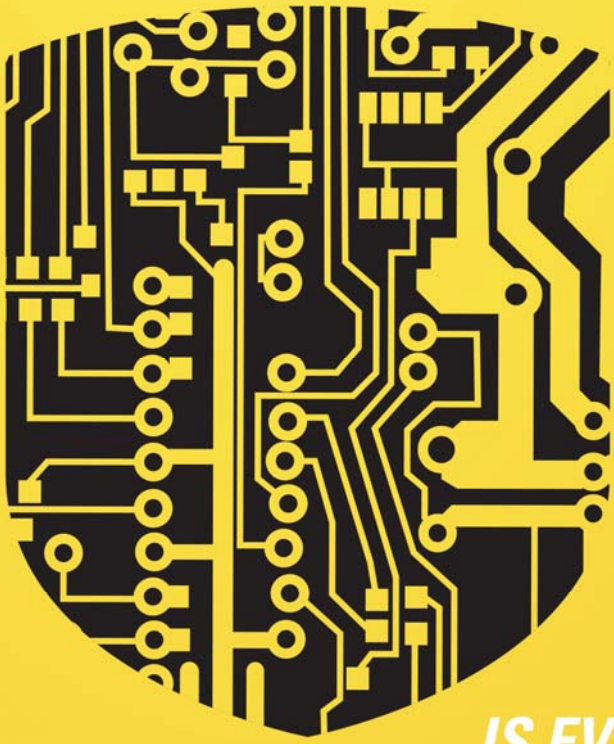
Guide to Conformal Coating & Cleaning Defects 2

Protection



IS EVERYTHING

WHEN IT COMES TO PROTECTING ELECTRONIC ASSEMBLIES AND IMPROVING RELIABILITY, TRUST IS KEY.

HumiSeal prides itself on being the only company in the market whose sole business is the development and supply of protective materials for electronic circuits.

This single-minded focus enables us to work as a partner with our customers, developing products that are tailored to exact requirements..

With excellent protection from moisture and condensation as well as airborne contamination and impurities you can be sure that assemblies are protected from even the most severe operating conditions.

HumiSeal®

CHASE Electronic Coatings
We make a material difference

HumiSeal Europe
505 Eskdale Road, Winnersh
Wokingham • Berkshire • RG41 5TU • United Kingdom

Tel: +44 (0)1189 442 333 • Email: **europeansales@chasecorp.com**

www.humiseal.com

Contents

Copyright and Disclaimer

Introduction

Welcome to this SMART Group Guide

This, the latest guide, is aimed at an often forgotten and neglected subject

Conformal coating is essential for many products that are required to undergo long term storage or will be subjected to harsh environments during their life cycle, yet because the process is applied at the end of the assembly it is often overlooked or misunderstood when it comes to DFM aspects such as masking areas, realistic keep out zones, inspection etc. and forgotten in relation to a test strategy. The increased use of coatings has also focused the old question of clean or no clean?

The SMART Group is looking to aid its members, affiliates and supporters with some of these aspects by providing a concise and highly visual guide to the basics of coating and cleaning defects to help with diagnosing some areas of concern and/or debate you may come across

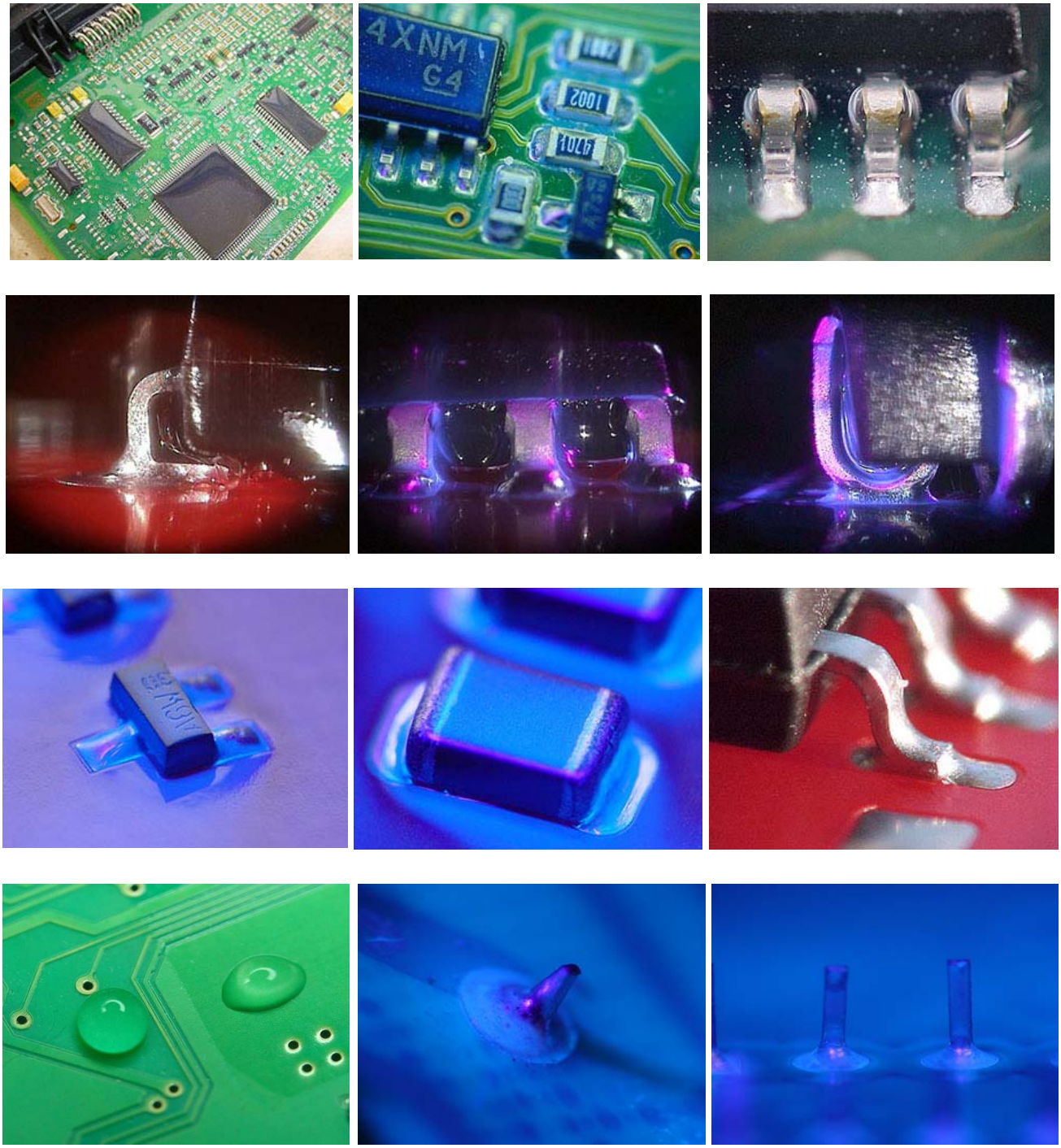
The guide has been written to aid the shop floor production team with identifying acceptable from unacceptable, to help the engineering team to understand the processes involved and to diagnose and find solutions for issues encountered, for the designer it can provide an introduction to what conformal coat is and how a good design can influence the ease and success of application.

