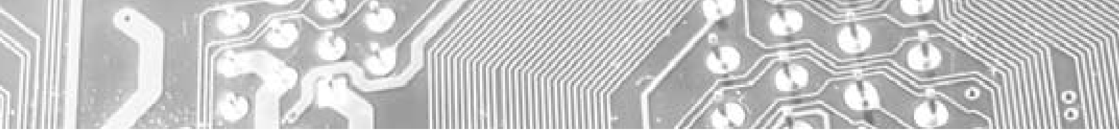
It is unfortunate but mechanical testing of joints can happen on production boards and the result is shown here, the ball has been broken off the BGA and the pad. Probing BGA joints is poor practice when inspection techniques like x-ray and endoscopes are readily available in manufacture.

Great care needs to be taken by skilled staff if testing of suspect joints is to be considered. It is clear from the damage to the ball surface and the joint surface has been subjected to a high force. It is very easy to apply well over 1kg of force with a tooth pick and this type of action should be avoided or very well documented.

## Lead-Free BGA with Tin/Lead Process



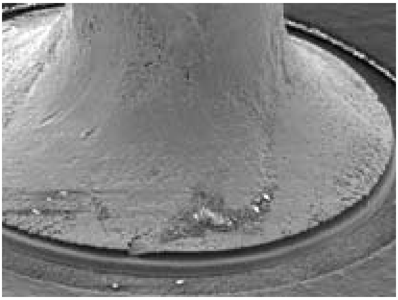
Lead-Free BGA reflowed with tin/lead solder paste profile is not uncommon in manufacture. Due to the tin/lead profile used peaking between 210-215°C the SnAgCu ball did not reflow and coalesce like a tin/lead ball would have done. The result is a good looking solder joint formed with a lead-free ball. The solder paste has not wetted the entire pad surface. Work is being undertaken to look at the reliability of joints in this interim period where lead-free BGAs are often being shipped for use in tin/lead processes and it's still a common issue for debate. The optical image shows the ball has not reflowed, the X-ray image of the same joints show a perfectly round ball when it should have been oval in shape in this angled view.



Reliability data to date shows that if all joints are soldered and the balls do not reflow in the same way that high temperature balls do not or, if the tin/lead reflows with the lead-free ball and the alloys are fully mixed, there should not be an issue. If, as is more likely in the real world of manufacture, only part reflow of the balls takes place, probably on the outer most terminations, reliability may be affected. There are many conflicting reports on this particular issue. One of the NPL Studio Projects looked at this specific issue and the report can be found at www.npl.co.uk/ei

If you were to microsection this joint you would see a sound intermetallic interconnection formed between the tin/lead solder, the lead-free ball and the PCB pad surface. Further details on this issue are covered in CD-ROM "BGA Inspection and Lead-Free Defect Guide" available from SMART Group.

## Lead-Free Through Hole Fillet Lifting



Manual optical inspection can pick up fillet lifting which has often been reported on lead-free soldering trials. The joint was produced with tin/silver/copper and clearly shows the solder fillet lifting from the pad. Often the pad can be lifted from the laminate as well. There was no lead contamination reported in the solder alloy or at the lead/pad interface, which suggests that there is still research necessary to determine the cause of this lead-free process defect. This type of fault has not been shown to cause joint failure on testing conducted in the UK and Japan. Samples of boards featuring fillet lifting have been through thermal cycle testing of 2200 cycles of -55°C/+125°C with dwell times of 40 mins with no electrical failure. The report on the NPL project is available with further details on their website. EPSON engineering in Japan has also undertaken reliability testing on this type of process issue and found it not to be a concern on plated through hole boards.

Much of the cause is related to the expansion and contraction of the printed board that occurs during the soldering and solidification of lead-free materials. This type of issue can be seen during wave soldering, pin in hole intrusive reflow on both sides of the board and on selective soldering. This process issue is covered in the IPC 610D inspection criteria.

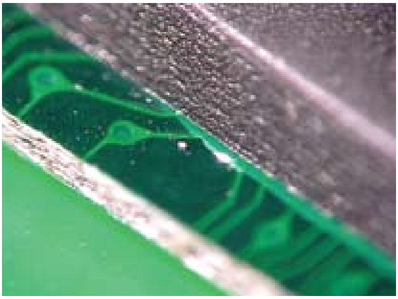
## Paste Non Coalescence

0201 chip component with lead-free solder paste that has not fully reflowed in air. This is a common fault on small paste deposits when people return to a traditional soak profile. Some solder pastes are far more tolerant than others, the resulting solder joint is sound but the balls have not fully reflowed into the bulk of the solder joint.

This type of issue has been seen during the last couple of years on 0201 and 01005 chip terminations in convection reflow but never in vapour phase soldering due to the inert atmosphere. Basically the small solder paste deposit is exposed to a long period at elevated temperature during a lead-free profile which reduces the performance of the flux in the paste. It is seen more with profiles that have a long pre-heat like a traditional soak profile. A profile which does not have long dwell prior to reflow is less affected.

Many paste suppliers have recommended the straight ramp profile but in the real world many users have not been able to achieve this with lead-free on existing ovens. In most cases the actual joint is satisfactory and only the outer surface shows this effect; however, most people would not accept this type of process fault.

## Component Cracking/Popcorning

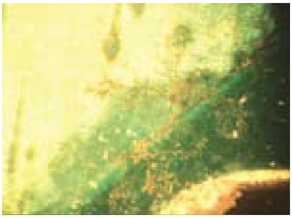


Component cracking is normally due to the incorrect use or specification of the component. Initially check the supplier's maximum soldering temperature and duration. Also consider any other special requirements for component storage. The first image above shows the crack in the laminate to plastic interface. The second image shows the lifting of a track during rework of a BGA. The part popped during videoing the rework sequence and showed the lifting of the copper foil, which is fairly unusual.

The most common cause, as in the example shown, is excess heat coupled with high water content. If the component supplier specifies a maximum temperature for reflow of <220 deg °C the component should not have been designed into the product. This is an example of poor design for manufacture, particularly for lead-free. Components can be assessed against the IPC or IEC process compatibility specifications.

Although the specifications do not necessarily relate to lead free, the specifications can be used as a reference source. The components can be assessed using the documents but the margin for error will be less. BGA parts are more susceptible to cracking than plastic TSOP or QFP parts and care should be taken to store the parts in a dry environment. It is also more common to damage parts during rework than in the initial production soldering steps.

## Copper Dendrites/Surface Corrosion



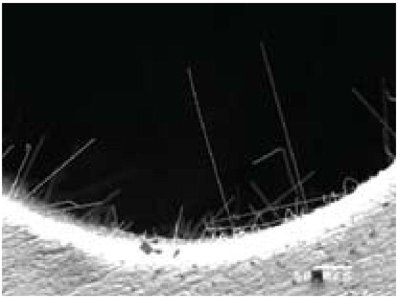
The three examples all show copper dendrites formed on the surface of printed circuit board assemblies. In each case they lead to intermittent failures in the field. They are not specifically lead-free process related but can occur if the correct controls are not maintained in a manufacturing facility, some engineers have reported this failure mode in lead-free when moving to VOC free fluxes. They will occur with a tin/lead and lead-free process and need to be analysed to find the root cause of the problem.

