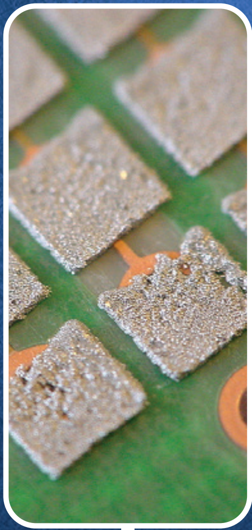
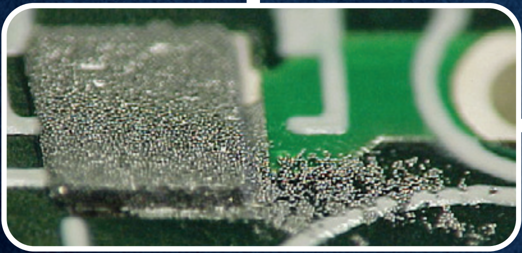
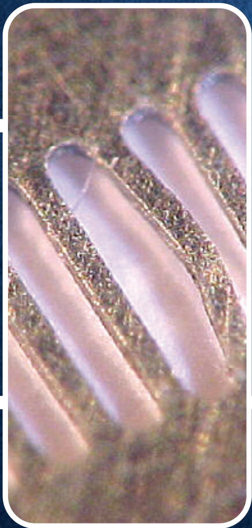
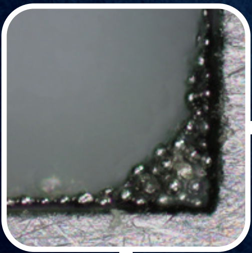
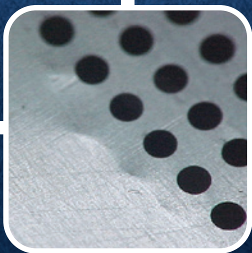
SOLDER PASTE
Print Inspection & Defect Guide

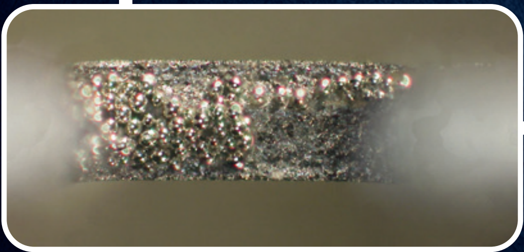


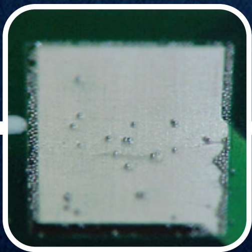


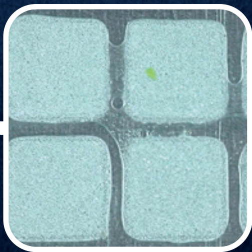


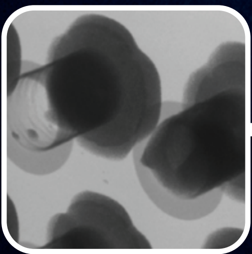












BOB WILLIS

WHAT'S NEW IN ELECTRONICS

BOB WILLIS

Solder Paste Print Inspection & Defect Guide

Contents

Introduction

This is our latest defect guide aimed at one of three fundamental processes in surface mount assembly. Solder paste printing is the first step in the process, and it is essential to achieving the highest yields. Consistent printing, placement and reflow allows reliable solder joints to be formed that easily exceed the requirements of IPC and other industry inspection criteria.

Solder paste quality and consistency have improved greatly over the years as has stencil manufacture and printer platforms to make zero defect achievable. There has also been a revolution in paste dispensing systems which, in turn, have required suppliers to develop new materials. With the increasing use of automated Solder Paste Inspection (SPI) engineers can detect variation in their process and quickly correct and avoid end of line defects. This guide focuses on solder paste defects and why they could occur plus some guidance on practical methods of inspection. Specifications and books covering printing, stencils and solder paste materials are listed for reference.

We hope newcomers to surface mount and the expert user will find this a useful guide and invite you to share the download links with your team and company colleagues to obtain their own copy plus future updates and other defect topics. Remember you can also see defects happen in real time with our unique defect videos. Any time you see My Caricature in the defects section there is a link to an online defect video to watch and share.

In conjunction with the release of this defect guide we will be presenting online webinars “Solder Paste Print Inspection & Defects – Causes & Cures” if you have missed the live event don’t worry as its available to watch with the rest of your team.

Many thanks to Claire & Rob Saunders who I have worked with on so many projects and exhibitions over many years and hope to continue to do so.

Bobwillis.co.uk

Solder Paste Storage

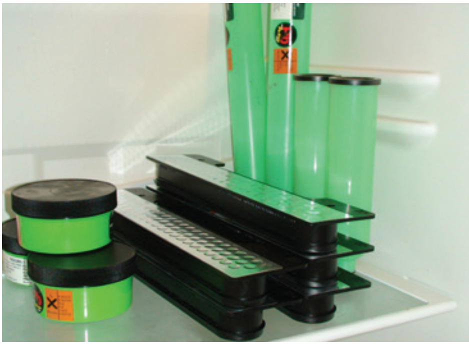
When solder paste is delivered to a customer's site suppliers normally provide guidance on the short and long term storage of paste. You should always follow the supplier's recommendation. The most common recommendation is to store paste in a fridge when removed from their original delivery packaging. Some products are specifically designed not to need cold storage on site in a factory. Always check paste specification to avoid poor performance.

Check that the paste being used is the correct product and alloy, either Lead-Free or Tin/Lead. Make sure your shop floor team are aware of the element symbols and their meaning to correctly identify incorrect materials. The supplier's packaging and logo may be the same but it is possible the material is not. A common fault is supplier's evaluation paste samples stored with production paste, you know we have all seen it. Assembly documentation file or batch travel card should always state the paste to be used

Sn = Tin	Ag = Silver
Pb = Lead	Bi = Bismuth
Cu = Copper	In = Indium

Storing paste in a fridge is common when the paste is to be stored for long periods. A guide of 2-10°C may be used as a reference but always confirm with the supplier and use in strict rotation, oldest date code first. Wherever possible solder paste cartridges should be stored vertically tip down, equally paste cassettes stored flat or as recommended by suppliers.

New stocks should be placed at the back of the fridge, so it is used in strict date code order. The date code should be checked before use. Solder paste should not be taken for use from the back of the fridge.



Solder paste should be allowed to return to ambient temperature and container should be left sealed. Paste required for next day's production should be removed from the fridge the night before to avoid delays or mistakes. Solder paste may be opened, and the seal removed when the paste has had time to reach ambient temperature overnight.

Slowly mix the solder paste before use if any separation has occurred in the container. Solder paste may be mixed on the stencil surface as an alternative to the jar or if a paste cartridge is being used in production before running pre print test cycles or test prints. Flux vehicle can separate in some solder paste materials during storage more noticeably in jars. The paste can be mixed but inspect the first few prints and the rolling action of the paste bead on the stencil surface.

Solder paste containers should be resealed after removing the required amount of paste and held at ambient temperature. The container should NOT be put back in the fridge as it will be required during production.

If solder paste is suitable for re-use remove it from the stencil surface and place it into a USED container. The paste should NOT be placed in the fresh paste container. Paste can be transferred direct to the next stencil if the quality of the printing is still satisfactory. This avoids unnecessary handling.

The USED paste should NOT be put in the fridge. Regularly check paste stock and re-order well in advance of your production requirements. It is simple to record your daily usage and scrap and good practice to forecast your production stock requirements.

Solder Paste Stencil Inspection

All new designs should be checked for manufacture, this is particularly true in contract assembly. Many cases are seen where contractors just accept the stencil files from the customer with their design. This is not good practice; fortunately many of the mistakes are picked up by stencil suppliers' front end engineers.

All new stencils should be checked prior to release to the shop floor for production. The stencils should be checked with reference to the design data used to produce the foils when first received. It is recommended that selected apertures be measured and compared with the Gerber files to check for any aperture modification previously confirmed with the supplier. Normally all stencil suppliers offer a check plot stencil file on new jobs so engineers can confirm the design.

On receipt into Goods Inwards, the stencil should be marked with the reference number, stencil thickness and issue number. This information can of course be included on the surface of the stencil by the manufacturer if required, the same as referenced on route cards or work instructions. A stencil log sheet may be attached to the stencil box for monitoring its use, age, any damage and request to re-order.

Stencil should be inspected for any obvious faults i.e. kinks, poor adhesion to the frame or mesh, missing or blocked apertures, thickness, correct orientation missing fiducial marks. The stencil should also be checked with a bare PCB. The stencil log should be completed to show condition of the stencil and used to record usage.

Stencil should be placed in store until needed for production and will be issued with the kit or taken from the stencil library. When printing has been completed and the stencil cleaned the operator will check visually. It is recommended that apertures in the four corners and the centre of the stencil be checked for correct cleaning. If the stencil has a box frame it should be checked for good adhesion of the bonding material to the foil and that the tension has not been lost allowing stencil ripples.

After cleaning check the smallest apertures and any step-down areas for paste residues. If the stencil is in poor condition the supervisor should be informed and should make a decision to scrap the stencil. If the stencil is in good condition it should be returned to the library. It is good practice to have a light box available to inspect stencils when received or after cleaning. It makes it easier to spot any blocked apertures.

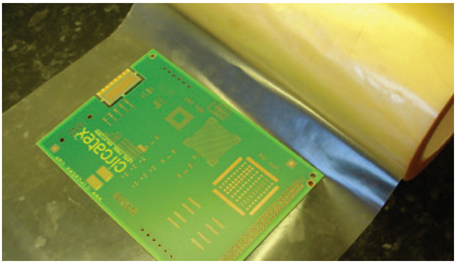
If any stencils are defective, Material Control should be notified to enable a new one to be ordered promptly. The log should be removed from the box and given to Material Control to indicate that the stencil has been scrapped.

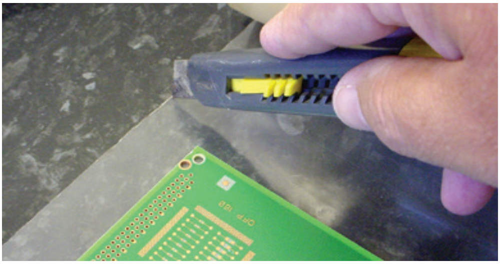
All stencils should be check prior to loading on the printer and before the application of paste. Again the apertures in the four corners and the centre of the stencil should be checked for paste residue as well as the finest pitch. It is much faster to check the stencil than load and have to remove it and the paste to start again. Any previous paste blockage will require the stencil to be re-cleaned prior to use.

Printing First Off Boards

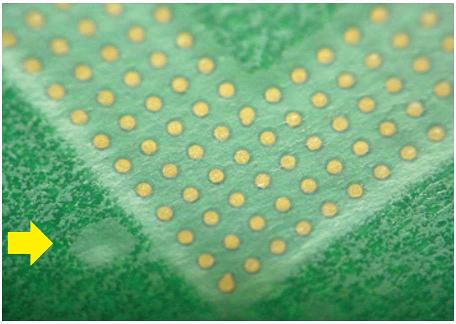
There are simple and practical ways of printing a first off without solder paste touching the board surface. Using a low tack clear film on the PCB surface allows successful printing to be confirmed prior to starting production. Alternatively, an overhead acetate foil and some magic tape can be used. The foil must be held on the PCB surface otherwise it sticks to the stencil. The ideal solution is a test board for the design being printed. If it is a gold surface finish it is much easier to use for multiple set ups, easy to clean and gives the best contrast between paste and pad

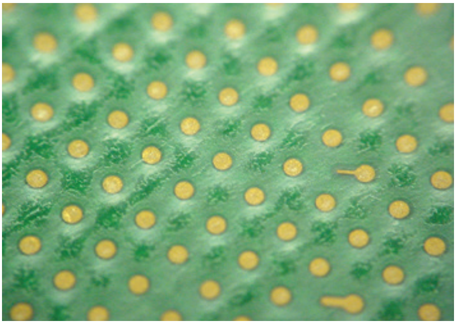
A sample board can be placed on the tacky surface of the film print face down. A knife is then used to cut and remove the overhanging film. It is important to check the film is making good contact with the board surface.