With information to help both the novice and the expert user we hope you find this a useful guide and invite you to share some SMART Group knowledge with your own team, your customers and suppliers. If you have any other defect types relating to coating or cleaning let our Committee know and we can add the images and your analysis of the defects technical@smartgroup.org

Nigel Burtt
SMART Group Technical Chairman

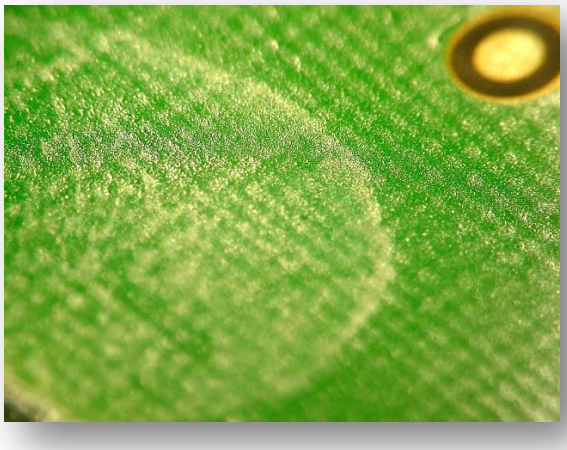
Conformal Coating Reference Joints

The following example images are typical of boards and solder joints that have been conformally coated with different types of protective coatings. The examples are shown under ambient and UV lighting at different magnifications



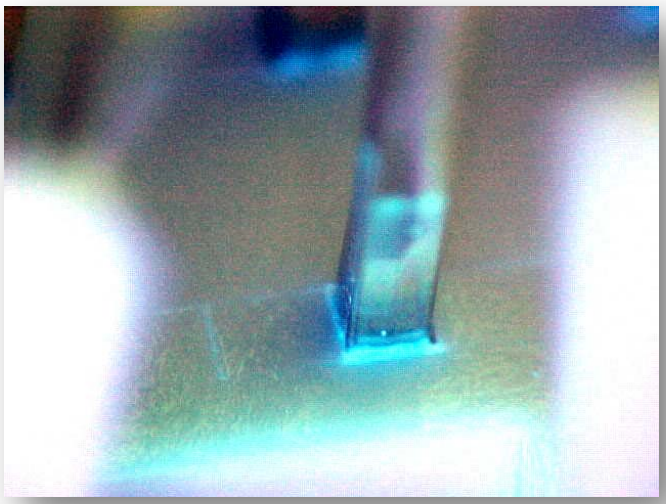
Conformal Coating & Cleaning Defects

Human Finger Print



This is very obviously poor handling of the board after coating or for some reason, the coating has not cured prior to inspection and may need to be investigated. It would be possible to call in CSI and finger print all the engineering and quality team to find out who has been handling the board!!

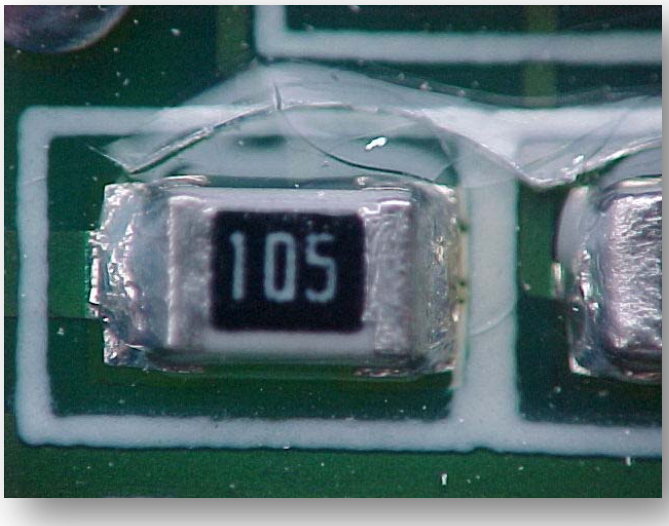
Connector Pin Contamination



Conformal coating can penetrate a connector body by capillary action and is dependent on the connector design, the coating material's solids content and the method of masking. Ideally masking is avoided when using robotic spray coating equipment provided the method or direction of application is carefully considered. On many connector design pins float in the body allowing for alignment with mating parts. This allows the coating material to contaminate the active surfaces of the pins due to capillary

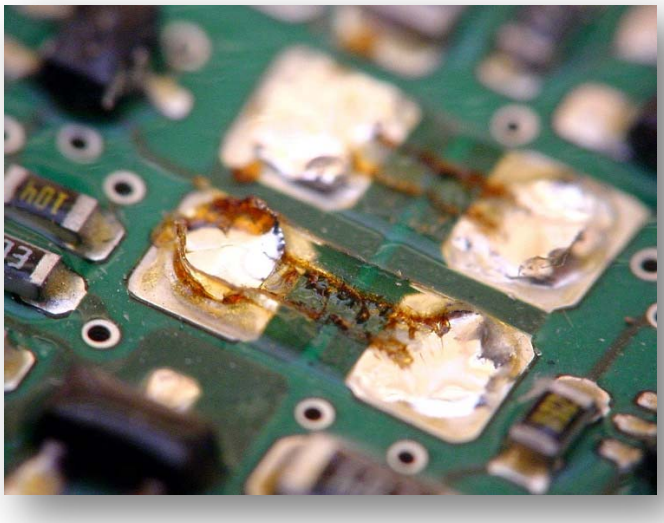
In recent years coating material suppliers have offered near 100% solid material for masking connectors. These materials are applied to the back of the connectors and don't penetrate. As the material is completely compatible with the coating material it is left in place under the conformal coating and overcomes the problem with connector design

Cracking of Coating Material



Cracking of coatings can happen due to the thermal cycling, poor adhesion of the coating or very brittle coating material. Use of coating material or the curing of boards outside the recommendation of the suppliers may cause problems leading to missing coating. Compatibility of coating with other materials on the board without cleaning may cause a reaction leading to lift. Some form of contamination under the mask has led to the cracking. There was no evidence of corrosion or electrical failure on the sample board assembly. Large amounts of no clean flux have been seen to cause lifting and not corrosion on boards. This is due to the expansion and contraction of the material during thermal cycling. This movement of the material can crack some masks. Review the board cleanliness and the coating compatibility with the intended application. Temperature cycling is probably the cause for the cracking but something under the mask can cause the mask to expand and contract.