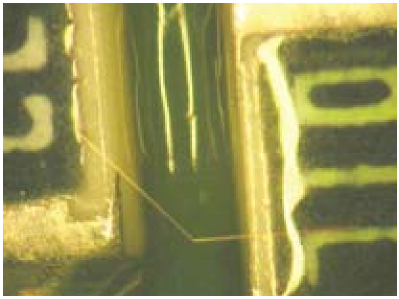
The examples show the formation of a copper dendrite/fern between two conductors. This fault may occur when flux residues remain on the board surface and are then subjected to high temperature and humidity. A circuit with a voltage applied of 5-10 volts can then allow the formation of a conductive path on or through the moisture layer.

A copper dendrite often creates an intermittent fault which can be very difficult to pinpoint. One of the examples shown was seen under the solder mask coating hence the contamination was from the printed board manufacture stage and not assembly. Contamination testing and Surface Insulation Resistance SIR assessment are two techniques often used to monitor and control the levels of harmful contamination on finished products to help avoid the possibility of corrosion.

Contamination that causes this failure mode is not only from the flux, it can come from the cleanliness of the printed board prior to use. It can also be caused by the design of the board, the way it is mounted in a product and exposure to changes in temperature and humidity.

## Tin Whiskers





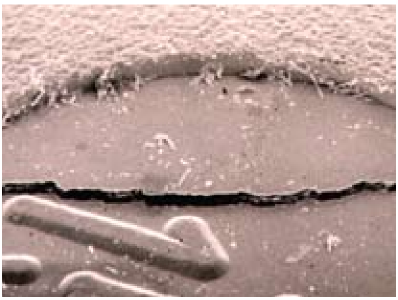
The images show tin whiskers on the surface of a plated through hole printed circuit board coated with tin. The boards were produced and shipped to a manufacturing site in Europe and when examined prior to assembly, found to have whisker growth. Tin has become popular on printed boards as one of the alternative coatings; tin has also become the finish of choice in the component manufacturing industry. However many people have shown concerns over the formation of whiskers.

There has been a considerable amount of work and technical articles produced on whisker formation and the potential for failure in electronics. There is still a lot of work being undertaken around the world on this subject. The reasons are that we still do not have guaranteed whisker free products or the process where the chemistry is being used is not being maintained correctly. If you follow guidelines on designing a whisker free process and use materials that should not form tin whiskers, somehow they still appear. This problem is not specifically a lead-free issue as it has been around for years as a possible problem. The increasing use of tin as a component finish and printed board alternative coating has highlighted the potential for failure. Organisations like NEMI and JEDEC have provided guidelines on what causes whisker formation, ways of accelerating testing for whisker formation and preventative strategies. Three tests, in conjunction with SEM (Scanning Electron Microscopy) assessment, are generally accepted for tin whisker testing:

Storage at 60°C and 93% RH for a minimum of 3000 hours.
Storage at ambient temperature (30°C) and 60% RH for a minimum of 4000 hours.
Temperature Cycling (-55 to +85°C) for a minimum of 1000 cycles.
Current understanding is tin whiskers occur due to stress formed in the plating. This causes tin filaments or single crystals of tin, commonly 2-5 microns long to be forced out from the surface of the plating during the life of the product. To reduce the possibility of formation engineers have looked at baking to reduce stress and changing or increasing the thickness of barrier layers.

This is the third practical example of whiskers examined on different products since the introduction of RoHS, each theoretically should have not have occurred based on current recommendations and specifications. The other two were related to RF shielding mounted to printed board assemblies. In both cases the whiskers fell from the surface of the shield on to the surface of the board assembly during storage and service.

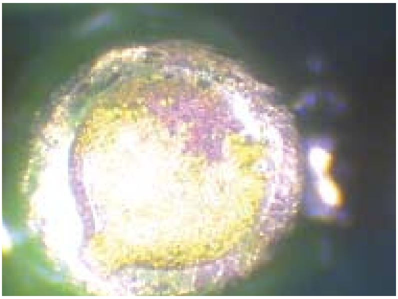
## QFP Damaged (Popcorn) Component

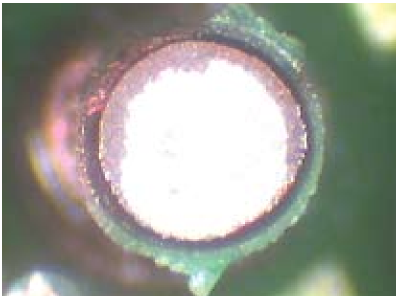


The Quad Flat Pack (QFP) is a typical example of popcorning during reflow. Moisture in the package has expanded during reflow soldering and eventually caused the plastic to crack allowing water vapour to escape. Considerable pressure can build up in a device if care is not taken, Ball Grid Array (BGA) packages are the most susceptible to this type of failure mode but many plastic parts can also be affected.

Examples like the above were found during Automatic Optical Inspection (AOI) of boards after vapour phase soldering. The AOI showed up solder shorts and component movement on the QFPs which was caused by the pressure of the water vapour escaping through the side of the device. The fault is nothing to do with the soldering process but is due to the storage conditions of this part, alternative QFPs did not exhibit the problem on VP or convection reflow. The examples above show the SEM images, the first being a microsection of package separation, the second image shows the cracking of the QFP. A video of the soldering process clearly showed the part moving during reflow and the vapour escaping the plastic body of the device. The video clips are available to view on line at www.smartgroup.org

## Cracking in BGA Joints





Two images of balls separated from a reflowed BGA which has been mechanically removed from the surface of a PCB. The joints on the board were questioned after environmental testing showed intermittent electrical results with cracking suspected. X-ray and optical inspection can often find it difficult to determine these types of faults.

Using a dye penetrant fluid to detect cracks on a board prior to breaking off the BGA can be very revealing. The so called "Dye and Pry" test is often used for joint assessment to quickly but destructively assess joints. Further information on the method and the type of results expected are featured on the SMART Group web site. The first example illustrates the test using a yellow dye showing 75% coverage, virtually all the joint has cracked away from the pad. The red dye used on the second example shows less than 10% crack propagation but in this case under the pad on the surface of the board. Temperature cycling, vibration or technology

# LEAD FREE PROCESSES:
## Electroless Nickel Immersion Gold

Artetch are RoHS compliant and have been offering Lead Free finishes since 1996 and are leaders in both the UK and Europe for the Immersion Silver Process.

Following increasing customer demand Artetch are now installing yet another Lead Free finish and will be offering Electroless Nickel Immersion Gold (ENIG) from its Littlehampton based facility. The equipment being installed will be fully automated to ensure a consistent and repeatable process to the very highest standard.

artetch circuits Ltd.