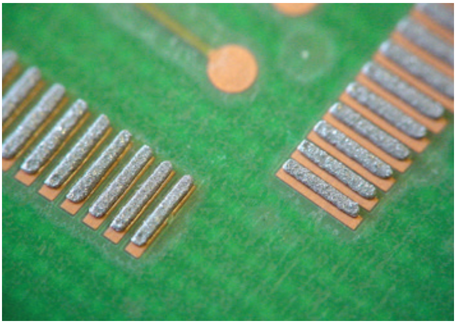




A small overhang of the film could be left in place provided it does not affect printing and automatic transfer of the board in the printer. The overhanging film can make removal after testing and inspection easier. The film should be flat on the surface of all pads with no evidence of the bubbles as this will affect printing quality.

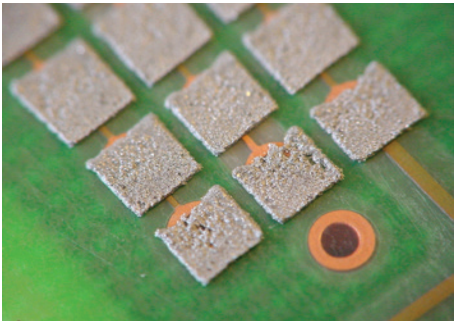


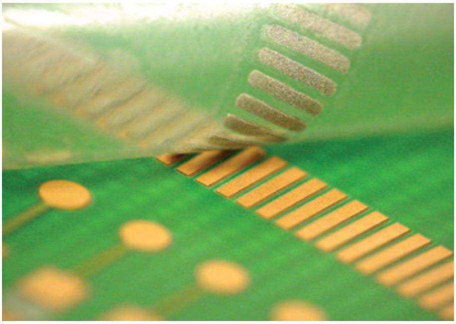




Here the paste deposit on these fine pitch pads is consistent but the stencil and the board were misaligned hence the paste is off the pad and unacceptable for a set up print

The paste deposit on these through hole pads is considered unacceptable. The print process has caused the paste to be lifted from the aperture producing inconsistent paste volume





When the printing test has been conducted and the results assessed the film can be removed from the surface of the PCB. With care test acetate print sheets can be retained for inspection and training and are very easy to photograph

Manual Solder Paste Inspection

Ideally when printing a first off test print production boards should not be used. If they are to be used to print paste direct to the board a clear film should be placed on the board. This avoids the difficulty of cleaning paste from small holes and apertures. Alternatively paste can be printed on a dedicated print test board for this purpose. The test board can be used many times provided it is cleaned properly after use or inspected as a first off. A gold/nickel surface finish gives good contrast for any form of inspection and will last longer than other finishes.

After printing this simple criteria can be used, alternatively there is criteria in IPC and NASA inspection documents.

Satisfactory Print

This is a satisfactory condition which should be achieved and used as the standard for manufacture. The solder paste deposit should equal the dimensions of the stencil, X, Y & Z plus conform to the shape of the stencil aperture design. The prints should be centrally positioned on the pad unless specifically designed with offsets.

Acceptable Print

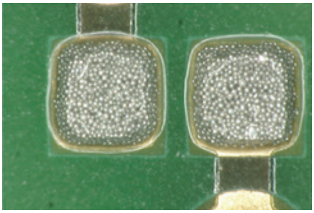
This condition represents the maximum acceptable departure from the "Satisfactory" condition. Examples within this limit of deviation will NOT require rework or washoff. Individual clarification accompanies each example illustration. Consideration should be given to modification/examination and corrective action to the printing process prior to continuing further printing. This level should only continue while investigations are in progress and not become the standard for manufacture.

Unacceptable Print

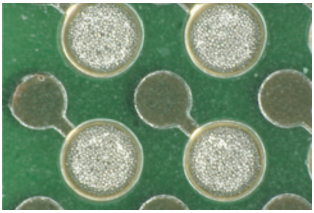
This applies to an unacceptable condition which should not be reprinted without the cause of the fault being established. Reprinting of the board may be possible after assessment of the fault and corrective action taken on the process. Any production board must be correctly cleaned prior to reprinting and should be marked on the edge of the laminate to allow future identification if required. The cleaning of any production boards should be noted on production batch documentation with the number of boards affected.

Solder Paste Inspection Criteria

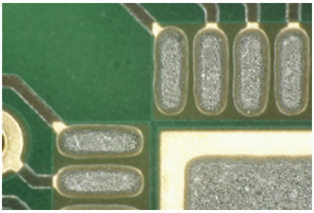
The paste deposit should be flat with no peaks and conform to the stencil aperture dimensions. It should be positioned centrally on the pad and approximately equal to the thickness of the stencil in the area of the aperture.



Chip Component

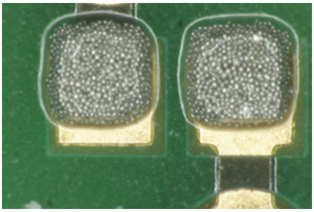


Area Array Package

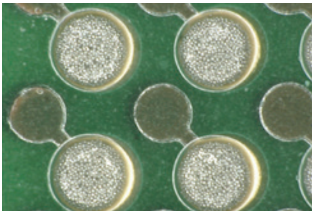


QFN Package

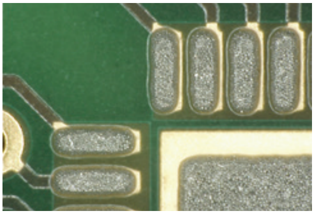
The paste deposit may be misregistered up to 20% from the pad surface provided 80% of the paste deposit is present on each pad.



Chip Component

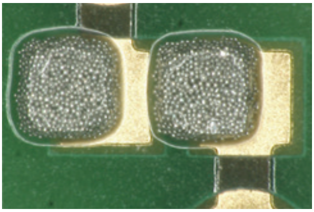


Area Array Package

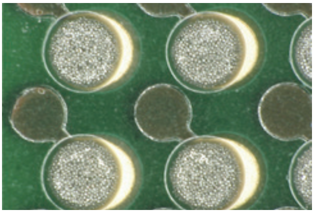


QFN Package

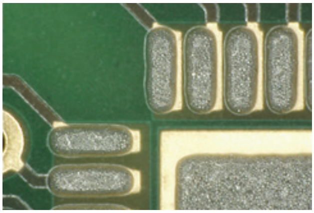
The deposit will not be acceptable if the print deposit is incomplete, or the print is mis-registered by more than 20% from the pad or wet paste shorts are visible on the boar.



Chip Component



Area Array Package

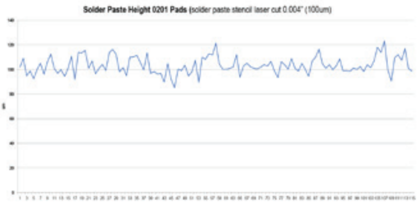


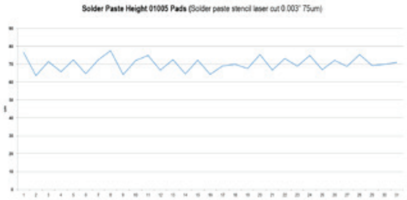
QFN Package

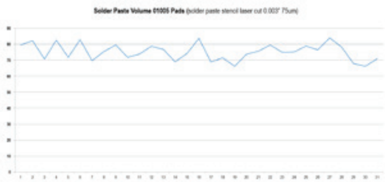
The area array paste deposit should be flat with no peaks and conform to the stencil aperture dimensions. It should be positioned centrally on the pad surface.

Automatic Solder Paste Inspection

There have been many techniques used over the years to measure paste deposits on the surface of a circuit board. Methods have now evolved to allow extremely accurate non-contact and repeatable size, volume and height measurement which is completely automatic and referred to as Solder Paste Inspection (SPI). This has become the method of choice in the industry.







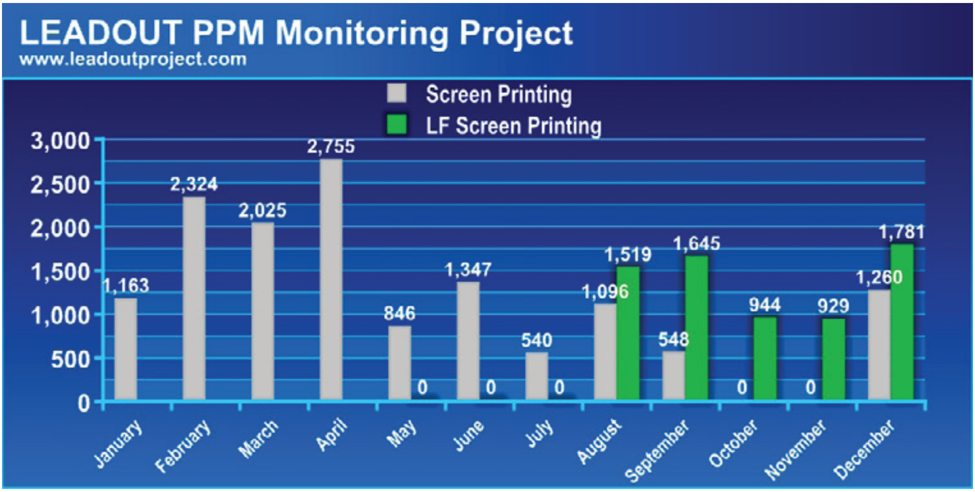
Graphs show the type of measurements taken from a SPI system.

PPM Monitoring

Counting defects manually is very time consuming but has been used in the past to provide simple comparisons based on the optical inspection guide outlined earlier. Normally speaking in PPM monitoring you would review periodically the yields from placement, reflow and printing. The following is the method used to calculate the defect level for printing, based on opportunities which are the apertures printed. A note would also be taken on the complexity level of the board design in terms of component size or pitch.

A minimum of five sample boards or panels would be taken for inspection. Any company can take a much larger sample if they wished. A minimum number of 5000 opportunities would be taken during sample inspection, this 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 in the samples.

$$\frac{\textbf{Printing Defects}}{\textbf{Opportunities}} \times \textbf{1,000 000} = \textbf{PPM Level}$$



Graph shows the results from one project using this manual method

OVERCOMING SOLDERING CHALLENGES

When looking for ways to improve production processes, solving soldering challenges can be the best place to start. Roy Goshawk, Sales Manager and Electronics Specialist at Fraser Technologies, explains how choosing the right soldering solutions for the job can have a significant impact.

Like most industrial sectors, our electronics manufacturing clients are always looking for more efficient production methods, and one way we can help them is with their soldering products.

Three of the most common issues related to soldering are: poor wettability, high levels of voiding and soldering iron tip erosion. These challenges can lead to reduced efficiency and higher costs, but there are products available that can solve all of these problems without compromising on performance.

At Fraser Technologies, we are proud to have an exclusive UK partnership with Koki – the leading global soldering solutions provider. The team at Koki has been pioneering and manufacturing cutting-edge soldering materials for over 50 years, and Koki's solder wires, pastes and other supporting materials have proven to solve these common issues, while also providing superior results to comparable products.

Improving poor wettability

If wettability is an issue, the 72M series solder wire and S3X58-M500C-7 solder pastes have a market leading powerful wetting performance. The 72M series has a new activator and a resin composition allowing better flux coverage, which can more than double soldering speed to save time and improve efficiency compared to competitors' products.

The improved wettability works well with copper, as well as nickel and brass, the latter of which are generally difficult to wet. The wire is so powerful it also prevents bridging in defect-prone conditions, such as low iron tip temperatures or fast sliding speed.