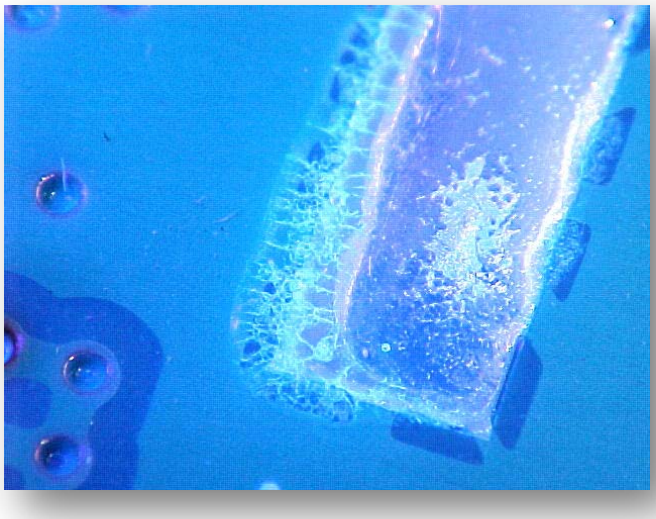
Coating Damage during Rework



When rework of solder joints is conducted through coatings there will be overheating of the coating unless mechanical techniques like blasting are used to remove the coating first. Heating discolours the coating and can impact their performance, in this case they would need to be removed and replaced either for functional or cosmetic reasons

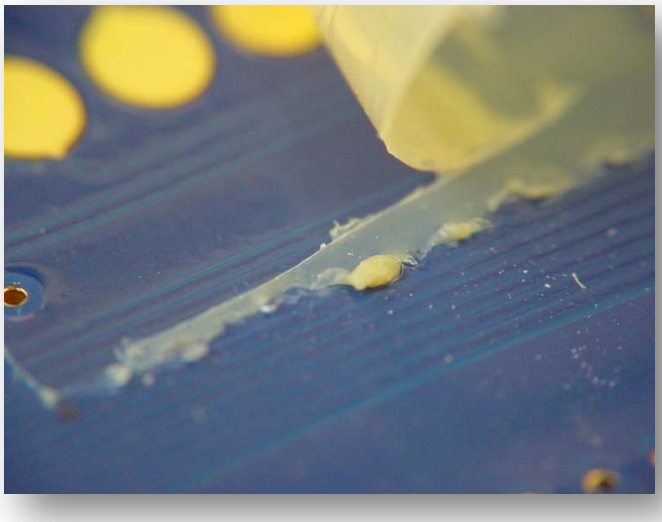
When reworking joints through coating and replacing parts flux is used and this makes it more difficult and may cause reliability issues with a poorly defined process. The example image is typical of the type of effect seen when reworking through coatings. It is very important that the process is well defined as it is in the IPC standards, IPC 7711/7721 Rework, Modification & Repair of Electronic Assemblies

Voiding on Board Surface



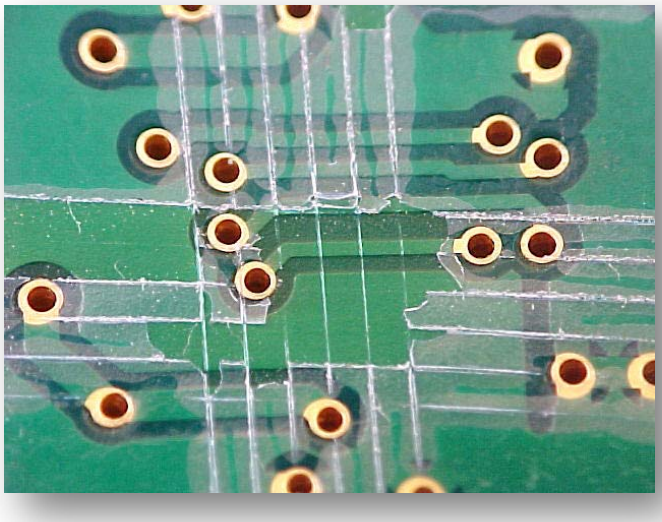
Small voids around a pad that has been soldered coating applied and cured and only visible in these areas must be related to the residues remaining on the surface of the board. In this case a reaction with the flux residues. Further trials need to be conducted on the material compatibility, the cycle times between soldering and coating and any thermal cycling on the board. Normally a no clean residue does not react like this. A very simple test would be to try a different paste on a couple of scrap boards following the same process procedure and times just to see if this is the issue

Conformal Coating Lift



When masking an assembly board with tape its important to remove the masking before the coating is fully cured or you will often lift the edge of the coating. There are a few old tricks of the trade which your mask supplier will advise but don't leave the masking in place too long. Minor lifted coating may have no impact on the reliability but it does not look very good and you will spend more time debating reliability than preventing the issue in the first place

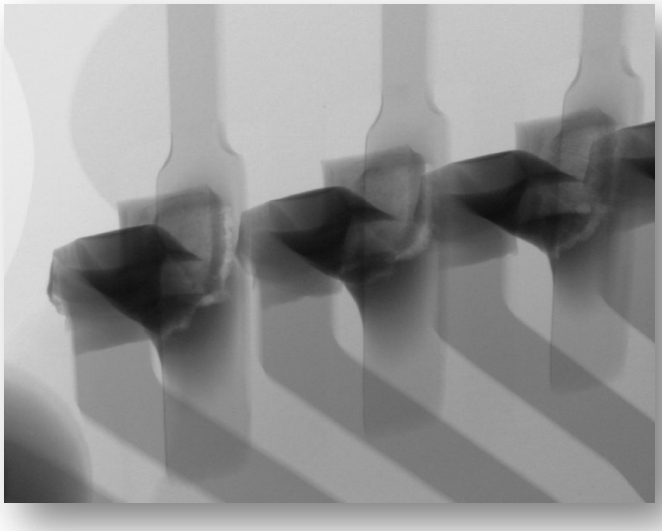
Coating Adhesion Failure



Cross hatch testing of coating and solder mask adhesion to the surface of a printed board is a simple and common process control technique. The example photograph shows separation of the coating from the green solder mask which may be related to the mask, contamination on the mask or poor processing. The first simple test is to coat and cure sample bare boards from the same batch and test. Bear in mind that the solder mask may be perfect but the assembly process the board has passed through may have altered the surface of the resist.

The crosshatch test is well defined in the IPC Test Methods Manual or search online for a guide from any coating supplier. You can use a special blade set to score the surface and peel with tape or measure the pattern manually and score the coating

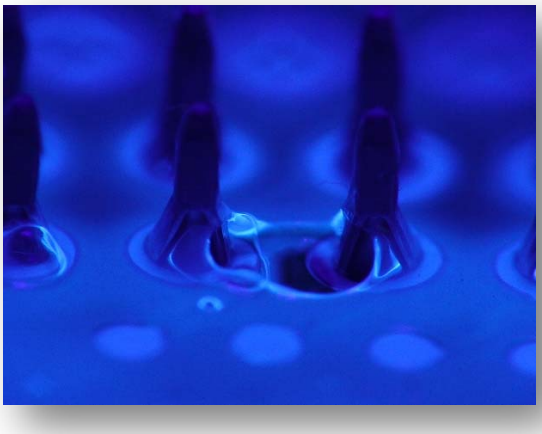
Solder Joint Failure

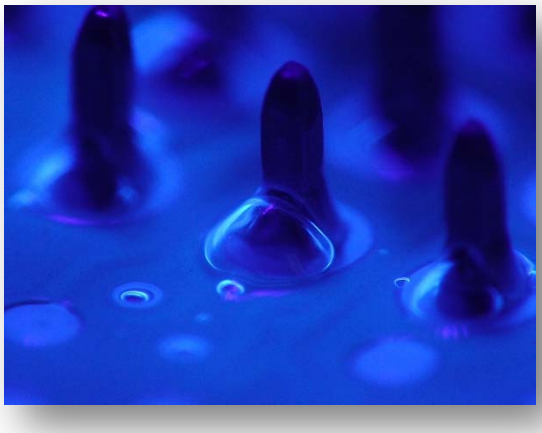


Solder joint failures on a QFP were investigated during the *NPL Process Defect Clinic* run by the author at IPC APEX. A delegate provided a board assembly that had failed after temperature cycling and was conformally coated. The thickness of coating varied around the package and had fully penetrated under three sides of the device. The coating material was not known during this evaluation but the cracking of the joints seemed to follow the thickness increase and the penetration of the coating under the package