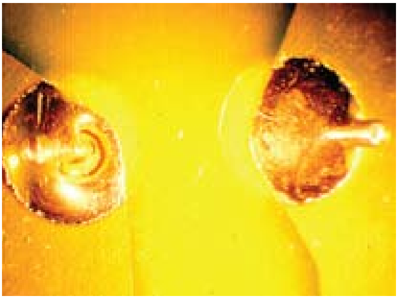
Artetch Circuits Limited
Riverside Industrial Estate
Littlehampton
West Sussex
BN17 5DF
+44 (0)1903 725365
sales@artetchcircuits.co.uk

drop testing can show up different failure modes. It should be borne in mind that, although lead-free alloys are stronger than tin/lead, mechanical shock can be an issue.

## Incomplete Pin in Hole Reflow

Through hole connector joint produced by intrusive reflow with tin/zinc solder paste which has been found in many trials to give a very poor visual appearance after reflow in air. In this case the solder joint has formed but many of the paste particles have not reflowed and coalesced fully into the joint. This can occur when there is too long a delay in the soak period of reflow prior to reaching reflow temperature. This can exhaust the protective layer provided by the paste. It can of course also be due to the joint not reaching reflow temperature, but unlikely as this is a lower temperature alloy.

The problem can also be related to the reflow of this paste in air. Although paste suppliers offer this alloy for reflow in air many companies have found nitrogen necessary to get the best results. The profile used did meet the recommendation of the supplier. Mechanically the joints were also sound. The second example shows the same type of defect on a pin using tin/silver/copper paste.



## Insulation Contamination in Joints

The incomplete solder fillet has been caused by insulation on the wire termination falling into the joint during wave soldering. This is a common fault when leads are soldered directly into a circuit board particularly if the insulation is a low temperature material. The same problem can occur if the encapsulant on components runs down a lead during manufacture, wound coils and transformers are a classic example. This is often also seen on radial inserted components as the body of the part normally sits on the surface of the board.

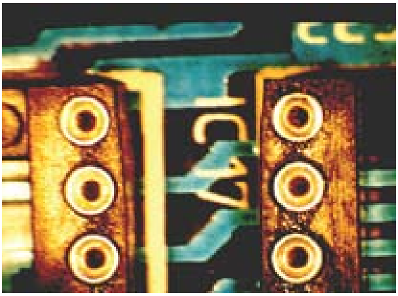
All components should be examined for their process compatibility, especially with the increase in temperatures that parts see in wave and reflow soldering. With lead-free the soldering temperatures will be 15-20°C higher as is the top surface of the board during contact with the wave. During pre-heat the topside of the board may also be hotter and, with the combination of solvent from the flux, may cause damage to component body material or solder masks.

## Lead-Free Pad Lifting

The microsection illustrates a typical pad lift that can occur with a lead-free process. It can be seen after soldering through hole terminations where the solder contracts during cooling causing the lift or peel back of the pad. During soldering the board material will also expand in "Z" axis deforming or placing strain on the pad. Either the pad is lifted, the fillet lifts from the pad, or fillet tearing can be seen on joints of this type. The joint image above shows some degree of fillet lifting and pad lift on the same joint.

Concern is shown for pad lifting as it could be associated with a track connection and in this case the track and the pad could separate. The mechanism is similar for pad lifting, mostly associated with the expansion and contraction of the laminate and the solder. The IPC 610D solder joint inspection criteria does cover lead-free, but does not give criteria for pad lifting as it relates to a lead-free process. The criteria for general pad lift is the pad should not lift any more than the thickness of the pad from the surface of the PCB laminate.

This is not really a new process problem as it has been seen many times with tin/lead on thick multilayer boards where the expansion in the "Z" axis is high. When and if the solder fully penetrates the through hole to the top surface of the board and solidifies, the laminate would still be contracting. This can and has resulted in pads lifting at the edges. Much work was done in the United States in the 70's to convince military customers that this was not a reliability issue.

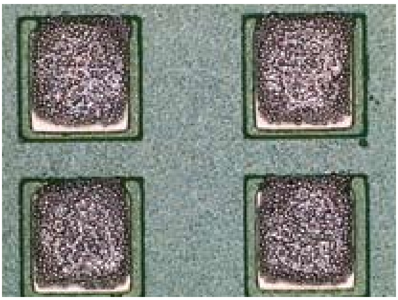


## Conventional Component Damage

Conventional components are often not rated at high temperatures for preheat and wave soldering with lead-free so care needs to be taken if they are used on the top surface of the board during wave soldering. In the example IC sockets have melted/distorted due to excessive heat. The same damage can occur on the surface of the board during rework of closely spaced surface mount components. Minor distortion of this type of socket can make it impossible to manually or automatically insert IC packages.

Some of the cheaper connectors and sockets on board designs will need to be re-evaluated for lead-free soldering. If rework is to be conducted it is possible to place a heat shield between the components and the surface mount device. The heat shield can be as simple as a piece of Kapton tape or a small section of thin laminate.

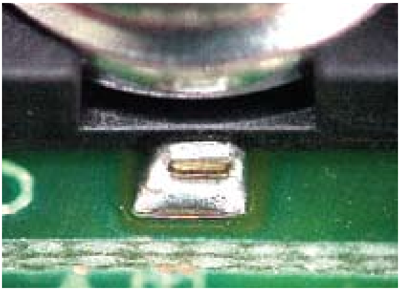
## Solder Paste Misalignment



Minor misalignment of the solder paste is shown on these 0201 pads. This has been seen on the surface of boards which also included fine pitch parts. These alignment problems are becoming a real issue with large panels, the pad to pad dimension on opposite edges of the board are often different to the design data which is used to make the stencil. Although the printing may be satisfactory the board material tolerance can make the alignment of a stencil to the pads difficult. Modern printers can provide correction to minimise the offsets but it does mean there will be a compromise on the exact position of the paste to the pad. The example board was being printed and reflowed with lead-free paste and also featured 0201 chip components on the opposite edge of the board, which proved difficult with the alignment of the board.

During trials it was also possible to see the differences on printing side one and two for alignment error after the first reflow operation.

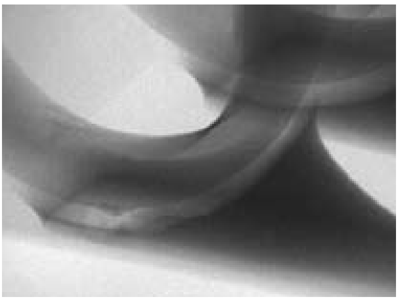
There are limits to standard board materials and we all need to appreciate the capabilities of the basic material. The problem is not specifically related to lead-free processing, but an issue for small part processing in the future, higher processing temperatures being a contributing factor.

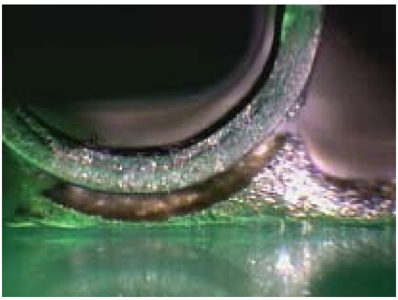


## Open Solder Joint

The electrolytic capacitor has an open solder joint, the solder paste has reflowed and wetted the pad but the joint has not formed on the lead. Normally this would be associated with poor solderability of the component termination and the parts would need to be tested using the glass plate test or a wetting balance. However, in this case the design of the pad and the paste deposit extended too far under the body of the plastic base of the component. The lead was lifted above the surface of the solder paste as it reflowed before the lead could wet. When paste wets, and depending on the volume of paste and width of the pad, the non-wettable component body may be lifted by the meniscus of the solder. If the surface plating of the lead is quick to wet a joint will form but if it's slow it may result in open connections. How often are design faults incorrectly logged as poor wetting, open joint, joint contamination in your defect and rework logs?