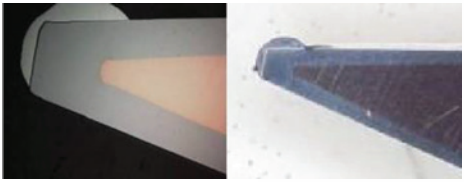
Examples of superior wetting found by using Koki M500C-7 solder paste

S3X58-M500C-7 is designed to provide superior and consistent wetting to oxidised metal surfaces, removing the oxide film at the pre-heating stage, and forming a protective film on the surface of the solder particles to effectively prevent re-oxidation during the remaining heating process.

Solving tip erosion issues

Many businesses are searching for more cost-effective solder wires. Typically, standard un-leaded higher tin containing solder alloys used for hand soldering are very aggressive and increase the erosion of soldering iron tips. As components get smaller due to design and space limitations, soldering iron tips have become more sophisticated and much more expensive.

Koki is at the forefront of R&D and technical innovation in this field and the special un-leaded alloys from Koki's 72M series of solder wires are designed to extend the life of soldering iron tips by up to four times! During the soldering process, the alloy forms a protective barrier to deter erosion. This, combined with a powerful wetting flux core, allows the operator to run the iron tip at lower temperatures, further improving the tip life, and leading to a significantly more economical manufacturing process.



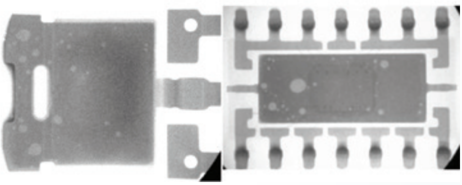
Example of how Koki's 72M solder wire creates a protective barrier on the solder iron tip, extending its life for much longer

Minimising voiding in solder joints

The formation of voids in solder joints also pose a significant challenge. Due to an increasing use of power transistor/bottom termination components in surface mount electronics, the reflow discharge of generated gas, and the subsequent voiding, is becoming a more critical issue with these types of components.

Koki's S3X58-G803, lead-free, ultra-low voiding solder paste is engineered so the oxide reduction reaction occurs before the solder melts or during the pre-heating stage. A very fast solder wetting action discharges flux gas as the solder melts, meaning almost no non-wet locations are left, no flux remains in the joint and there is no continuous outgassing, significantly reducing trapped flux and therefore voiding.

This formula ensures stable and consistently low voiding, regardless of the metallisation condition of the component or PC board; the type or shape of the components; or the reflow profiles. It also provides consistent, continual printability with fine pitch component board pad patterns. And, in terms of electronic compatibility, a heat-resistant flux activator system enables good solder coalescence with micro-components, even under harsh reflow profiles.



G803 voiding results within a solder joint

Why choose Koki products?

Koki offers the full range of printed circuit board assembly materials, leaded and unleaded solder pastes, solder wires, liquid fluxes for both wave and selective soldering, tack fluxes for repair and rework and SMT adhesives. One of the capabilities that sets Koki apart from other suppliers is that it manufactures its own solder powder, which ensures quality and consistency across products and technologies.

A simple switch can improve performance, save time, and save money, all while delivering the high quality results the electronics industry demands.

98% of Fraser Technologies' customers have seen the benefits and switched to the 72M series, and have seen impressive results from all of Koki's product range – are you ready to make the change too? To find out more about these products please visit our website **www.frasertech.co.uk**.

Tel: 01506 443058
Email: sales@frasertech.co.uk

fraser TECHNOLOGIES

Weight Gain

Adding solder paste to the PCB does add weight and measuring the board before and after printing does show a change. The digital scales used must be accurate to three decimal places to allow comparison. It is, however, more accurate to use some acetate sheets which are much lighter than a board so that the paste weight increase is more significant.

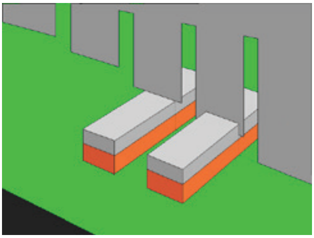
Printing to acetate sheets also allowed direct measurement on large deposits. In the early days when paste tended to dry quickly it was possible to measure the paste height with a micrometre or vernier gauge. This method is an average and does not provide the data to easily solve process problems.

Solder Paste Gauge

Early methods also included contact height measurement with a Solder Paste Gauge (SPG). This was a thin metal gauge with teeth set at different heights. These were used on test prints on the solder mask of waste sections on cut-outs of sections of boards in panels. This method is still used to measure the wet thickness of conformal coatings (The author had these etched by a stencil supplier and provided full instruction on how to measure paste).

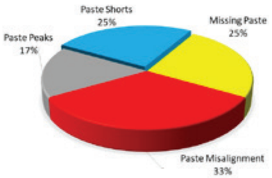
Solder Paste Printing Defects

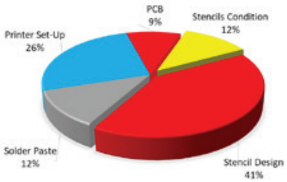
Our photographic guide provides examples of common and not so common paste and printing defects. These are not necessarily all related to the printing process, or the solder paste being used. There are many reasons why we get defects during assembly and hopefully this guide will highlight some of the causes allowing further investigations to take place to find the exact cause.



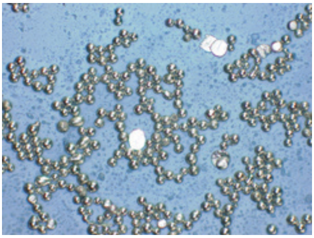
Simple method of solder paste height comparison used in the early days of SMT

Printing and inspection have become very sophisticated today with automated Solder Paste Inspection (SPI) able to inspect area of paste coverage, height and volume on every pad on every board. Previously inspection on a printer was confined to area of coverage. However, there is still a benefit and much to be learned at looking at pasted boards manually when trying to find the root cause of defects. Let the SPI machines do the hard work but please take a look at your printing closely as there is still much to learn. A past online survey by the author provided some insight to the type and possible reasons for defects. The results of future online surveys on paste and printing defects will be available online with the upcoming webinars associated with this defect guide.



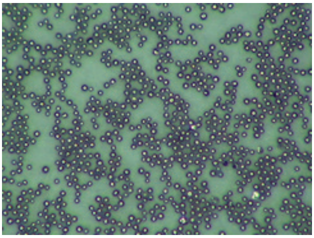


Solder Splats in Used Paste



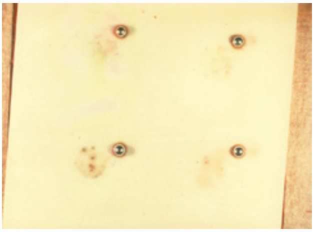
Close up of solder paste metal particles after a small sample of paste has been placed in IPA then allowed to evaporate on blotting paper. The example is very bad and the result of paste being abused with no care taken during recovery from a stencil. It is common practice to remove paste from one stencil to another or used paste container provided the paste is still printing well. When particles are compressed and deformed they can block apertures. This type of deformed particles can be caused by excess print pressure from the squeegee blade. Most suppliers will give recommendation on the tools to use during stirring and adding paste to a stencil.

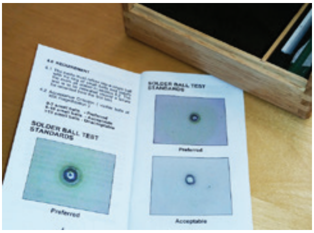
Solder Splats in New paste



Today's solder paste products are very consistent with good control of particle size, shape and distribution. Normally we would not sample paste to look at existing products unless poor printing was observed. However, when looking at new products or new alloys there is a value to use existing test methods defined in IPC or practical examination. The sample images show alloy particles from low temperate solder paste Tin/Bismuth/Silver (SnBiAg) with variation between size and a mixture of shapes. There are also some examples of compressed and deformed particles which may impact the printing process by blocking apertures. The paste sample was separated in IPA then allowed to evaporate on blotting paper before inspection and measurement of the solder powder.

Solder Balling



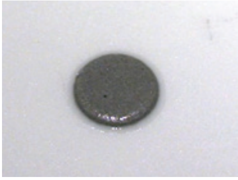
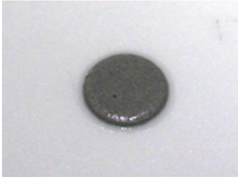


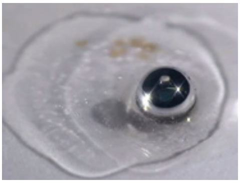
During production testing of solder paste samples it is fairly uncommon today to see solder balling on new or correctly handled products. Great care is taken by suppliers to maintain products during shipment and detailed advice is available from producers on their website. When testing solder paste in line with IPC or in-house developed tests paste should not show evidence of slump or solder balling. Please remember there are other reasons for balling not just the quality of the paste.

Example images show paste solder balling on the surface of a ceramic tile. The clean ceramic surface is used during the printing and reflow test. Many years ago one supplier provided customers test kits in lovely wooden boxes with full instructions like the one shown and loaned to the author.

Watch Bob's video

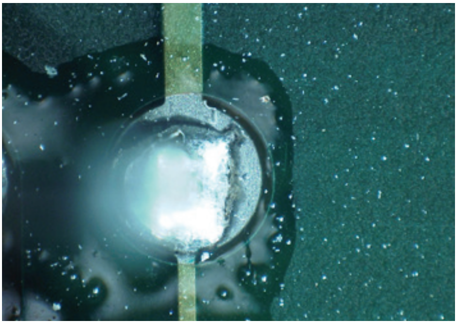
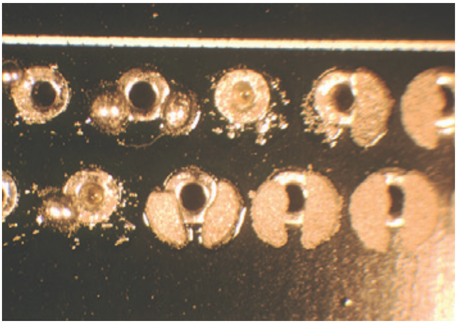
Solder Balling



Solder balling that is paste related can be seen where paste is a poor quality, incorrectly stored or slumps badly during initial preheat and reflow. All of the paste particles cannot coalesce back to a single ball which is what is expected during testing a paste on a known non-wettable ceramic surface. The images above are taken from a solder paste test where balling has not occurred, the first three images show the printed deposit, preheated and then reflowed. The last image shows the type of balling that can be experienced.

Solder Balling on Solder Mask

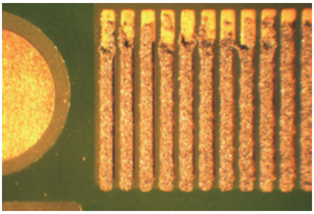



Solder paste should not ideally be printed on to solder mask, also referred to as solder resist, unless by process design. Solder mask can vary from supplier to supplier. Any company purchasing boards or any design department specifying printed boards should always specify the solder mask by type and product name. They should not just quote a generic type or specification like IPC SM840

Solder mask was not originally designed to have paste reflowed on its surface. If the process requires, as we often do in through hole or intrusive reflow, then it is important to test and evaluate both paste and solder mask compatibility to coalesce across the surface without separating into random solder balls

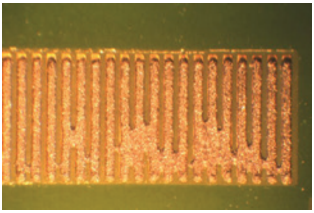
Engineers often change stencil aperture sizes on one or more pads to increase solder volume on selected joints. This is also done where there is a known coplanarity issue with corner pins on a device, connectors or area array package that warp during soldering. The increased volume of solder helps make the joints possible. A special modification and a new stencil is often a small price to pay and avoid rework.