It is important to make sure that any coating is compatible with the intended application and that the material CTE matches all materials associated with the product. From the examination, optically and with x-ray the joint failure occurred due to the temperature cycling and the force applied by the coating

Outgassing

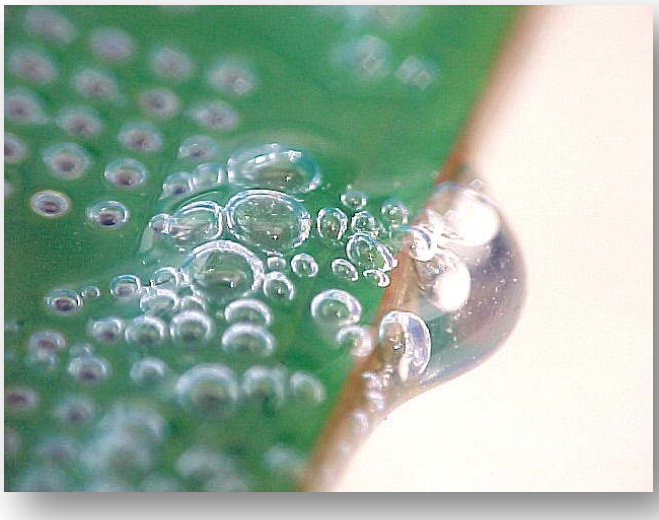




The two examples of bubbles have resulted due to excess coating in the plated through holes. Basically, voids were seen in the surface of the through hole solder joints probably due to outgassing from the board during soldering. The bubble was visible after curing at low temperature and probably the result of multiple bubbles accumulating in the coating but not necessarily breaking through the surface

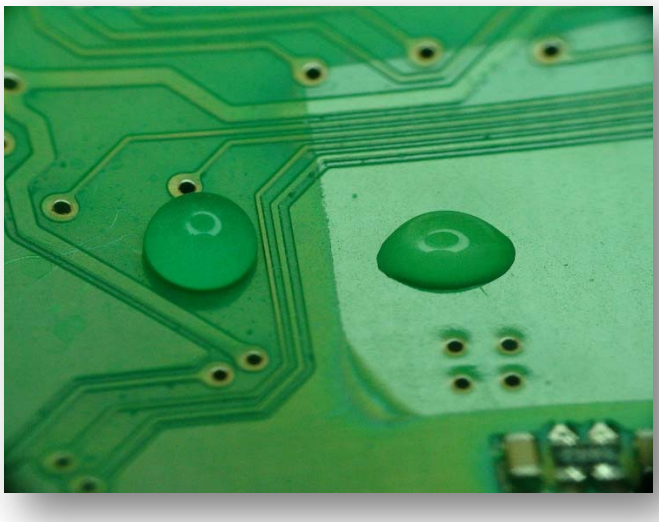
In some applications, the bubble around one pin may not be an issue but between two there is the possibility of corrosion failure. The same type of defect has also been seen around press fit connector pin in plated through holes where a large quantity of coating had accumulated

Poor Dip Coating Drainage



A dip coating process where the board's trailing edge leaves the bath of coating material is very dependent on monitoring the condition of the bath. The speed of removal of boards is automatically controlled and would be defined during set-up. However, the age of the coating, the specific gravity, temperature and any other contamination in the bath may affect drainage and a build up as show in the photograph. The number of small bubbles is a function of the thickness of the coating in this area

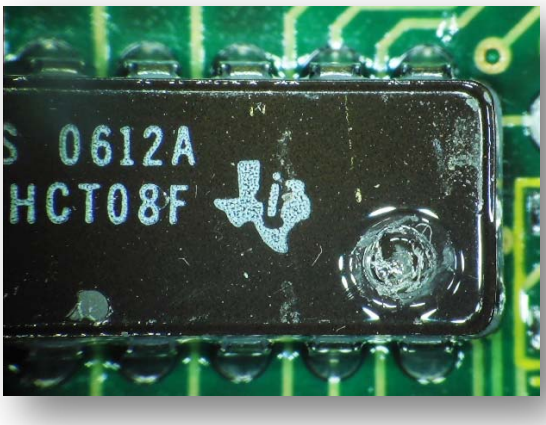
Variation in Surface Energy



Two droplets of water on the surface of the solder mask show the difference in surface energy of the droplet and the mask. All coating suppliers provide guidance and recommendation on the range of surface energy to get the best from their products. However, it is important to remember that during assembly we subject board assemblies to different processes and materials that can impact the solder mask. In turn these changes may cause poor or dewetting of the coating which is not down to the solder mask and PCB supplier

In this example, it may be the assembly masking!!

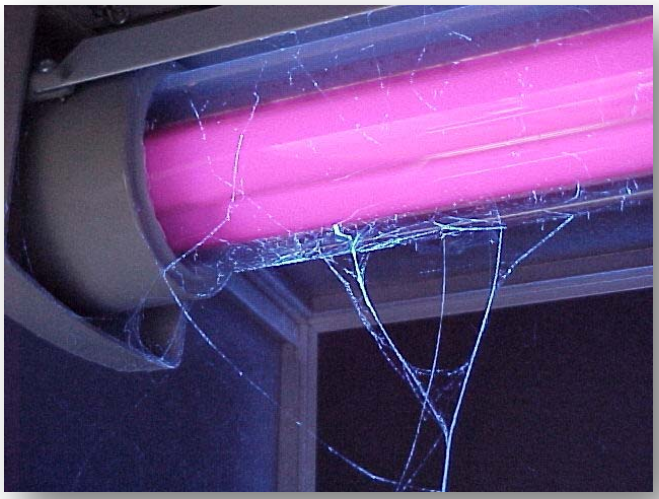
Bubbles in Coating



A thick deposit has been applied to the surface of the IC and a bubble has formed around the orientation mark. Our two photographs show inspection with and without UV light, reducing the coating thickness with reduce the possibility of this happening but this is not going to cause a problem in the field.