## Secondary Reflow of Lead-Free Joints

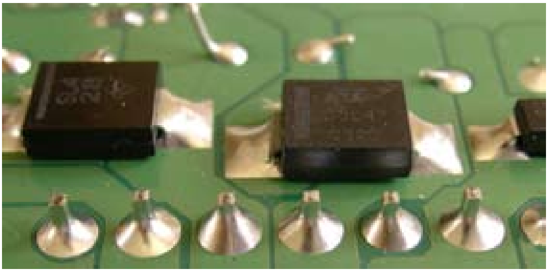


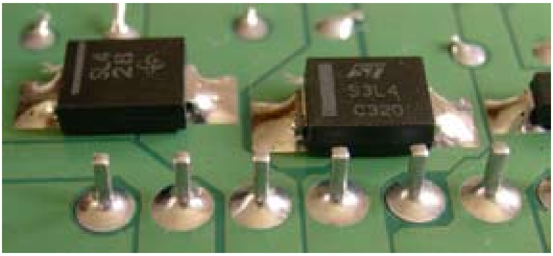


Control of board temperature is very important during wave soldering to avoid secondary reflow on the topside of the board. This can occur when a board passes over the solder wave and the heat is transferred

# LEAD FREE SOLDERING FROM THE UK'S No 1 SOLDER MANUFACTURER

## Can you tell the difference?


Board Wave Soldered with SN100C Lead Free Solder


Board Wave Soldered with Sn63 Pb37 Tin/Lead

**DKL Metals Ltd**, is the **only licensed** UK solder manufacturer for the three top selling RoHS compliant solders in use today across the globe

**SN100C** Licensed by Nihon Superior Co. Ltd, Japan
**SAC305** Licensed by Senju Metal Corporation, Japan
**SAC387** Licensed by IOWA State University (ISURF), USA

Why waste time with "clones" and untried or tested "look-alikes" when the **proven** lead free wave soldering alloys are available in the UK and Ireland from us and worldwide through our Global Partner Alliance. Full product details on all our soldering products are available for download on our website www.dklmetals.co.uk

DKL METALS LTD
**www.dklmetals.co.uk**

**DKL Metals Ltd**
**Avontoun Works**
**Linlithgow**
**West Lothian**
**EH49 6QD**
**Tel: +44(0)1506 847710**
**Fax: +44(0)1506 848199**
**E-mail: sales@dklmetals.co.uk**

GLOBAL PARTNERS FOR LEAD-FREE SOLDERS

through the board by conduction. Correct topside profiling can eliminate this issue by use of a process control tool to monitor changes in contact time and temperature.
Secondary reflow occurs when the temperature of the reflowed alloy is reached or exceeded during wave soldering. Secondary reflow of tin/lead joints and joint separation can be exaggerated by:

High solder wave temperatures
Long wave contact times
Higher pre-heat temperatures
Variation of board thickness or density
Sag and flex of the printed board during processing
Sealed nitrogen chambers

It is understood that some companies have experienced problems over the last couple of years with secondary reflow of lead-free joints during wave soldering. This has led to separation of the termination from the bulk of the joint. In the past the same phenomenon has occurred with traditional tin/lead alloys when the top surface of the board reached or exceeded 180°C.

Lead separation has been seen in the past on large components where the board and the component body can flex against each other during wave contact effectively pushing corner joints apart and leading to an intermittent joint. The lead can be left sitting on the surface of the joint like a ball in a cup. On some occasions with tin/lead joints the lead and the bulk of the joint has been seen to separate from the pad surface. The first conclusion is this is a solderability issue with the PCB plating but it may not be.

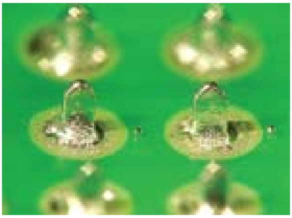
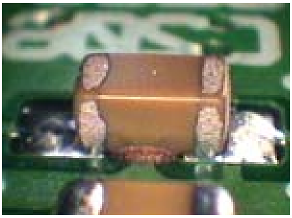
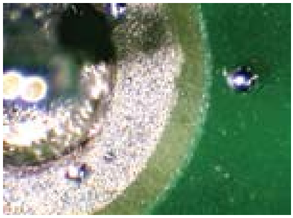
In the case of a lead-free process it is assumed that if tin/lead is used on the PCB or on the components and it does not fully dissolve into the bulk of the solder, the layer at the interface could weaken or reflow at a lower temperature than the paste alloy. The same problem has been highlighted by researchers and early users of lead-free with Bismuth alloys, who have experienced lead contamination issues as well.

Practical testing by the author has shown this to occur on tin/lead components soldered with SAC paste and then reflowed again with a peak temperature of 175-180°C. Simulations conducted by NPL and measurements taken on peel strength at elevated temperatures clearly shows a reduction in strength of the joints. SEM examination of the joint interfaces show the difference in joint separation with a hot tear on the joint surface as opposed to a brittle fracture that occur when mechanically separated.
A video clip is available in the Members Area of SMART Group web site showing this type of defect happening. It is a simulation of what can happen when a board reaches 180°C with lead contamination at

the lead interface. A series of still images are shown below from one of the videos. They show the joint prior to reflow and with lead separating from the bulk of the solder alloy at different stages.



## Copper Dissolution in Lead-Free

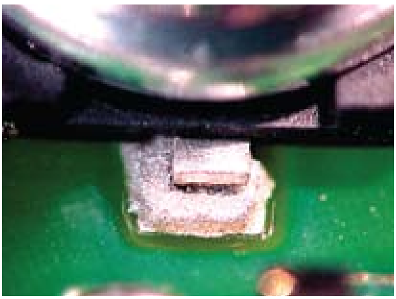
Dissolution/erosion of the plating or coating is not a new issue, but with the introduction of lead-free soldering materials, new process stages like selective soldering and higher process temperatures, it is a real issue. Three examples above show:

Copper dissolution of the copper plating and foil during selective soldering
Metallisation dissolution of capacitor terminations in wave soldering
Copper dissolution of the copper plating during rework and repair
(The dissolution of copper has also been seen on soldering iron tips if the protective plating over the copper core is thin or damaged during use)

To date this issue has had little attention but potentially, when looking at the images above, some engineers may have concerns. Example microsections have been presented in international seminars and discussed in technical forums, in these cases perfectly formed joints have been produced but below the surface virually all the copper has been dissolved. To date no specific details on the process used have been circulated, however, the "IDEALS Lead-Free Project" an early European Funded Lead-Free Project reported this potential problem. When the issue was first discussed with other engineers very few people had observed the phenomena. In fact, a review of the author's previously produced microsections showed limited evidence of the problem. During a review of many of the lead-free sections produced by National Physical Laboratory (NPL) no significant evidence of erosion could be found. Typically all the joints would have been produced under strict controls and a copper reduction of 2-4um is not untypical on joints. More evidence of the problem surfaced during the "SMART Group Lead-Free Experience" first run in 2003 in the UK, this was specifically on solder levelled boards.