Solder Paste Smudging



Smudging solder paste on the surface of the board is most likely to occur when handling boards or leaning over the board on a conveyor to inspect. Remember the wrist strap, cuffs on a work coat or better still the visitor/security pass on a lanyard can make a mess. This can be very embarrassing but please own up.

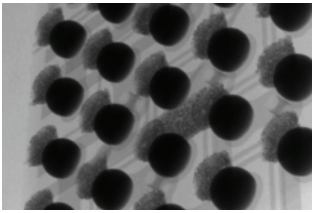
Solder Paste Shorts



A wet short or wet paste short are just terms used by the author to describe solder paste bridging the gap between two paste deposits. These may or may not lead to a solder short after reflow but the cause needs to be understood. In the example, which is severe, they are more likely to cause shorts, particularly after the fine pitch Quad Flat Pack (QFP) has been placed then reflowed.

Paste shorts like this are likely to occur when people conduct a double print operation due to the stencil lifting between cycles. It can occur when the stencil is not in contact with the board for the full print, letting the paste squeeze between stencil and substrate.

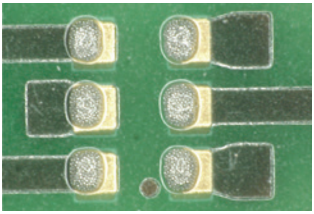
Solder Paste Wet Shorts



Solder paste wet shorts can become a solder short after reflow. Using X-ray to look at paste deposits after component placement with area array packages can be helpful. This allows any displacement and squashing of deposits to be seen. This is particularly useful when setting up new processes and a technique often used by the author on Package on Package, QFN and intrusive reflow assembly.

Paste Misalignment

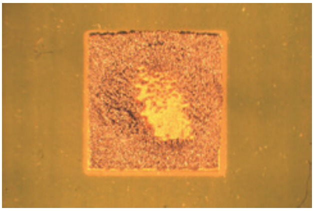
Paste misalignment on boards is not uncommon. Provided the printing process is set up correctly on smaller individual boards this should not be an issue. However with larger boards and multiple panels print misalignment between boards can be a problem, in this case we are often forced to find the best fit rather than perfect prints on each board. If the error is known and the direction of the



error the stencil suppliers can help with modifications.

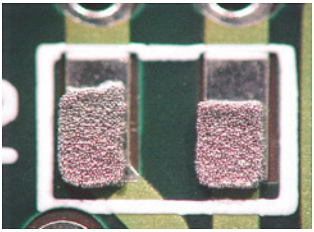
Consider checking the boards to the design information and the stencil. Check the position of the boards in the panels for any change in dimensions Your PCB and stencil supplier have the equipment to help with this process. Changes in board dimensions can occur due to first reflow, change in laminate type, break out techniques and poor etching.

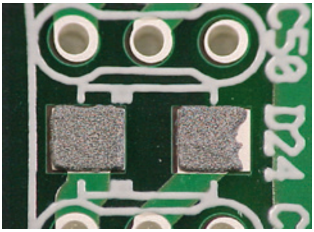
Paste Scooping

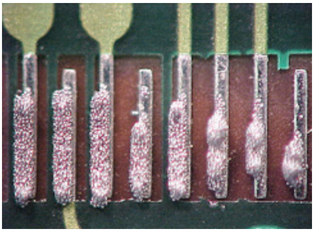


As the size of a stencil aperture increases and the pressure applied during printing increases it is possible to see solder paste being scooped off the pad. Reducing the size of the print or decreasing the pressure can overcome the issue. Where it is difficult to reduce the aperture then it should be broken up into multiple smaller apertures. This is very commonly seen on Quad Flat No Lead (QFN) and large power diodes.

Incomplete Paste Deposit

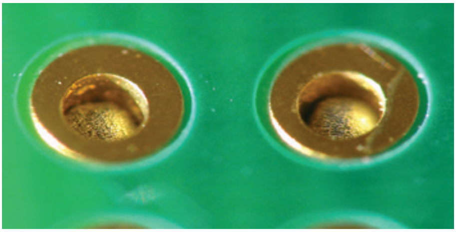


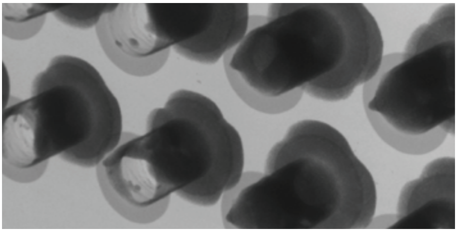




The most common reasons for incomplete paste deposits is the printing process. Incorrect amount of solder paste added to the surface of the stencil prevents the apertures filling. Too little paste on the surface of the stencil will in turn stop the paste rolling in front of the squeegee blade. It is a less common fault today but if the stencil surface does not promote the rolling action of the paste, allowing it to slide across the metal, the result will be poor aperture fill.

Incomplete Paste Fill

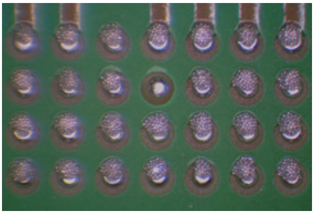




Solder paste can be printed on to the surface of Plated Through Holes (PTH) and into the barrel of a hole for a process called Pin In Hole Intrusive Reflow (PIHIR). This allows through hole components to reflow with other surface mount parts rather than using hand, wave or selective soldering.

The key feature of this process is to get consistent prints on the surface of the board and hole fill. This will provide a through hole joint to meet IPC standards and be as reliable as standard processes like wave and selective soldering. The example shows inconsistent hole fill which may result in variation in the through hole solder joints. If you have X-ray it makes through hole inspection easy for paste fill, some engineers may wonder what you are doing using X-ray for paste quality, but it can work well. If you are interested in PIHIR assembly the authors FREE book on the technology is available to download.

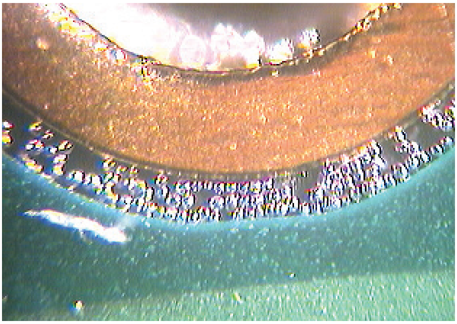
Incomplete & Missing Paste Deposit

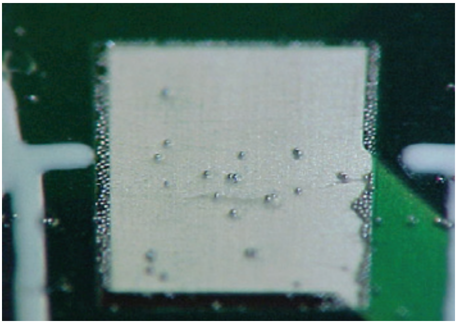


This is a great example of missing and incomplete paste deposit from a production line the author was running at an exhibition for three days. The printing problem was seen on different pastes, different stencils with manual optical inspection and highlighted by automatic SPI. There is of course a little misalignment on this example but the reflow soldering results were perfect with the exception of the missing deposits.

The simple answer was engineering error and not checking the stencil supplier's check plots. This stencil was replaced with larger area array apertures with overprinting of the pads which provided superior paste release on all pastes and stencil types. The author had a more relaxed exhibition with less process defects with a new stencil.

Solder Paste Particle Contamination





The most common reason for this problem is poor board washoff. A board has been printed with paste and due to some errors an attempt has been made to manually clean the board. Paste is forced into plated and non-plated holes and solder mask apertures around pads. You can normally always spot a board washoff when investigating soldering problems. However, all boards and panels and boards that do go through washoff should be marked for future reference. If you do allow washoffs please make sure you have a procedure and train staff.

INDIUM CORPORATION:

PROVEN MATERIALS ARE THE FOUNDATION OF PCBA RELIABILITY

At Indium Corporation, we help ensure reliability in printed circuit board assembly (PCBA) and, ultimately, consumer products. And while many factors contribute to reliability, the importance of using quality materials cannot be overlooked.

So, how do *we* deliver reliability?

- We start with quality materials and a global supply chain
- We have storage and handling guidance for every step of the soldering process based on testing and a long history with SMT products
- We provide training and technical support for key process steps including:
    - Printing
    - Dispensing
    - Stencil design
    - Reflow settings

Preventing soldering defects is fundamental to our commitment to lifetime reliability. In fact, our team of engineers is available to not only help select the best material for a product, but also to provide technical support during the process.

One of the first key defects to avoid is detectable - and preventable - prior to reflow: **Solder Paste Insufficients**. To avoid this issue, we recommend first using a high-quality solder paste such as Indium8.9HF.

Indium8.9HF delivers versatility and stability to the printing process, offering enhanced electrical reliability to ensure product life reliability and excellent response-to-pause performance and long stencil life.

These factors help prevent variability so we can help our customers and partners set up printing parameters for consistent, high-quality solder deposits.

Many common defects which appear after reflow can still be addressed by optimizing the solder paste and/or solder printing process.

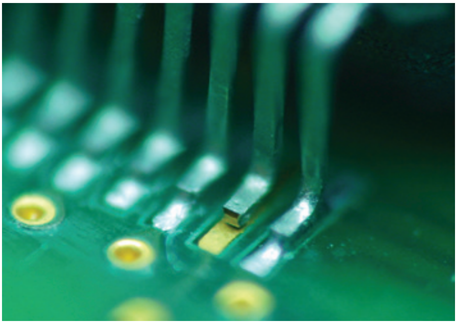
- **Solder beading** is a common defect in which a small volume of solder material does not coalesce into the solder joint. This is commonly seen on passive components with excessive solder paste deposition. This defect can often be avoided by improving stencil-to-board gasketing or aperture design during the printing process.
- **Graping** - when solder particles oxidize rather than coalesce into a joint - is more common when using finer powder sizes and can sometimes be fixed by choosing the correct flux. Graping can also be caused by solder insufficients or overly long reflow profiles.
- A common cause of **Tombstoning** - when a passive component has one end lifted off the board - is variation in solder volume. With a controlled and consistent printing process, we can reduce uneven surface tensions between solder joints during the reflow process.
- **HIP** and **NWO** are defects on BGA components in which the solder doesn’t fully connect the component to the board on some joints. Since this defect is often caused by warpage of the

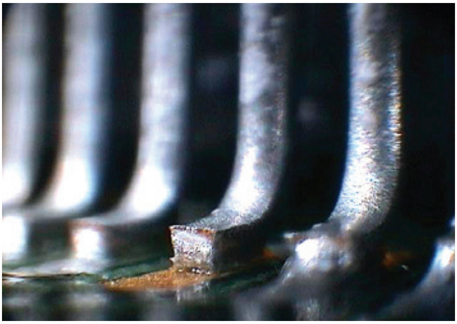
component combined with oxidation during reflow, the recommended solution is switching to a solder paste such as Indium10.8HF, **which is specially engineered to address NWOs**, rather than making modifications to the printing process.

- Although not all voids are defects, excessive **Voiding** certainly is. We have worked with many customers to **Avoid the Void®** with **our proven portfolio** of low-voiding solder pastes, carefully selected reflow profile modifications, and aperture designs.

At Indium Corporation, we believe in collaborating with our partners to inspire innovation and advance the industry - From One Engineer to Another®. Contact our experts at askus@indium.com.