The bubble may have formed due to the volatile material in the depression, excess thickness of coating or contamination in the package indent

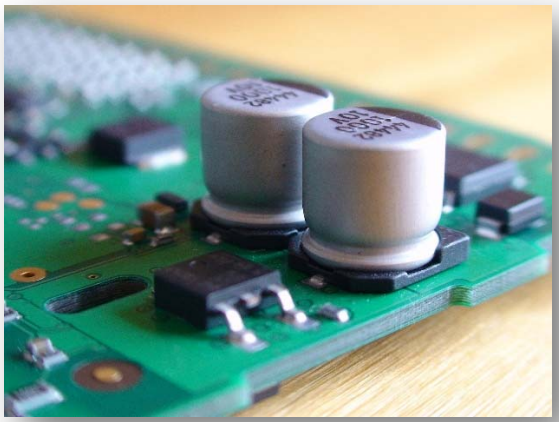
Coating Spiders Web

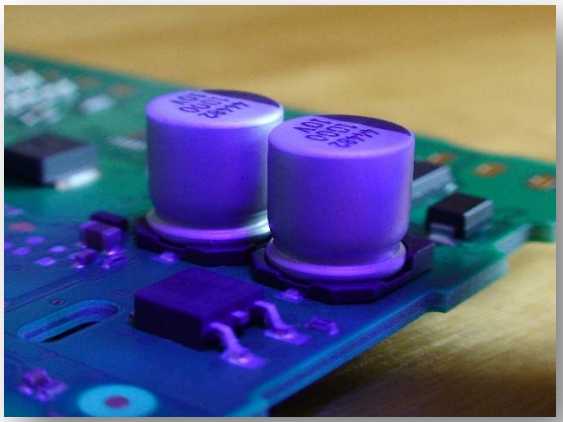


Example photograph of “Cobwebbing or Spider Webbing” in a spray booth under UV light and is due to the incorrect ration of coating and solvent. The solvent has evaporated during spraying leaving the webbing on the walls and the inspection lamp, fortunately not on the boards.

This example was not poor control in the coating shop but a practical example to show the author what the webbing looked like in practice as he had never seen it before in the real world

Missed Coating or Not?





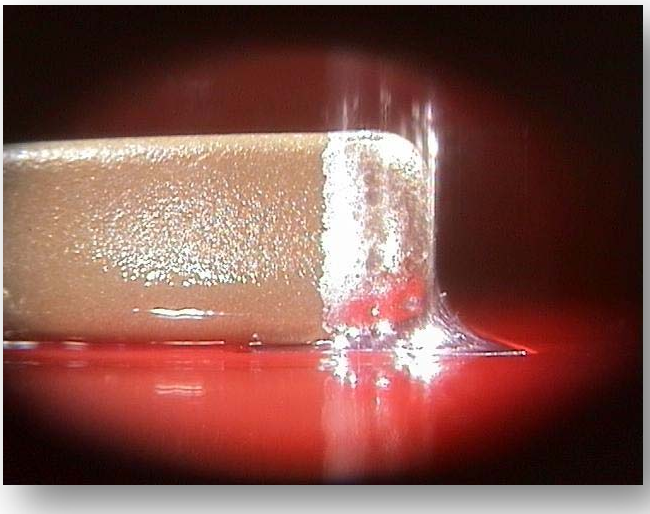
The first board example shows no coating but the second does show coating under a UV inspection light. Both boards do have a parylene coating which is considered the Rolls Royce of protective coatings but can be very difficult to inspect as it can be very thin but perfectly consistent in thickness. The difference in the two boards is simply the board on the left has parylene with UV trace the other one does not.

It should be noted that some liquid conformal coating materials are now provided without UV trace to avoid any problems with LEDs mounted on the board. Coating can discolour and impact the performance of the LEDs if it is designed to be coated

PCB Corrosion

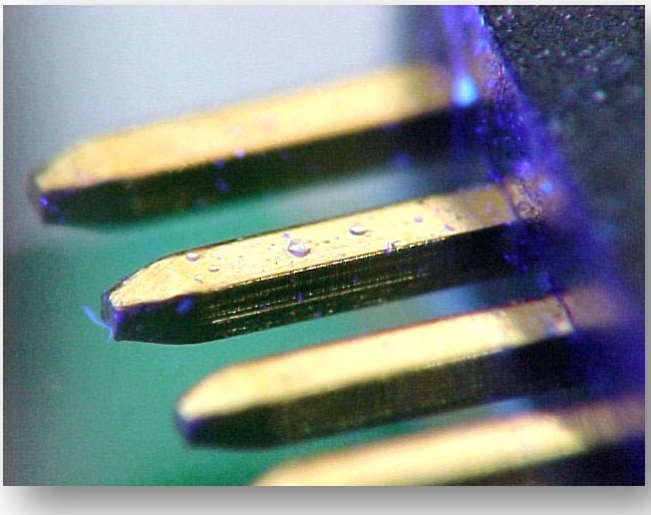
A coating can provide some level of protection in a sulphur environment but not all coatings provide this. In time sulphur gas can penetrate a coating. Removal of the coating shows evidence of corrosion. There have been a number of projects on the formation of sulphur corrosion products on printed boards. These were initially seen on silver finish boards close to the interface of the solder mask. In this area it is possible to have the silver and copper exposed. With a combination of dissimilar metals, sulphur, temperature and humidity similar formations have caused failures. There are two forms of this type of corrosion, blooms which form mainly due to the exposed metal surfaces and sulphur. The example above shows this along the pad and lead interface. Creep corrosion is where corrosion occurs across surfaces in higher humidity environments.

Satisfactory Coating



Satisfactory coating on the surface of a board assembly, but it's not possible to easily identify the successful coating process without a UV trace in the coating. This example does not appear to have a layer of coating over the body of the capacitor. The majority of all coatings now allow inspection by UV light. There are no UV traces in this Parylene coating which can make inspection difficult; however, due to the coating process coverage is assured. It is the inspection in areas that have to be masked that then become a challenge after de-masking operations.

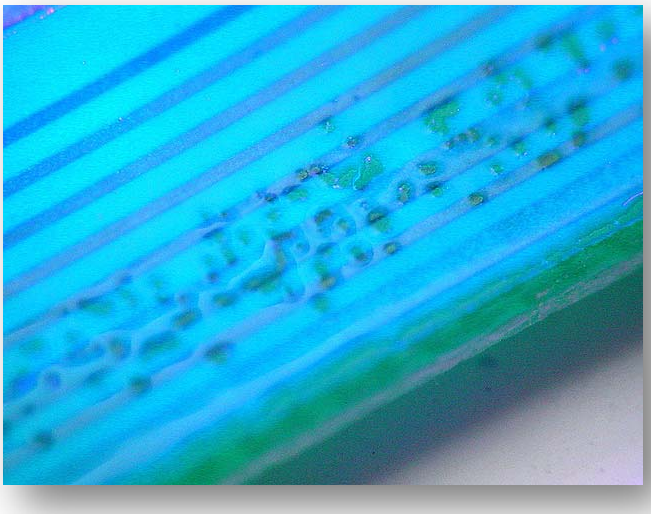
Conformal Coating Contamination



Coating contamination was found under UV light inspection, the coating process has caused the material to contaminate the surface of the connector pins. This board assembly was coated with an automatic spray system which was probably incorrectly set up for production. The overspray is visible under UV light on the surface of the gold connector pins and the body of the part.