It is fair to say that increased copper dissolution into the solder wave or selective pot was considered inevitable due to increased soldering temperatures and higher tin content. Further, we have experienced copper dissolution during rework of joints and lead-free solder joints often with double the thickness of intermetallic layer even on reflow suggesting greater copper removal. However there is no direct correlation between the intermetallic layer thickness and copper removal. Measuring the copper thickness on the barrel of a hole that has been reworked can often show a greater dissolution rate on one side where the solder and heat are applied. Solder levelled boards, when examined by microsection, tend to show copper reduction. This was especially prominent when levelling is conducted twice to overcome thermal and wetting issues.

Concerns were expressed at the impact of high copper levels in wave and selective soldering processes but this really relates to our experience with tin/lead alloys. Copper is the main contaminant found during normal production; it does impact the soldering yield by increasing solder shorts. Higher copper levels do make a difference to the visual appearance of the joint and temperatures. However, it has not been shown to impact joint reliability. With lead-free copper it is already part of the alloy system, between 0.5 - 0.7% added to the mix to lower the melting point. SMART Group is helping to promote a new NPL Studio Project specifically launched to examine dissolution problems and how industry can prevent this issue occurring in production. For further details visit www.npl.co.uk/ei/

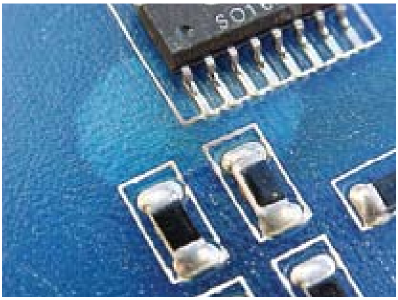


## Non Reflow of Paste

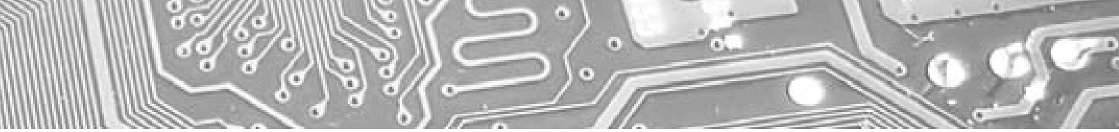
Non wetting in this example is due to the incomplete reflow of the solder paste. The profile temperature was too low and only just reached 230°C on one measured point on the board. There is a reason for having multiple channel profilers, monitoring different areas of the top surface and the bottom side to balance heat input. Clearly this component termination on the board did not reach or remain at reflow temperature for the required time due to the wide difference in temperature across the board, typically 40-60 seconds above liquidus may be required even though it only take a few seconds to form a reliable joint. In the case of tin/silver/copper alloy the peak reflow temperature on the board can be 235-240°C.

Although most professional reflow ovens can handle lead-free reflow profiles, care needs to be taken during reflow by correctly profiling new products and pastes. Most 5-7 zone reflow ovens can achieve a DeltaT of 10-15°C across the surface of the board at a conveyor speed of 50cm/min. But to successfully achieve this a lower DeltaT profiling needs to be done correctly by mounting thermocouples on the surface of the board with aluminium tape or high temperature solder, not sticky tape. Ask the question, how many times have you been shown a profile with one plot, has the profile been done correctly?

## PCB Delamination



Example of delamination on the surface of a board and one after cutting the surface of the laminate to show the copper inner layer and epoxy and glass bundles. Delamination is a blister, air gap, formed when water vapour forms and expands during heating, not like a blister on your skin which is filled with fluid. Delamination may occur when moisture in the board expands during soldering or rework, with higher temperatures of lead-free there is more energy placed on the materials. When and if moisture accumulates in specific areas of the board more energy is focussed in thoes particular areas. The second image shows a lack of proper adhesion of the epoxy to the surface of the inner layer which may be related to the copper surface preparation, the prepreg or the press cycle.



## Solder Dross

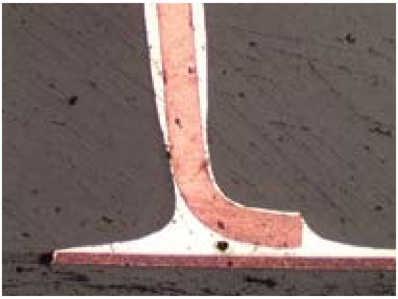
Lead-free dross will look different to tin/lead and certainly it is more valuable with increasing tin prices. The dross removed from the surface of a solder wave or skimmed from the main bath will have a different appearance. The example shown has been skimmed from a solder bath running a tin/copper alloy. Just like tin/lead it is possible to get fine solder shorts between terminations on the base of boards, If the bath is not maintained or there are parts of the baths that have cold spots, this could lead to needles shorting like we have experienced dross shorts in the past.

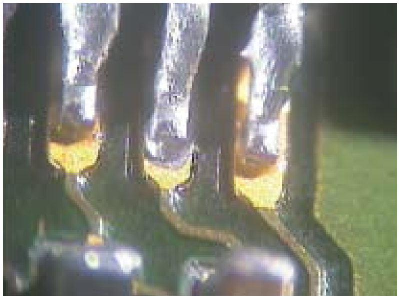
Care needs to be taken when taking samples for the bath; follow the procedures provided by material suppliers to get a representative sample from the running wave after it has been running for a period of time. Two types of needles can be seen either tin/copper or tin/iron, in this case due to possible attack of the solder on stainless steel surfaces. Accurate surface analysis is required to determine what may be causing the shorts.

## Incomplete Wetting

The introduction of alternative surface finishes has created a difference in inspection criteria; this has been exaggerated by the reduced wetting characteristics of most lead-free alloys. In order of printed board solderability performance, solder is best, then gold, silver, tin and copper OSP. This is of course based on the specification, quality and consistency of the finish provided. When lead-free solder paste is reflowed on a pad there may be incomplete coverage of the pad. There are many factors contributing to the degree of surface coverage: engineers should of course optimise their process to obtain the best performance from the material set selected, however, these examples are all acceptable and should not be reworked.

## Solder Wicking

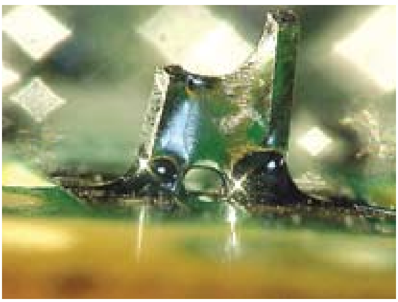


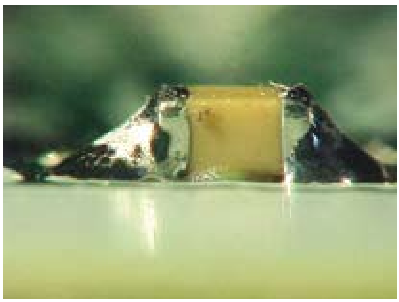


Solder wicking refers to a problem where the solder paste reflows and tends to wet the component termination rather than the pad and lead as it should do. Reasons for these phenomena are slow wetting of the pad or much faster wetting of the termination. Pad solderability can be tested or monitored in production very simply with a wetting indicator and it's a great tool to compare boards, lines or changes in process parameters. Documents on wetting indicators and how to use them are available from the SMART Group web site.