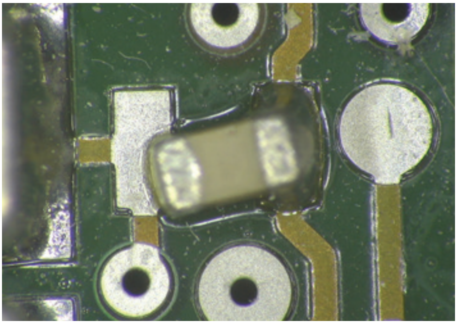
Open QFP Joint

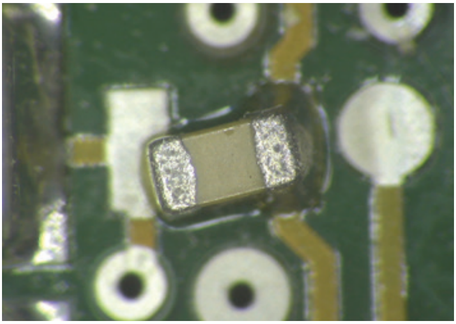




This is a clear example of an open joint but is it a solderability problem with the pad or a printing problem on the first image. Close examination shows all of the other joints are fine with paste reflowed. If there were a random issue with the gold surface paste would still reflow and the pin surface would show a buildup of solder around the pin wicking away from the pad. This is shown on the second image also with some flux residue visible. In this case it is a print aperture issue as there is no evidence of solder on the pin plating or flux on the pad.

Open Chip Capacitor Joint

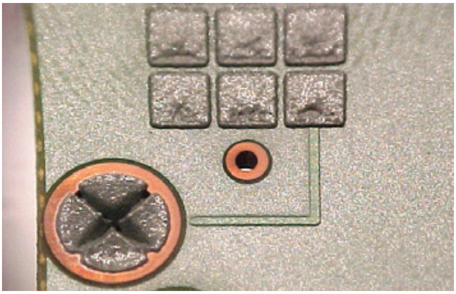


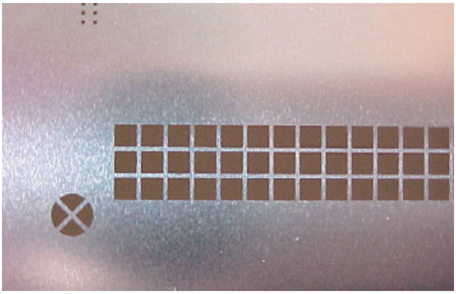


There is no evidence of solder on the pad or on the 0402 capacitor termination after reflow. If the solder paste deposit was printed successfully there should be evidence of the solder and some evidence of flux residues on or around the pad. Solder paste inspection would have captured this defect and as this is a fairly large aperture to print there should be other evidence of inconsistent printing when examining this type of error.

Paste Over Printing Compromises

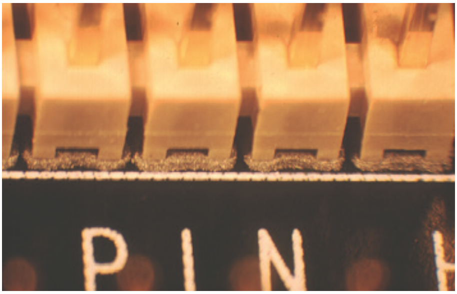
It is common practice to overprint solder paste on plated through holes to increase the volume of solder and create perfect joints in a PIHIR process. The images show the paste deposits and the stencil used to form them. Increasing paste coverage reduces the requirement to change the printing process to maximise paste filling the plated through hole and, in some cases, compromising the print process.

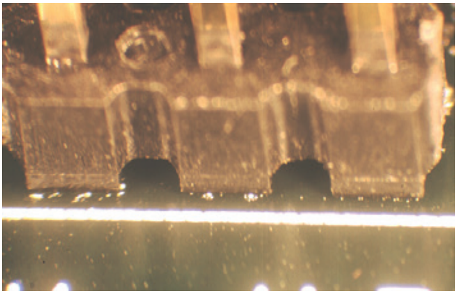




Paste is missing from one hole, but in this example is being used as local fiducial for automatic through hole insertion. The split reduced area paste deposit on the larger hole avoids contact with the body of the connector and reduces potential paste loss during preheat and final reflow.

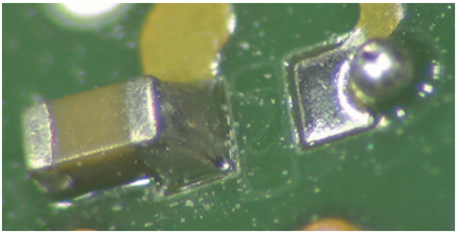
Poor Stencil Design





In through hole printing checking the connector design is essential; do not just accept the supplier's pdf design file please measure real parts, its good engineering. Always check any new or alternative parts for correct paste location between standoff points on the connector to avoid problems after printing. If your stencil supplier asks to see a new part please help him as he is trying to give you the best stencil design for your process.

Poor Stencil Design



Solder ball visible on the surface of solder mask adjacent to pads is not usual but unlikely to be as large as this or be a solder paste or printing problem. In this case the paste was printed correctly and reflow then formed a joint at one end of the capacitor but with the part flipped. It is believed that the solderability of the pad may have been an issue or component misplaced or flipped during reflow also displacing the paste deposit.

kolb CLEANING TECHNOLOGY

Solder Defects

Optimal cleaning of stencils is a Must in electronics production to assure reproducible quality and low reject numbers. Residues from SMD-adhesives and -paste particularly in the apertures of stencils / screens lead to printing mistakes in PCB assembly manufacturing and subsequently to malfunctions of the final product. Ongoing miniaturization requires ever smaller designs with ultra-fine pitch soldering and, as a consequence, ever finer stencil apertures. Micro-residues of paste and dirt in those tiny openings make error-free printing impossible.

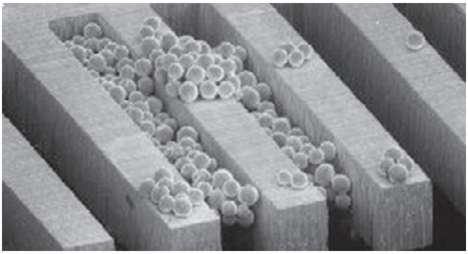
The printer-internal stencil cleaning removes paste or adhesive residues by underside wiping, using wet or dry-cleaning tissues. They primarily clean the surface but not necessarily the apertures. To achieve here the best cleaning results only special, preferably aqueous detergents should be used, which do not affect the viscosity of the solder paste.

Nonetheless, to assure high and constant reliability it is required that the stencils are regularly completely washed - if possible in an automatic stencil cleaning system.

Decisive questions when designing a cleaning process:

- Can the process clean thoroughly and material conserving fine- and ultrafine-pitch apertures?
- What exact contaminations need to be cleaned now and in a foreseeable future?
- Can the process clean fresh as well as dry / older contaminations?
- Is the whole process environmentally safe and compliant?
- What is the total cost of ownership (investment, operating cost, maintenance time)?

The governing parameters choosing a cleaning system should be **reliability, reproducibility** and **total cost of ownership**.



Technologies

Manual cleaning

Manual cleaning of stencils is still used by process technicians. However, the inherent limitations and hazards far outweigh possible benefits.

- Time consuming cleaning and drying – means low efficiency and high labor cost
- Cleanliness is highly depending on human factor and often not safely reproducible
- Spreading of solder balls and sticky residues caused by (compressed air) drying
- Cleaning operators may not be around to attend to printer problems. A printer stand-still costs more than savings due to manual cleaning gain
- High alcohol content means high evaporation and quick drying hence a higher amount is needed to achieve good results - means higher costs
- High evaporation bears health and safety risks for the operator

▸ Water based detergents gain momentum

- Low VOC content and longer impact time on the contamination surely is an advantage
- No considerable evaporation
- Lower consumption per cleaning
- Slower drying

Machine supported cleaning

The industry standard IPC recommends the use of a cleaning system instead of manual cleaning.

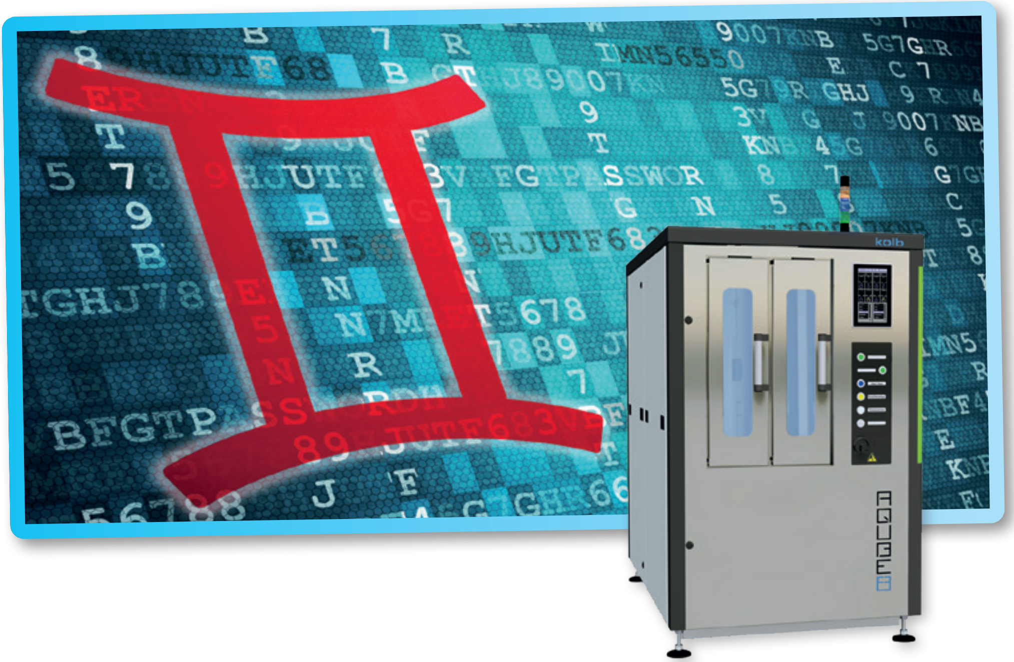
- The most popular technologies used are spray in air, air-in-immersion and ultrasonic technologies
- Each technology has different characteristics, but all machine supported cleaning is more reliable and repeatable short- and long-term
- Even small apertures are cleaned thoroughly
- Most stencil cleaners operate with a single chamber utilized for both washing and rinsing operations
- Spray or ultrasonic systems mostly offer PLC controlled, programmable process

Even if the initial investment in a machine cleaning system is higher, the total cost of ownership will be lower in the long run.

Our Gemini project:

Sequential stencil cleaning

kolb AQUBE MV8 sTWIN is the world's first stencil cleaning system with two full-fledged process chambers for parallel or sequential cleaning of screens and stencils. The only system that can start cleaning not only two stencils at the same time but one stencil immediately and sequentially a second one later.

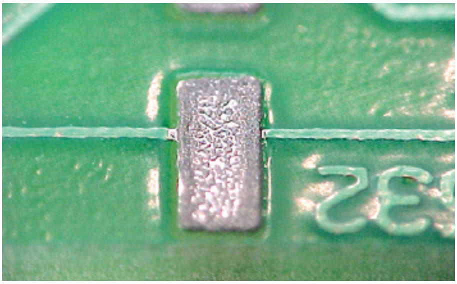


Stencil cleaning systems for all individual requirements

AQUBE® MV8 sTWIN cleans even the smallest apertures thoroughly and efficient in a short (approx. 7 min.) cycle time. However, it is only one of a whole series of screen and stencil cleaners from **kolb**. Whether small quantities or mass cleaning, entry system or industry 4.0 integradable high-end machine - we have a solution for your demands. Find out more about all our stencil cleaning products at www.kolb-ct.com.