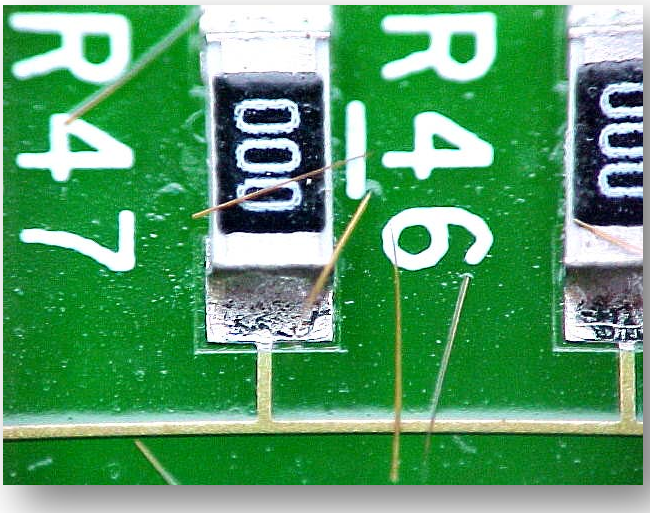
Rework of the connector pins in a UV inspection booth can be conducted using a cleaning fluid recommended by the coating supplier. The cleaning material should not damage the board assembly or the connector body. As gold contacts are inert there should not be an issue with contact resistance but other plated connector termination surfaces may need to be tested. It's important to run test boards or cards to check for correct set up and spray application then inspect the sample prior to running the process with product for the first time. It is not time consuming and can be conducted in less than 5 mins.

Coating De-wetting



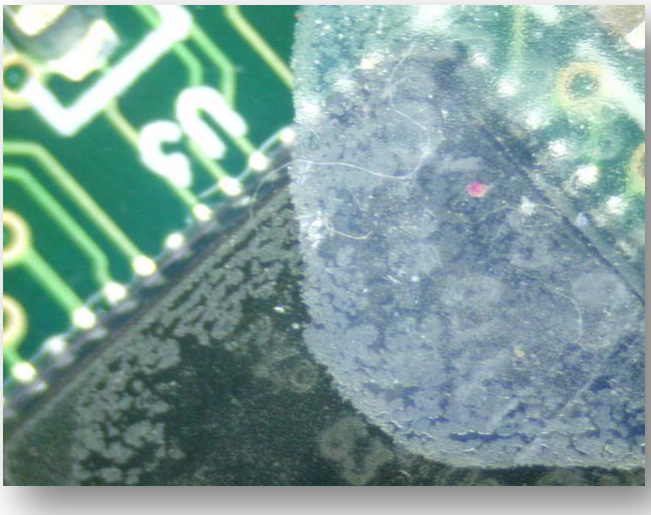
The coating is showing some de-wetting due to surface contamination on the surface of the solder mask. This would not be a general solder mask related issue but just local contamination possibly by handling. As this cosmetic defect is seen on the surface of the solder mask it should not be an issue but under most specifications it would be a cause for rework. The options would be to apply some more coating and cure or strip and recoat the board. This particular area of the board was always associated with V score lines from PCB fabrication and may be associated with contamination from the PCB manufacturing process.

Manual Coating Contamination



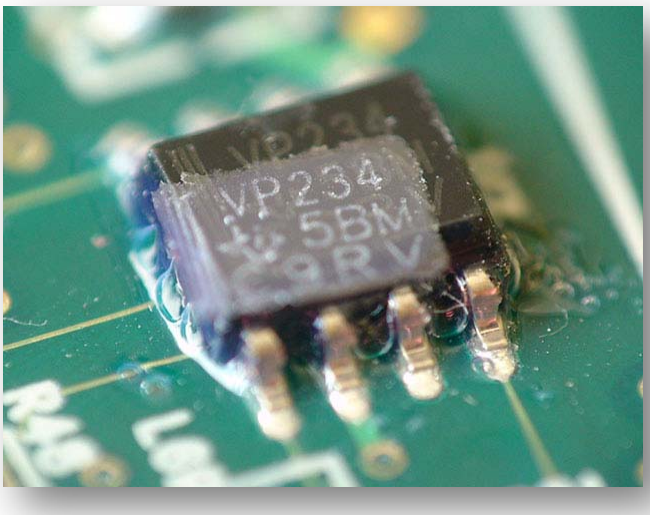
Manual coating a PCB is possible in small volume or after rework of selected areas. This can be conducted with a brush or spray can but needs to be conducted with care. Using a spray will probably require remasking selected areas on the board assembly hence brush application is attractive. Clearly manual application with a brush which is shedding fibres is very poor practice. Selecting a suitable non shedding brush is more expensive but if done correctly can be effective, provided the area is inspected after with a UV light source or the operation is conducted under UV.

Coating Lifting on QFN



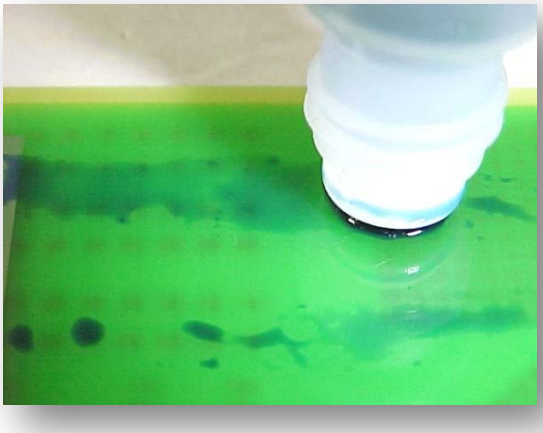
Coating has separated from the surface of the plastic QFN/LGA body which is not uncommon on many plastic SMT parts. Devices often have release agents on the surface which make coating adhesion difficult to maintain, minor loss or separation may not normally be reworked. In this case the root cause should be established as all the coating has completely lifted. Cleaning the board assembly after soldering may overcome the problem in future production runs but equally it may not. The fault can be seen on most coating systems, more likely on thinner coating, although the poor adhesion will still be apparent on thicker coating that may not be easily displaced.

Poor Coating Adhesion



Example of poor coating adhesion on the surface of an SOIC. This may be due to contamination on the surface of the plastic component as manufactured or due to contamination introduced during the assembly process. The interesting point on this example is the part removal of the component identification marking which is now visible in the surface of the coating. If the actual identification marking on the component body and the marks transferred in the coating were different we may be looking at a counterfeit component problem rather than just adhesion of the coating?

Poor Solder Mask/Coating Compatibility



It is very important to specify the type of solder mask coating when purchasing bare printed boards, don't leave it to chance. Most coating material suppliers and contract assembly services will want to make sure the coating materials specified are compatible to avoid dewetting. All suppliers will provide recommendation on the range of surface tension measurements on the surface of a mask to be coated. The image shows a dyne pen being used to check the surface energy on a solder mask surface. Normally a set of pens will be used to confirm the wetting range, or not, based on different fluids in the pens. Remember also that the surface tension of a solder mask can change based on the assembly process the boards are subjected to during assembly.