Faster wetting of the lead can be caused by the profile or the soldering process. Vapour phase can be set-up to allow the leads to get to soldering temperature before the board surface. Old tin/lead terminations in a lead-free process produce great wicking, the tin/lead surface coating wets very quickly, literally sucking the solder up the lead before wetting can take place on the pad. The image is a microsection showing wicking on a fine pitch lead, despite this the joint is still satisfactory. The solder can be easily seen on both sides of the lead. The second image shows wicking on SOIC leads. The root cause was a poorly cleaned board during paste wash off.

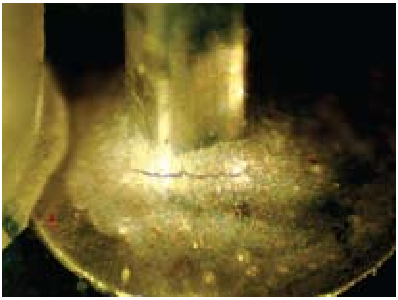
## Flux Exhaustion

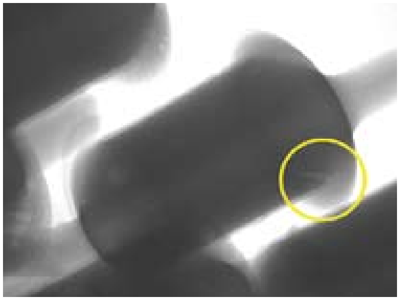




Lead-free wave soldering with increased pre-heat and wave temperature can be demanding to existing low solids fluxes used in a tin/lead process. Changing flux type and increasing solids content is often a solution to improving soldering yields and reducing solder shorts. Solder shorts are more common in lead-free, hence the need to re-examine the design rules and the materials used to open up the process window. Simple design rule changes can be very effective in making products more processable. Looking closely at defects under higher magnification on the shop floor can often show the root cause for problems. The first example is a solder short on two long leads, the second example on the same board shows poor wetting to the chip terminations but good wetting to the pads. Both could indicate that the fluxing stage is the problem or, alternatively, when the board enters the wave there is not enough activity left in the flux.

## Lead-Free Fillet Tearing





Lead-free fillet tear in plated through hole joints is not uncommon, two examples are shown of this phenomena optically and using real time X-ray inspection. This is a typical example found on the surface of the solder joint after wave soldering, it can also be seen on selective solder or intrusive reflow joints. The X-ray image shows visible evidence of tearing on joint surface and just below the surface. In IPC –A--610D the latest version of the criteria refers to this as a hot tear/shrink hole and is acceptable on lead-free joints if the bottom of the tear is visible and the tear does not contact the pad or lead on class 1, 2, 3 products. It is a defect on each inspection class if it's found on tin/lead, the bottom of the tear is not visible or the tear touches the pad or lead. In reality it is only seen on lead-free tin/silver/copper alloys and is normally very difficult to see the bottom of the tear hence making it a defect, according to the standard, requiring rework.

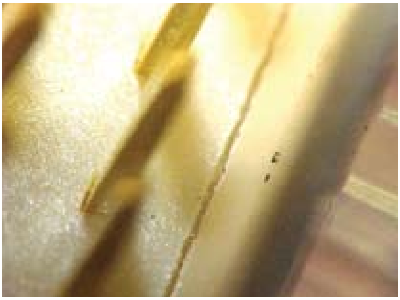
Selected combinations of tin/silver/copper with a silver content of 3-4% are prone to cause this tearing during solidification of the solder. The solder is rough in appearance due to the shrinkage of the metal between the tin dendrites. There is no evidence currently that this has any impact on the reliability of the solder joints.

## Outgassing & Blowholes

New lead-free processing and materials: old problem, same causes. Outgassing from a plated through hole board is caused by moisture in the printed board expanding during soldering. The gas comes out of the hole as water vapour while the solder is in a liquid state. Voids are mostly seen on the base of the board because the solder has solidified first on the top side hence the expanding vapour has only one way to escape. The size of the voids, blowholes or pinholes are related to the amount of escaping gas and the point at which the solder starts to solidify.

Testing boards is easy with the oil outgassing test: old problem, old test and same old solution. A paper on the test method is available in the download area of the SMART Group web site. Conducting the test shows where and how the gas escapes from holes and can show if the position is random or on selected areas of the barrel.

Baking boards can eliminate the moisture from the board and often does wonders to the solderability of new surface finishes, it does not necessarily get to the root cause of the problem. The most common reason is the thickness of copper in the plated through hole which may not be able to evenly cover poor drilling. This can also be impacted by the greater dissolution rates in lead-free assembly.



## Cracking of Connectors

High temperature connectors are used in a Pin In Hole Reflow (PIHR) application with lead-free paste at a peak temperature of 240°C; this is perfectly reasonable. Cracking has occurred at the corner of this boxed connector and despite being made of a high temperature material rated at 260°C, a crack has still occurred. The problem was related to the material and the flow in this section of the moulding leading to a brittle area.

When introducing PIHR do not accept all the information provided, always run some basic trials on parts prior to full production. It is always recommended to have samples before you design the board and stencil layout to look for standoff height, positions and pin size. This type of material issue should be picked up during prototype build or NPI (New Product Introduction) assessment and would have been very obvious during any connector mating trials.

# PPM/DPMO Monitoring Benchmarking Procedure

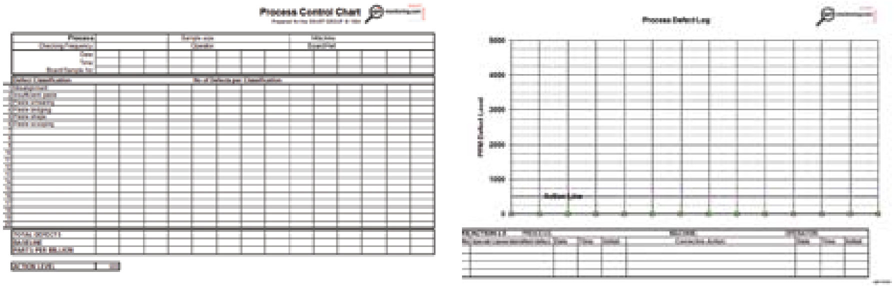
Monitoring Parts Per Million (PPM) defect levels at different steps in the lead-free assembly process is a common way of capturing information and assessing improvements. PPM monitoring is an ideal way of comparing yields on an existing design or tin/lead process prior to the transition to lead-free. Using defect levels as reference allows benchmarking a process or different company processes providing it's done consistently. SMART Group have a fair amount of experience in the implementing defect monitoring through involvement in the DTI and LEADOUT PPM projects.