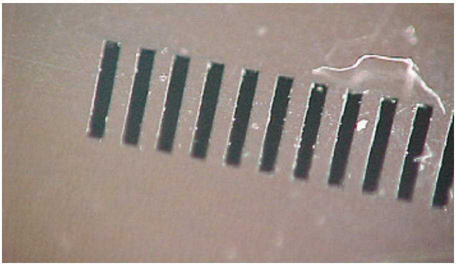
We are Process supplier: Total cleaning solutions from a single source
Partner for engineering, chemistry, process design, software and service
kolb Cleaning Technology GmbH • Karl-Arnold-Str. 12 • D-47877 Willich
Mail info@kolb-ct.com • www.kolb-ct.com

Print Thickness Variation



The solder paste deposit on the pads has been scooped, the squeegee blade has lifted the paste leaving minimum paste on the pads. This may have been exaggerated by the solder coating on the PCB. In cases where the stencil apertures are 1 – 1 with the pad dimensions the combination of solder height and blade pressure can exaggerate this issue. It is also possible that excessive copper etching and a large stencil aperture could allow the pad to sit in the aperture and reduce the thickness of the stencil for printing.

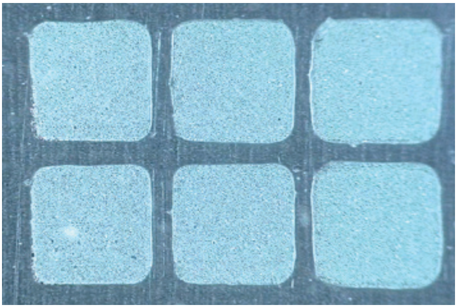
Dirty Paste Stencil



This is not the worst cleaning problem we see during factory visits but it is not ideal. Contamination of any level can lead to printing issues. Manual cleaning can achieve a higher level of cleanliness if the correct procedures are followed along with occasional process audits. Ultrasonic hand cleaning equipment is very effective with fine pitch stencils if handled correctly. Automated cleaning can achieve a high level of consistency without handling any cleaning solutions.

Stencils must be kept clean with the apertures completely clear of dried paste as this will prevent full transfer of the desired volume of paste. The stencil should also be clean and tack free to avoid hairs or other particles sticking. Fibreglass from the PCB can contaminate the paste deposit or again block apertures. Using a light box is a simple way of inspecting stencils when received or after cleaning. The process of cleaning and inspection should be fully documented and the staff trained.

Solder Paste Bleed



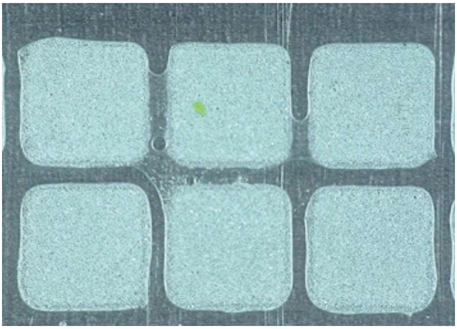
During printing it is possible for solder paste to pass between the bottom of the stencil and the PCB. This is one of the reasons we have periodic under stencil cleaning. Paste on the bottom of the stencil must be avoided as it can cause shorts through paste transfer to random locations on each board.

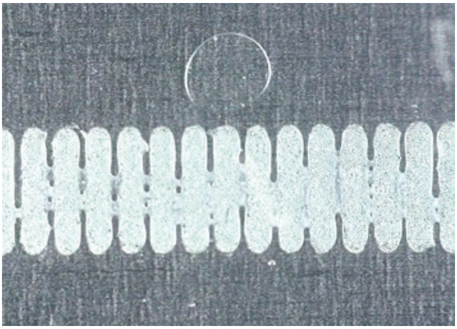
The image shows paste printed on to glass and is useful for engineers to understand the process and why certain defects occur. It is also possible to have a solder mask printed and developed on glass to make it more realistic. Bottom and side view macro video cameras have been used for many years to understand paste transfer and the impact of stencil separation after printing.

From experience we have seen the PCB surface impact the quality and consistency of printing. Variations in solder mask and legend height will lift the stencil. Reworked solder mask and identification labels will cause uneven prints as will plating or copper weight variations across the board surface.

Slow printing, much slower than recommended by the paste supplier, can cause the paste vehicle to capillary between stencil and surface.

Solvent Bleed Under Stencil



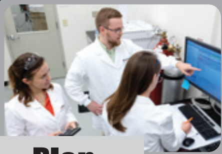


Solder pastes from different suppliers can perform in different ways. In the example paste is being printed onto glass but it is clear that this product is more prone to solvent bleeding between apertures. This can lead to solder particles also flowing out from the print area. This in turn can contaminate the underside of the stencil leading to more frequent stencil wipes.

Solder Paste Slump

Ideally when you print solder paste through a stencil the resulting print will closely match the stencil aperture dimensions X, Y and Z height. When a solder paste is heated to simulate the reflow soldering conditions there should be a minimum of paste slump, much less than illustrated in the photograph where the paste has slumped excessively, the product would not be ideal for use. Paste slump testing is documented by suppliers and featured in IPC Standards and in their Test Methods Manual. It is common practice to use this as one of the more simpler tests to compare paste products from different vendors.

5 Steps to Reducing Solder Paste Defects



Plan

cleaning BEFORE printing [1]

1

MicroCare Applications Experts[1] can help you engineer your SMT process to achieve perfectly clean solder stencils even before production begins. Choose the profile, tooling, equipment, process AND SPECIFY the cleaning options first.

Replace

abrasive cellulose-based roll wipes [1]

2

Switch to softer, durable and absorbent ***Fine Pitch Fabric Stencil Rolls***[1] from MicroCare. With no glues or binders to dissolve, they reduce FOD in your machines from high linting paper rolls.

MicroCare™

Discover Perfectly Clean

Experts say up to 65% of solder defects start with dirty stencils

Bridging, incomplete prints and paste left in apertures or vias are common defects caused by dirty stencils.

DID YOU KNOW?

MicroCare™

Get Better Stencil Cleaning at

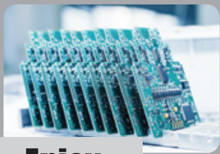
MicroCare.com

Print

more panels between cleaning cycles [1]

Clean

faster with no soaking [1]



Enjoy

the benefits of cleaner stencils

3

MicroCare Fine Pitch Fabric Stencil Rolls[1] limit dry wipe frequency, increasing your throughput and delivering more boards per hour. The softer fabric extends the life of stencil nano coatings and improve stencil aperture cleaning with fewer stencil-caused defects.

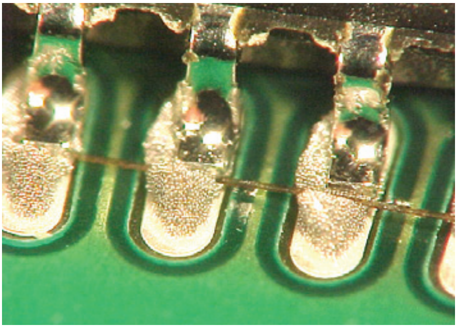
4

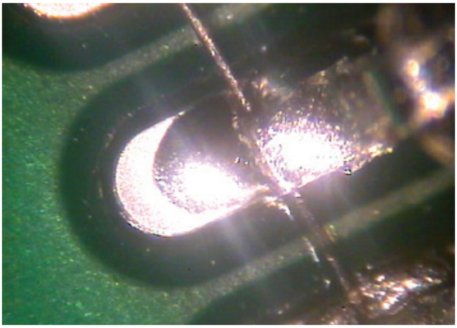
Use ***MicroCare heavy-duty presaturated wipes***[1] to clean stencils, squeegees, spatulas and tools. Choose IsoClean™ IPA-Based Wipes (BACW), ScreenClean™ Stencil & Squeegee Wipes (CDIW) or MultiClean™ MultiTask Surface Cleaner Wipes (MLCW). All are no-rip, non-linting and ESD-safe.

5

Improved print quality directly translates to fewer defects, less rework and fewer scrapped PCBs. Produce more boards-per-day at a ***lower cost-per-board*** to save time and money.

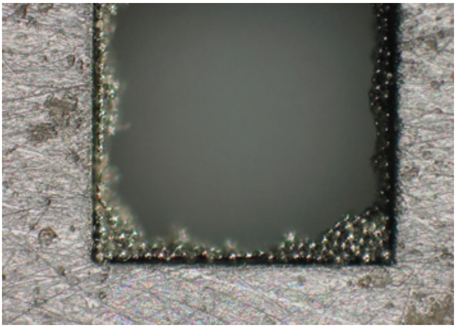
PCB Fibre Contamination

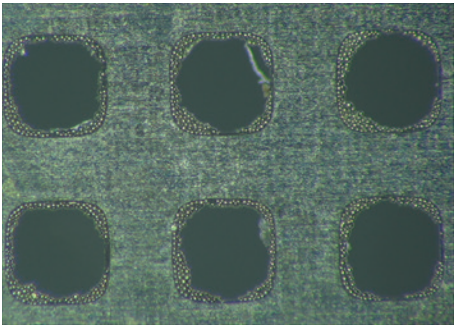




Contamination does occur from PCB glass fibres and due to airborne particles which can be trapped by the tacky nature of solder paste. The example shows a fibre on the surface of the paste after placement. The resulting solder joint was not affected by the fibre, but all contamination should be avoided wherever possible. It is one of the reasons that the automotive industry have been very keen on preprint board cleaning as this type of contamination would not be acceptable.

Poor Cleaning of Apertures

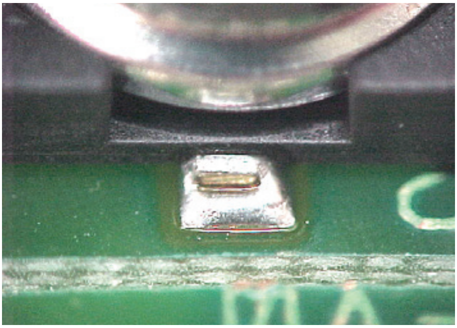




If the stencil is not cleaned correctly paste will remain in the apertures and dry in place. When the stencil is used again then it will be very difficult to obtain successful prints. Having dried paste on the same pad as a limited deposit of fresh paste is also less likely to provide a perfect joint. It will be very difficult, if not impossible, to fault find after reflow.

Any stencil must be cleaned after use and inspected before returning to the stencil library. A stencil should always be checked before mounting on to a printer. A lot of time can be wasted if the printer is set up then has to be broken down again due to dirty stencils.

Component Lift

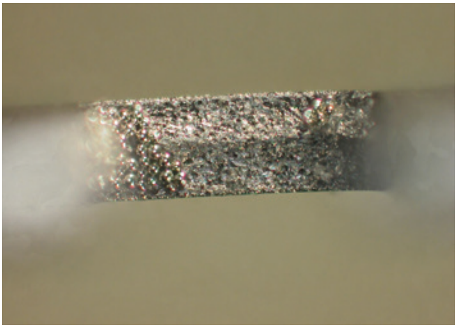


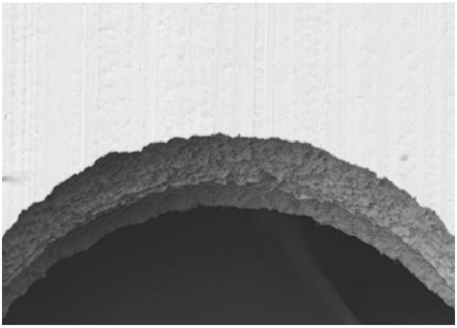
In most cases component lifting during reflow would be blamed on the reflow process. However, in this case a combination of stencil and pad design have caused the component to lift on the solder as it reflows forming a bump. The pad width is wider than the gap in the plastic standoff body of the component. The incorrect pad design has been used on this part, leading to a wider stencil aperture. To be fair there may have been a change in the component supplier but if that is the case the new part should have been checked with the board design before accepting new stock. Alternatively the engineer should have reduced the stencil width and increased its length to maintain paste volume for the joint but not lifted the part.