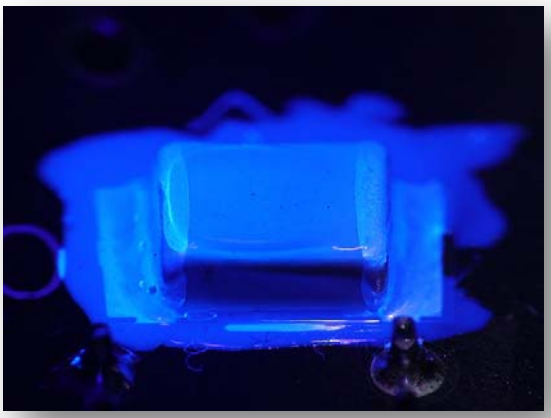
Satisfactory Coating

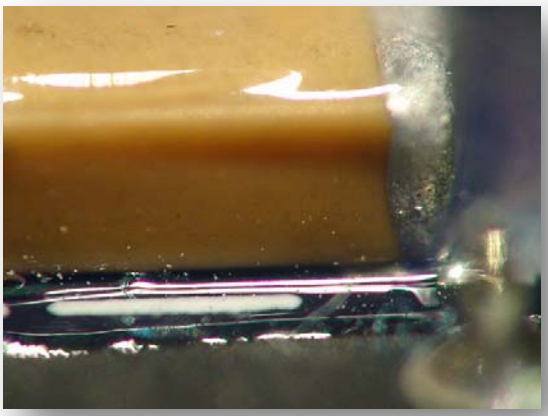
Satisfactory dip coating of this SOIC is shown during inspection under UV light. The coating has completely covered the surface of the board, terminations and body of the component. If a spray coating process were used all the surfaces would be coated to an equal standard but there would not be coating filling the gap between the component termination and body. It's always important comparing like for like.

Poor Rework of Coatings

There are different methods of reworking conformal coating; some create more mess and problem than others. In the examples above heating or soldering through the coating is quick but then requires further cleaning steps. The image on the left shows the lifting of the coating with the component when using thermal tweezers. On the right flux residues or the coating material has overheated during coating removal. IPC standards provide a number of different techniques based on the material type, coating thickness and rework operation required. Although more expensive, using abrasive coating removal is simpler and more versatile than thermal or chemical coating rework

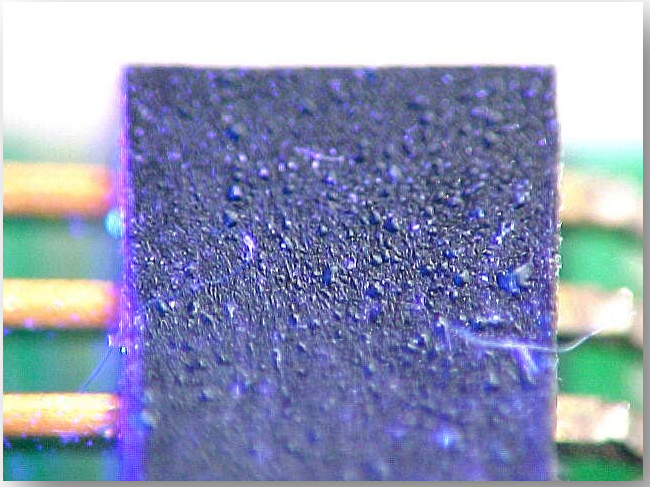
Selective Conformal Coating





Two examples of manual selective coating the one on the left is under UV light. It is clear that the image on the left has been added manually but, provided it meets the requirements of the design, is satisfactory despite the uneven coating border. The close up image on the right shows the coating has fully covered the capacitor and its terminations.

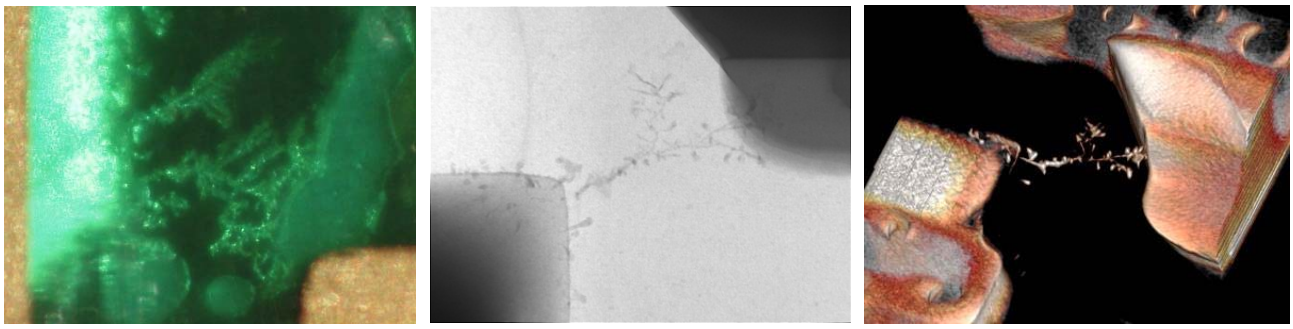
Poor Coating Due To Overspray



Coating overspray is shown on the surface of this connector body during inspection under UV light. There was also evidence of contamination on the surface of connector pins which would be more of a practical issue than the connector body. Coating material on the surface pins may lead to high resistance or intermittent contacts. The board assembly had been spray coated and it is likely that the spray patterns and coverage area had not been correctly defined.

Copper Dendrites/Surface Corrosion

Three dramatic examples of 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. Copper dendrite 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.

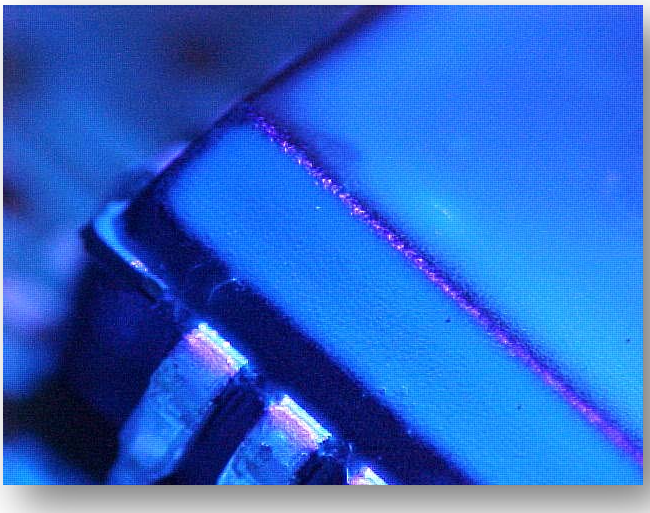


Copper dendrites, one of the most common failure modes, and three ways of seeing the same failure

A copper dendrite often creates an intermittent fault which can be very difficult to pinpoint. Examples like these can be seen under and over solder mask coating or under conformal coating when there is poor surface adhesion of the coating hence the contamination on basic printed board is important. Contamination testing and Surface Insulation Resistance (SIR) assessment are two techniques often used to monitor and control the levels of harmful contamination or assess the assembly material's performance 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 level of 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. Three different techniques were used to investigate the failure shown above. The first image shows back grinding a microsection up under the chip component to avoid removing the evidence of failure as so often happens in test departments. Standard high resolution x-ray was used in the second example after the location of the product failure had been established. This technique was used to examine for failure in similar locations on other products. Using x-ray with CT capability allowed a three dimensional view and a clear understanding of the position of the dendrites without sectioning.

Satisfactory Coating on PLCC



Coating on sharp corners can be difficult. The flat surfaces of this PLCC device shows satisfactory coating. Provided the surface of the board and all terminations and separation gaps between adjacent exposed pads are coated the reliability of the product should be satisfactory. However, based on many standards no coverage or edges of plastic parts would be considered unacceptable. It is felt that inspection standards and criteria do need to be regularly reviewed and updated. To achieve a better coverage on devices like this it would be necessary to coat the board assembly again but also look at the viscosity of the material being used. If the solids content was low it may have reduced the coatings coverage.