Sample boards can be taken after any of the assembly stages and inspected to the requirements of IPC 610 level 3 and with reference to IPC 9261 (In Process DPMO and Estimating Yield for PWAs). Where visual criteria does not exist reference documents are available from the SMART Group, these can be used for solder paste printing, intrusive reflow assembly and selective soldering. SMART also offer a training workshop to introduce defect monitoring processes and tools to help achieve zero defect manufacture. A spreadsheet provided with the workshop records the defect opportunities, inspection results, process information and the technology level of the product being assessed.

To determine the PPM Defect Level for any of the process steps simply use the following calculation for the PPM or DPMO level:

**Number of Process Defects Found x 1,000 000 = PPM Level Opportunities for Defects**

Number of process defects are the total number of defects found during inspection of the sample boards or panels at the process step being examined. Opportunities for defects is the total number of defects that could occur at that step. As an example for stencil printing of solder paste the opportunities would be the number of apertures printed on the side of the board. The number of defects would be the number of solder paste printed apertures that did not meet the defined standard.




**Example of the type of spreadsheet used to capture information on the defect levels at different stages in manufacture.**

The following are the typical inspection steps that may be conducted:

## Stencil Printing Solder Paste

A minimum of 5 samples boards or panels are taken for inspection. A company can take a much larger sample if they wish. The minimum number of 5000 opportunities are to be taken which may require additional boards or panels to be inspected if the minimum number of opportunities is not achieved. The opportunities for error to be determined are based on the stencil apertures printed on the samples.

## Adhesive Application

A minimum of 5 samples boards or panels are taken for inspection. The minimum number of 5000 opportunities are to be taken which may require additional boards or panels to be inspected if the minimum number of opportunities is not achieved. The opportunities for error to be determined are based on the number of adhesive dot positions printed or dispenced on to the sample board.

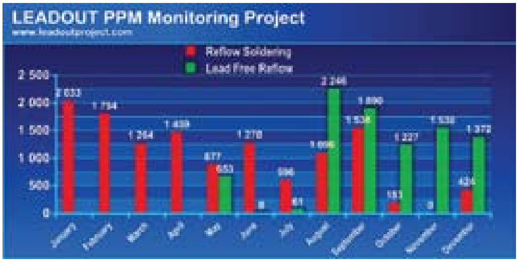
## Component Placement

A minimum of 5 samples boards or panels are taken for inspection after the final component placement step, this may include any manual placement. The minimum number of 5000 opportunities are to be taken which may require additional boards or panels to be inspected if the minimum number of opportunities is not achieved. The opportunities for error to be determined are based on the components placed on the printed board samples.

The defects considered after placement are not limited to misalignment they may include missing, reversed or damaged components. Reference should be made to the relevant sections of IPC 610.

## Reflow Soldering

A minimum of 5 sample boards or panels are taken for inspection prior to any rework. The minimum number of 5000 solder joint opportunities are to be taken which may require additional boards or panels to be inspected. The opportunities for error to be determined are based on the number of reflowed solder joints produced on that side of the board being soldered.



Example of the PPM monitoring results comparing tin/lead and lead-free produced during the LEADOUT project

The defects considered after reflow may not limited to solder shorts or opens, this should be fully defined before stating the monitoring process. Reference should be made to the relevant sections of IPC 610.

## Wave Soldering

A minimum of 5 samples boards or panels are taken for inspection prior to any rework. The minimum number of 5000 opportunities are to be taken which may require additional boards or panels to be inspected. The opportunities for error to be determined are based on the joints which are wave soldered at that stage in the process. When the number of wave soldering defects have been determined the PPM level is determined.

The inspection results from the process and number of sample boards or panels are also to be recorded for each stage.

## Selective Soldering

A minimum of 5 samples boards or panels are taken for inspection prior to any any rework. The minimum number of 5000 opportunities are to be taken which may require additional boards or panels to be inspected. The opportunities for error to be determined are based on the joints which are designed to be selectively soldered on the sample board design. When the number of selective soldering defects have been obtained the PPM level can be determined.

Monitoring a group of processes is simple to implement but it must be implemented correctly so that the data gathered is consistent and meaningful. This is also true when comparing yields between different board designs and manufacturing sites. It is recommended that all the process data is also captured for comparison with the defect levels. Maintaining a record of the technology level of the boards passing through the process during the monitoring steps is also important. For further information on the PPM Monitoring Implementation Workshop contact the SMART Group office.

# Lead-Free Design Assembly Checklist

Defects do occur and many engineers new to lead-free manufacture may experience defects they have experienced with tin/lead, however, some of the defects are new to industry. This check list is not exhaustive but may help eliminate some of the most common problems provided you have considered each of these questions.

☑ Place a tick in the corresponding box.

## Commercial

- ☐ Do we have appropriate equipment to carry out high temperature solder and rework if needed?
- ☐ Check if supplier qualification is required to ensure due diligence is carried out?
- ☐ Check operator(s) are trained to carry out lead-free process?
- ☐ Does the system allow us to control two processes without cross contamination?
- ☐ Check controls/gates are appropriate for incoming RoHS compliant material?
- ☐ Has the board been designed for Pb free or converted from an old design – if so has a review been done for suitability?

## Printed Circuit Boards

- ☐ What alternative solder finish are you going to adopt and why?
- ☐ What new solderability tests are needed by your supplier for these finishes?
- ☐ What new storage and handling procedures / equipment are needed for the new finishes?
- ☐ Is the finish appropriate for your product type?
- ☐ Do you need to consider a higher glass transition laminate for your product?
- ☐ What experience does your PCB supplier have with boards for lead-free assembly?
- ☐ Have you specified your PCBs for lead-free correctly?
- ☐ Do you require cross-sectional analysis from your supplier to demonstrate due diligence?

## Connectors

- ☐ Make sure all terminations have lead-free finishes on their termination?
- ☐ Check moulding is compatible with the higher temperature, consider cracking in high stress point areas of the moulding?
- ☐ Can higher temperatures affect the mating of connectors?
- ☐ Have you checked the top side board temperature and the connector body?

## Components

- [ ] Make sure all components have lead-free finishes on their termination?
- [ ] In the case of surface mount can components meet new soldering profiles?
- [ ] Can conventional components meet higher topside temperature over preheat and wave solder bath?
- [ ] Check conventional connector compatibility for intrusive reflow and wave soldering?
- [ ] Have moisture sensitive component requirements changed?

## Cables and Wires

- [ ] What plating is used on the cable strands, is it lead-free?
- [ ] Will the insulation stand up to higher manual and automated soldering temperatures?
- [ ] Is the insulation colour free from hazardous substances?
- [ ] Does the use of lead-free solders effect the flexibility of the wire or insulation?
- [ ] Does the use of lead-free effect the minimum wire strand size used due to copper dissolution?
- [ ] Have you evaluated the wire size for copper dissolution?