Stencil Manufacturing Quality

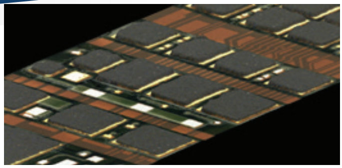
There are many very good stencil producers in our industry offering etched, laser-cut and electro formed stencils. Each supplier may also offer other proprietary processes and treatments which may enhance paste printing and release from each aperture. It is up to the customer to work with and evaluate sample stencils. Very few companies have the resources to evaluate a stencil in detail. A side by side comparison on a demanding board is the simplest way. Then compare the ongoing performance of the supplier based on yield, delivery, support and finally cost.

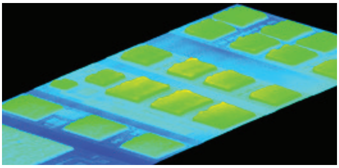
The image shows a stencil aperture which is etched but has a very uneven wall surface, there is also some evidence of the paste still present in the opening. The second SEM image is also etched but with a superior surface and provided a good performance.





Fast, Accurate 3D Solder Paste Inspection for Multiple Applications | Case Study





Benefit Summary

Flexibility tops the list of benefits as the SQ3000™ proves to be the most accurate and reliable solution for both AOI and SPI applications.

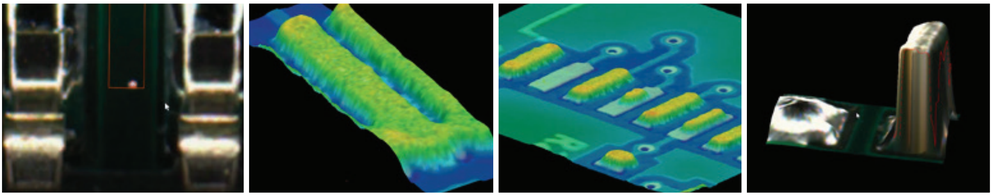
Challenge

Our customers were in search of a single solution that could handle both Automated Optical Inspection (AOI) and Solder Paste Inspection (SPI). A flexible and robust system, would allow them to accurately and repeatedly measure both small, short components (~50µm) and large, tall components (<25 mm). The ability to manage both inspection cycles with one system would greatly reduce cost, training and operator time, and minimize the required footprint.

Solution

CyberOptics' SQ3000 is the ideal inspection solution for these customers. This Multi-Function system has the flexibility required for a variety of applications, including AOI, SPI and CMM. The sensor, software and system all work together to completely automate the full-line inspection process. With high precision, customers are able to utilize this solution without changing process and production based on layout for features like plot line design. The SQ3000 offers repeatability of 6 micrometers in 3 sigma for X, Y, Z measurement.

The Multi-Reflection Suppression™ (MRS™) sensor technology delivers unmatched accuracy by identifying and rejecting reflection based distortions caused by shiny components and surfaces. With multiple MRS sensors available, with various speeds and resolutions, our customers are able to find defects sooner and mitigate any measurement inaccuracies for a variety of AOI and SPI applications, helping reduce cost and operator time. Additional advantages of the SQ3000 include ease of use and reduced training time.



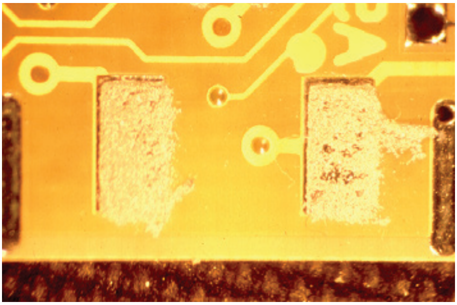
This proprietary system is a best-in-class solution that significantly improves yields and processes, and continues to create operational efficiencies for our customers.

For more information about CyberOptics, visit our website at www.CyberOptics.com

Poor Stencil Cleaning

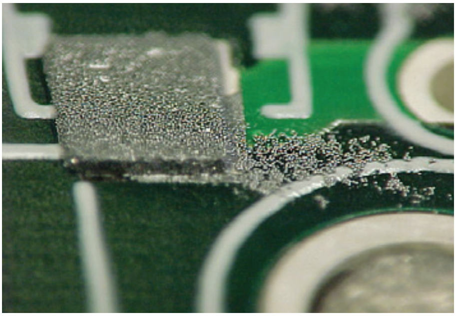
Cleaning stencils can be conducted manually or automatically to a very high standard. The first thing that needs to be established is the cleaning material to be used is compatible with the solder paste being cleaned. The cleaner must be able to dissolve the residues, the flux/vehicle must be soluble in the cleaner and the cleaning process must be able to clean and then rinse the stencil before drying. The surface of all apertures must be clean with no residue or solder particles left behind. The complete surface of the stencil must be tack free. The image shows a dirty stencil with streaks and residues and it was tacky to the touch making it difficult to remove from its shop floor protective packaging.

Incomplete Print



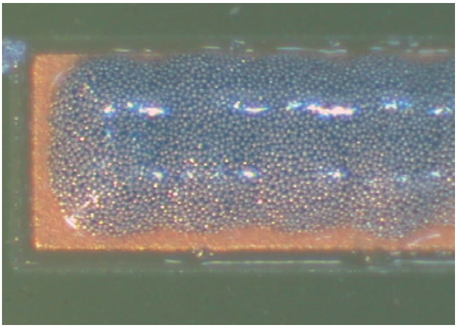
There are a number of reasons for incomplete and smeared prints. This example was caused by movement of the board during separation of the stencil. The board was not being help by the clamping system. The board could be seen to move as the pressure of the squeegee blade was released. This was also indicated by the lifting of the stencil.

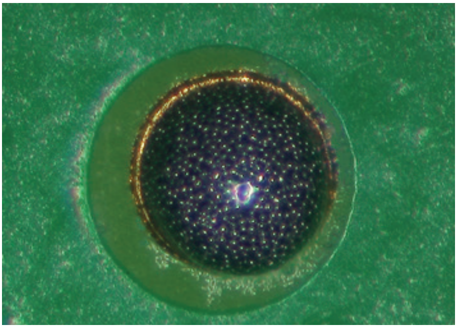
Solder Paste Bleed



This is a less common problem today but paste is contaminating the board surface away from the pad being printed. Where there are through holes and vacuum tooling used to hold boards in position problems can be experienced. Literally the paste is being sucked off the pad surface. This was also a problem with hybrid circuits before surface mount technology.

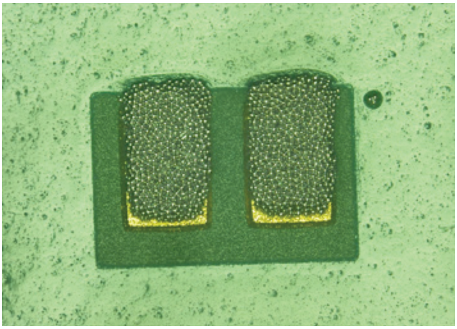
Wet Paste Deposit

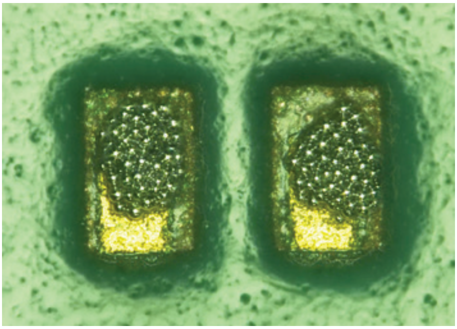




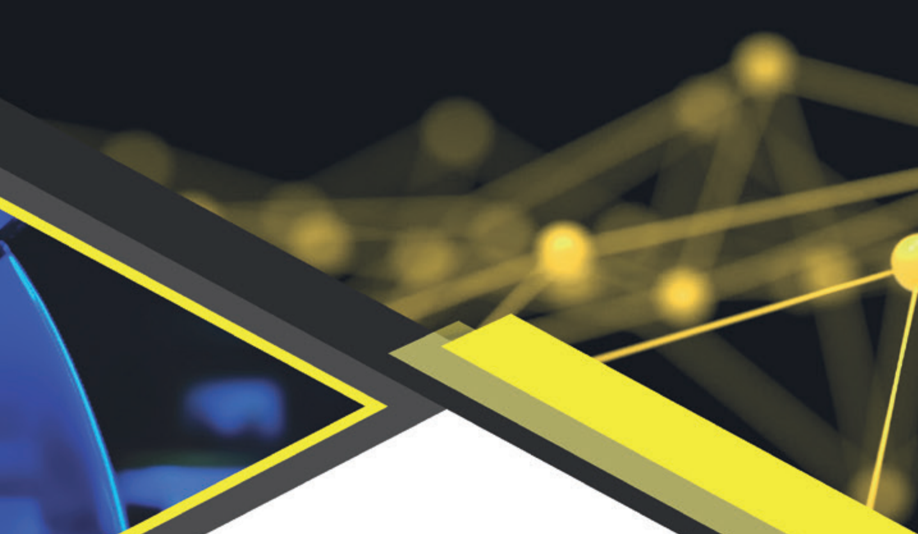
When looking at solder paste deposits they often look dry on the surface of the pad after application. This example looks different. Well it is different and it has not been printed; it has been jetted on to the surface of the pad. The main reason for the difference in appearance is the paste. When products are designed for dispensing or jetting they generally have a lower metal content and different viscosity. There is nothing wrong with either of the examples above it is just they are different, and both helped formulations from suppliers to create perfect solder joints. The dispensing and jetting paste grades still must perform correctly when soldered. It is just another challenge for the paste developers.

Solder Paste Alignment





Printing ultra-fine pitch and small passive components 01005 and below is a challenge and it is all about fighting tolerances. Circuit boards expand and contract during soldering. There are differences in the relative position of the boards in multi panels and stencils.



BETTER BY DESIGN

A NEW RANGE OF CONFORMAL COATINGS 'PROCESS-FRIENDLY, LOW TOXICITY'.

Each new coating takes the performance advantage of HumiSeal's Industry-standard materials, and enhances them with processing and application improvements. Low toxicity solvents have been introduced across the entire range to eliminate aromatic hydrocarbons such as toluene and xylene. All products provide excellent moisture protection including chemical and corrosive environment resistance.

1R32A2 Range
Solvent-based modified acrylic with ease of repair, excellent physical and electrical properties, excellent adhesion to cleaned and no-clean substrates. - UL94-V0 rated.

1B73 Range
High performance solvent-based acrylic with improved temperature stability at 150ºC and excellent insulation properties. UL746E 94-V0 rated.

1A33 LOS & LOF
High performance Polyurethane coating with excellent chemical and physical protection. Low odour and fast curing options. - UL746E 94-V0 rated.

1B51 / 1B59
Industry leading Synthetic Rubber with excellent flexibility, thermal shock and moisture protection. Increased thermal stability to 150ºC - UL94-V0 rated.