## Printing

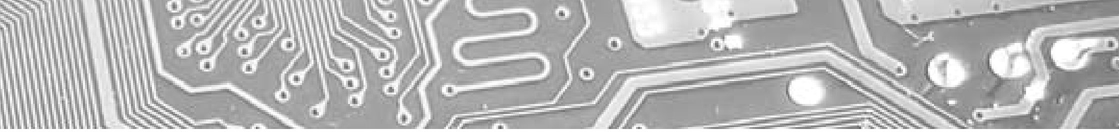
- [ ] Can staff identify the alloy types on solder paste containers and do the travel cards / build instructions define the paste?
- [ ] Check that the existing stencil with reduced apertures adequate for lead-free process?
- [ ] Check that stencil and tooling is correctly cleaned before using a different alloy?
- [ ] Check separation speed is adequate for lead-free process?
- [ ] Have you a procedure in place for swopping paste for the next batch?
- [ ] Do you have separate stencils, blades, and paste spatulas for lead-free?
- [ ] Make sure waste paste containers are correctly marked for alloy type?

## Component Placement

- [ ] Can you identify tin/lead and lead- free if part numbers remain the same?
- [ ] Make sure hand assembly and rework component stocks are correctly marked?
- [ ] Check misplaced parts are correctly identified before use?

## Reflow Soldering

- [ ] Have you checked profile requirements and oven capability for lead-free alloys?
- [ ] Have you determined what temperature fluctuation occurs in your reflow process?
- [ ] Do you have a profiling check procedure, how often and to what profile?
- [ ] Have you trained staff to reset the process from lead-free and wait till the temperature profile has correctly returned to the tin/lead profile before starting further production?
- [ ] Do the reflow cooling zones allow for control of cooling?



☑ Place a tick in the corresponding box.

## Through Hole Assembly

- [ ] How do you check components have a lead-free surface finish on the terminations?
- [ ] Do components have a different marking for lead-free, how can tin/lead terminations be identified?
- [ ] What impact will higher topside temperatures have on the component body?

## Wave Soldering

- [ ] How are you checking components and PCB finishes are lead-free to avoid bath contamination?
- [ ] Have you checked temperature control limits for solder alloy and pump operation?
- [ ] Change dross bucket marking to avoid mixing dross from different lines?
- [ ] Have you trained staff on bar solder identification; include purchasing, stores, operators and maintenance staff?
- [ ] Have separate de-drossing tools been provided?
- [ ] Are height and solder feed sensors compatible with lead free alloys?
- [ ] Considered your budgets, most lead-free alloys are 3 times the price of tin/lead?
- [ ] Have you reference solder joints for operator and customer comparison?
- [ ] Have you changed the frequency of solder bath analysis?
- [ ] In case of error or contamination, how will you know what/how many products have been contaminated with lead?

## Hand Soldering

- [ ] Are staff trained to clean and tin tips correctly?
- [ ] Do you switch off irons, desoldering tools during tea/lunch breaks and end of shift?
- [ ] Have you checked quality and type of plating on solder iron and desoldering tips?
- [ ] Do you have a new procedure for checking temperature on the shop floor?
- [ ] Do you have a method of identifying and isolating Pb and Pb-free solder tips?
- [ ] Have you trained your operators on the techniques and timings of Pb-free soldering?

## Inspection

- [ ] Has a reference for each solder joint type with lead-free solder been established?
- [ ] Are you setting different criteria for joint and area of coverage, IPC level three should be achievable?

## X-ray inspection

- [ ] Are you using the same criteria for x-ray as with tin/lead?

Many engineers are eager to find out more information on lead-free or have a reference book on the subject. Over the last ten years there have been many books covering lead-free and have been reviewed on the SMART Group web site. For reference here is a list of some of the most well known titles.

**Lead-Free Soldering – Jasbir Bath, Editor, Solectron** *Published by Springer Publishing*

**Lead-Free Electronic Solders – K.N. Subramanian, Editor**
A Special Issue of the Journal of Materials Science: Materials in Electronics *Published by Springer Publishing*

**Lead-Free Electronics - 2006 Edition – Edited by Sanka Ganesan & Michael Pecht** *Published by Wiley Interscience*

**Lead-Free Solder Interconnection Reliability – Edited by Dongkai Shangguan** *Published By ASM International*

**Handbook of Lead-Free Soldering Technology for Microelectronic Assemblies – Edited by Karl Puttliz and Kathleen Stalter**
*Published by Marcel Dekker*

**Implementing Lead-Free Electronics – Author: Jennie S Hwang** *Publisher: McGraw-Hill Professional Engineering*

**Lead-Fee Soldering in Electronics - Science, Technology and Environmental Impact – Edited by Katsuaki Suganuma**

**Environmental-Friendly Electronics: Lead-Free Technology – By Jennie S. Hwang** *Electrochemical Publications*

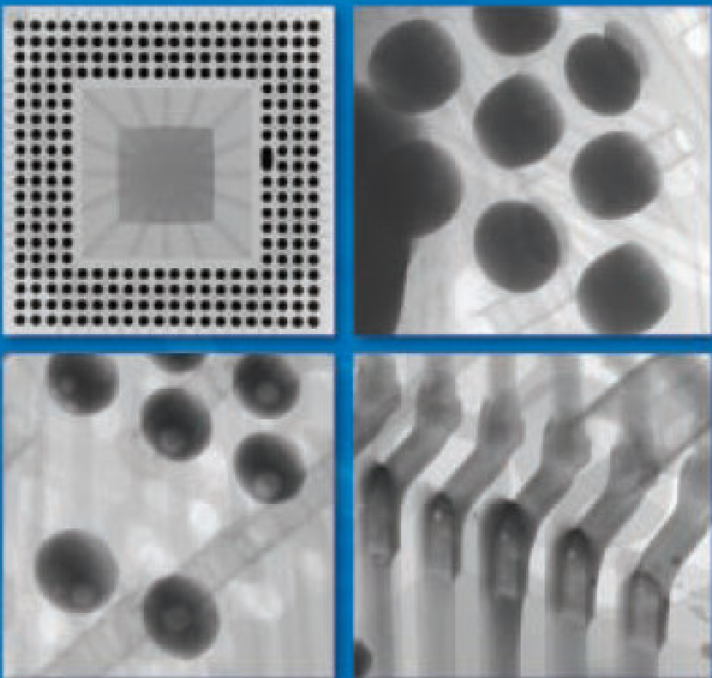
**Guide To Lead-Free Soldering for Assemblers & Sub-Contractors – by Roger Bilham**

**Lead-Free Handbook – Katsuaki Suganuma, Osaka University** *Horizon Inc., K Books Series*

**Reflow Soldering Processes and Troubleshooting: SMT, BGA, CSP and Flip Chip Technologies – By Ning-Cheng Lee**
*Published by Newnes*

**Structural Integrity and Reliability in Electronics – Enhancing Performance in a Lead-Free Environment – W.J Plumbridge, R.J. Matela and Angus Westwater** *Published by Kluwer Academic Publishers*

**Electronics Manufacturing with Lead-Free, Halogen Free & Conductive Adhesive – Authors: John Lau, Ricky Lee, Ning Cheng Lee and Wong** *Published by: McGraw-Hill*



A practical guide to

# X-RAY INSPECTION CRITERIA & COMMON DEFECT ANALYSIS

Dr David Bernard