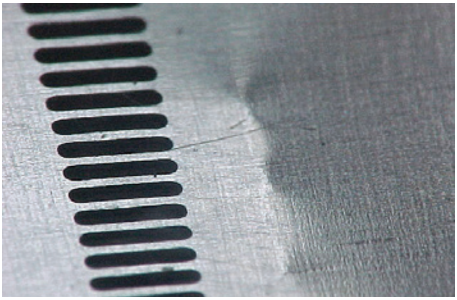
"

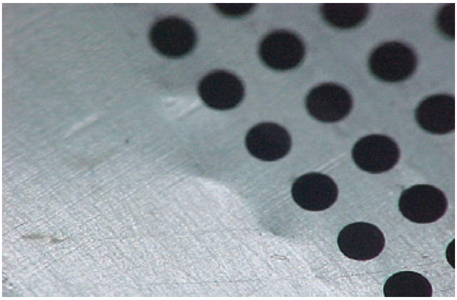
Film Coater Friendly

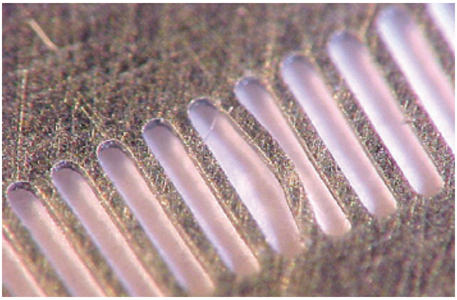
Designed specifically without Toluene or Xylene . Take your process to the next level

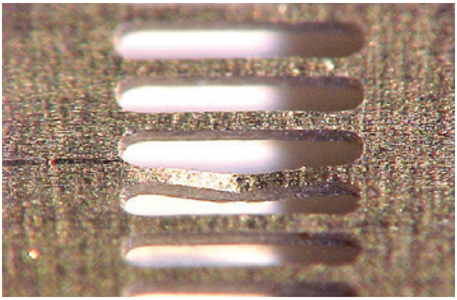
"

Solder Paste Stencil Damage





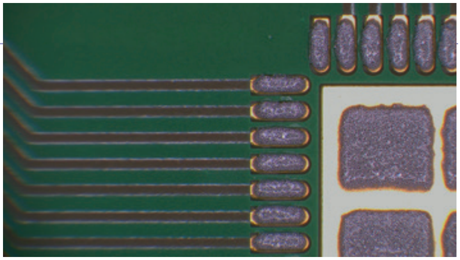




Solder paste stencils are a precision tool that must be handled, used, cleaned and stored correctly following supplier recommendations. If the stencil becomes damaged the print quality will suffer. We are very fortunate today that stencils are more economically priced and quick to turn around replacement stencils. Back in the day a laser cut stencil could cost over £1000.

The examples above all show damage to the foils which will cause print errors and should be replaced. With decreasing stencil thickness from 0.006” – 0.003” (150 - 75um) damage can occur. Most stencil supplier offer fast turnround times, you pay a little more but if you have to spend more then hopefully more care will be taken with these tools in the future. Many years ago it was common to have back up stencils and this is still possible as stencil foils take up so little room in high volume. However, fast manufacture and delivery is readily available so the need for back up stencils are not necessary.

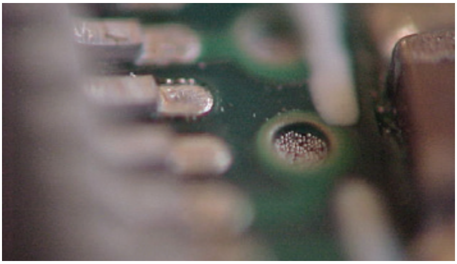
Solder Paste Deposit

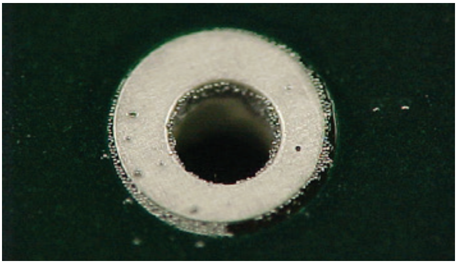


Just to round off the defect section here is a perfect example of paste deposits on a QFN footprint. The side termination pads have consistent paste coverage and volume. The centre pad has nine split aperture designs to avoid excess paste, voiding and paste displacement during component placement.

Solder Paste Washoffs

Although every effort is made to eliminate printing problems, they do occur. Some boards do require cleaning if the required standard of paste printing is not achieved. If this is the case a method needs to be defined on the shop floor and monitored for its effectiveness unless you want to have boards like the examples shown below?

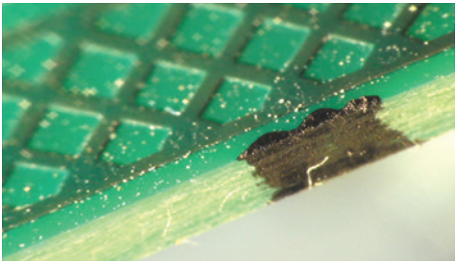


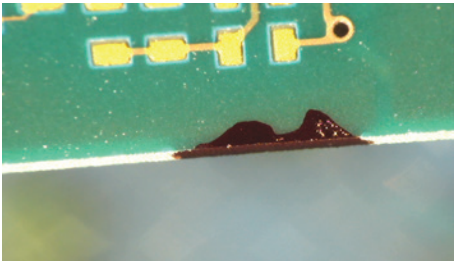


The most common process is to wipe the paste off the board with a cloth or rubber blade which removes the paste from the surface of the pads but deposits it everywhere else. Wiping paste manually places paste in the resist windows, via holes, tooling holes and plated through holes and may not be removed with subsequent cleaning operations. Process problems may be experienced during through hole component insertion as the paste will reflow partially filling the holes. As the fluxing agent in the paste has been removed during cleaning the solder particles normally just bind together in the holes like frogspawn. If the paste is left on the surface of the board the cleaning system will have to handle more paste removal; both situations can be evaluated during testing.

Although the cleanliness of the printed board after cleaning is very important, testing for ionic residues is only one issue to consider. The impact on solderability of the surface finish and the solder particles on the surface of the board, and in the through hole, is also very important to monitor. The time taken to clean the boards manually or semi automatically is very important to the solderability of the boards. How easy is it to put boards into a cleaning tank and totally forget they were there, we have all done it??

Final inspection will undoubtedly detect solder balls on the surface of the board after final assembly. On many occasions the solder paste is blamed for poor reflow and a debate ensues. A simple trick for any boards that are washed off by whatever means is to mark the edge of the card with a wide felt tip pen.





In the case of a multi panel each board, there is no point in marking the scrap breakout sections for investigation. Marking the edge makes it quite clear that the board has gone through a different process and may be considered during defect investigation. Of course we assume that washoffs are also recorded in the batch documentation.

The practice of washing off boards should be defined and a process available to conduct the operation with suitable training for staff. The number of washed off boards should also be monitored as part of a process control strategy to provide an indication when a process is drifting. Cleaning and drying boards prior to re-printing is important to consider. If the cleaning material is not fully evaporated from the surface of the board or solder mask it will affect satisfactory reflow of the solder paste. Often this looks like incompletely reflowed paste. The author has been in some factories where the pile of boards for washoff was nearly as high as the printer. This was of course quite a few years back, the goal is never to create defects and have to wash boards with paste.

International Standards & Reference Guides

Stencil Design Guidelines - IPC-7525A
Stencil & Misprinted Board Cleaning Handbook - IPC-7526
Requirements for Solder Paste Printing - IPC-7527
General Requirements & Test Methods for Soldering Paste - J-STD-005
NASA Workmanship Standards - Solder Paste Printing
Bob Willis Solder Paste Printing & Inspection Posters
Bob Willis Solder Paste, Printing, Inspection & Defect Photo Library

Technical Books on Solder Paste & Printing

Reflow Soldering & Process Troubleshooting
Dr Ning-Cheng Lee

Solder Paste in Electronics Packaging: Technology and Applications in Surface Mount
by Jennie S. Hwang

Solder Paste Technology Principals & Applications
by Colin Johnson

Guide to PIHR Technology – Design, Assembly, Inspection & Defects
by Bob Willis

Authors Profile

Bob Willis currently operates a training and consultancy business based in United Kingdom and has created one of the largest collections of interactive training material in the industry. He is a member of the SMTA Europe Technical Committee. Over the years Bob has been Chairman and Technical Director of the SMART Group and held the title of Honorary Life Vice President for his contributions to the Group since its inception. With his online training webinars Bob Willis provides a cost-effective solution to training worldwide and regularly runs training for SMTA, IPC and in the past ICT & EIPC. Although a specialist for companies implementing lead-free manufacture Bob has provided worldwide consultancy in most areas of electronic manufacture over the last 35 years. This earned him the SOLDERTEC/Tin Technology Global Lead-Free Award for his contribution to the industry. Bob has travelled in the United States, Japan, China, New Zealand, Australia, South Africa and the Far East consulting and lecturing on electronic assembly.

Bob was presented with the "Paul Eisler award by the IMF (Institute of Metal Finishing)" for the best technical paper during their technical programmes. He has conducted SMT Training programs for Texas Instruments and ran Wave & Reflow Soldering Workshops in Europe for one of the largest suppliers of capital equipment Electrovert/Speedline. This is based on many years of practical experience working in telecommunications, military OEM, contract assembly, printed board manufacture, environmental test and quality control laboratories. He has also been presented with the SMTA International Leadership Award and IPC Committee Award for contribution to their standards activity.

He has also run training workshops with research groups like ITTF, SINTEF, NPL & IVF in Europe. Bob has organised and run lead-free production lines at international exhibitions Productronica, Hanover Fair in Germany. Nepcon Electronics in England plus IPC APEX and SMTA International in USA providing an insight to the practical use of lead-free soldering, high temperature electronics, cleaning, conformal coating on Ball Grid Array (BGA), Chip Scale Package (CSP), 0210 - 01005 chips and through hole intrusive reflow assemblies. Bob has also been presented with a Best Speaker at SMTA International Conference in Chicago. In his early career he worked with the GEC Technical Directorate as Surface Mount Co-Ordinator for both the Marconi and GEC group of companies and prior to that he was Senior Process Control Engineer with Marconi Communication Systems.

Bob Willis Solder Paste Defect Videos

Here are some of our video clips showing common and less common problems with the use of solder paste. We have created and used these types of clips for many years to help engineers understand what can happen and to investigate problems for customers.

Poor Paste Volume and Rolling Action

Excessive Under Stencil Cleaning

Poor Paste Seperation

Solder Balling & Solder Paste Fines

Solder Beading & Poor Design Rules for Pads

Solder Beading Due to Solder Paste Slump

Solder Paste Stencil Contamination

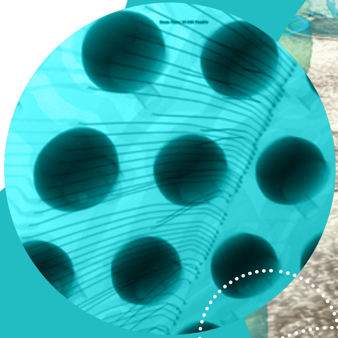
Solder Balls & Solder Fines

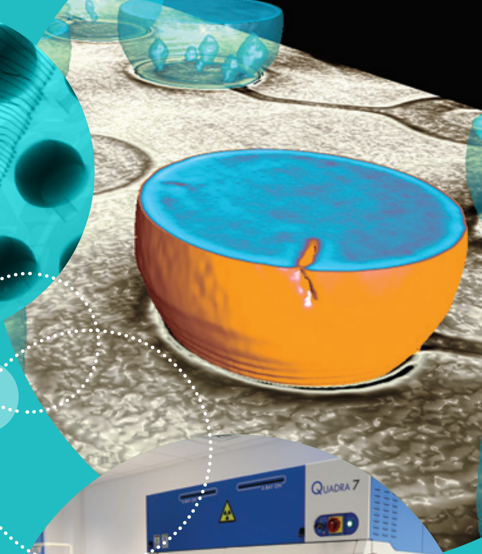
Excess Stencil Solvent Cleaning

Voiding in Pin In Paste Reflow

Solder Paste Spitting & Condensation

Cupio SERVICES





THE PLACE FOR

Failure Analysis and In-depth Non-Destructive Inspection

Cupio Services brings knowledge and expertise gained through many years of working with some of the world's best X-Ray, test and AOI systems to our new facility in Chineham Business Park, Basingstoke.

Whether you need to diagnose production or in field failures, validate complex manufacturing procedures or perfect new product introductions, we're here to help.

We have a variety of cutting-edge inspection and test equipment on hand, including functional testers and counterfeit part detection, high resolution x-ray with multiple CT imaging options, and acoustic microscopes for delamination detection within boards or complex devices.

We have the capability to find the smallest defects within your samples and image them with sub-micron resolution. Full failure analysis reports can be provided to help you understand where and how issues are occurring.

Cupio Services Ltd

+44 (0) 1256 262800 info@cupioservices.co.uk www.cupioservices.co.uk

Beechwood, Chineham Business Park, Basingstoke, Hampshire, RG24 8WA