Copper Dendrite Failure

Coating failure has occurred allowing the copper dendrites to form across the surface of the solder mask between two terminations. This was found after removing the coating during failure analysis of the circuit board. Either the soldering materials used or the cleanliness of the basic PCB was inadequate for the product environment. Comparing the soldering materials, process and coating combination can be conducted using a standard test board and measuring Surface Insulation Resistance (SIR) when exposed to a controlled environment.

Copper dendrite formation is one of the most common failure modes experienced in no clean or under conformal coatings. The dendrites, however, are often missed when engineers try to remove the coating to investigate failure. Correct failure analysis techniques are often not used hence no fault found

Coating Bubbles “Bug Eyes”

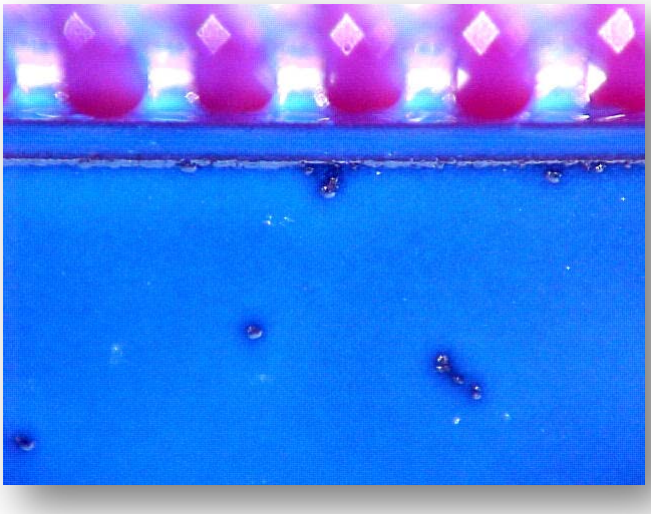
Conformal coating bubbles or, as I would call them “Bug Eyes” due to their size, are caused by entrapped air or other material outgassing from under parts. It is also possible to get these by fast/force curing the coating before the solvent has had time to evaporate or air to escape. If present across a row of termination points it will be a reliability problem as the coating will not offer any protection. It will in fact allow a moisture trap if the product is in a high humidity environment and possibly lead to failure.

This would not be acceptable under any current specifications.

Satisfactory Coating on SOIC

Coating is satisfactory and is shown under a UV light source. There is good coverage on the component leads, component body and the surface of the board between the 0.050” pitch terminations.

Minor De-wetting or Bubbles



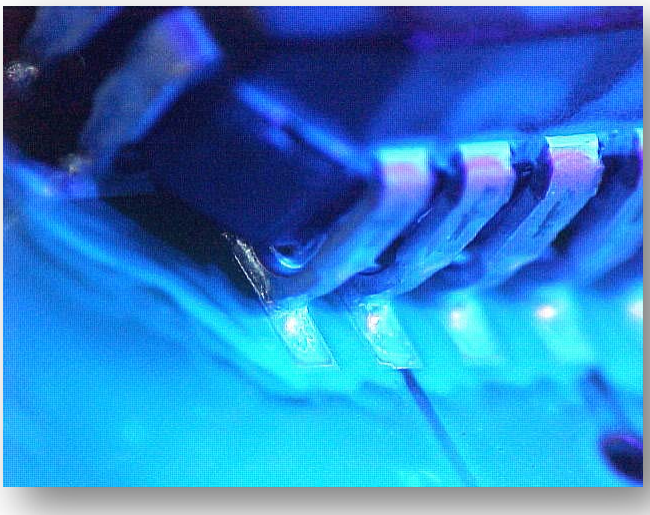
Coating has not wet to small areas on the top surface of a plastic surface mount IC. This may be contamination on the surface, however non wetting would be more pronounced. The limited coverage seen on the edges is standard and commonly seen. A high solids material or repeating the coating process would overcome the limited coverage on the edges but only cover over the voids making them look more like bubbles.

If the coating is satisfactory on all other surfaces where there are connections between terminations, slight blemishes do not really matter or impact reliability and are totally a cosmetic issue.

PCB Surface Contamination

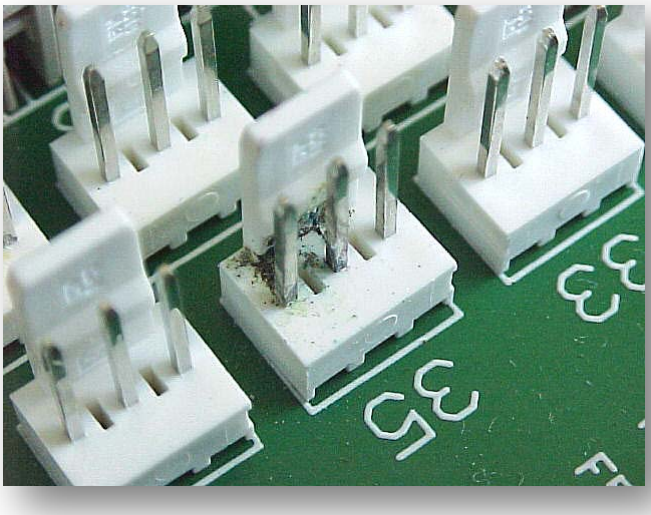
Poor manual handling of the board assembly prior to coating has resulted in poor coating adhesion. Boards should always be handled by their edges after cleaning or in a no clean assembly process. Contamination on the surface of the board may have come from other sources and needs to be reviewed and the product re-cleaned. The solder mask and its compatibility with the conformal coating should be reviewed with adhesion tests. If the board has been through a cleaning process the solder mask may have absorbed material from the cleaner and need time for it to evaporate from the surface of the mask prior to coating. Always consider baking the board assemblies before coating.

Coating Capillary Action



Coating inspection under UV light shows the impact of capillary action, although there is evidence of coating coverage on the board and component surfaces coating has capillaries under the PLCC. This is indicated by the tide line around the component, based on inspection criteria, this still meets common standards but the thickness of the material will vary on the surface of the board. Changing the viscosity of the material, reducing the travel speed of the spray, increasing the amount of material or solids applied during spray application should improve the results.

Corrosion Dendrite Formation



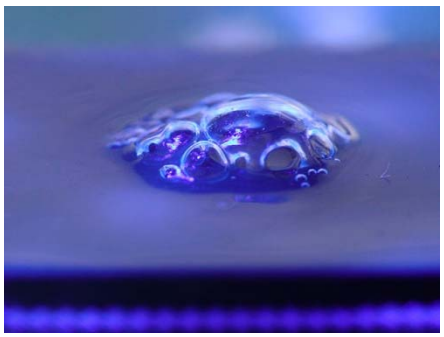
Corrosion and dendrites have formed across the surface of the plastic connector body between two terminations. This was caused by excessive flux being sprayed on to the top surface of the board. The flux could have penetrated up through the connector by capillary action between the pin and connector body. In the case of no clean materials they need to be exposed to the correct preheat to allow full evaporation of the carrier solvent and deactivation of the flux. If a coating had been used in this application the same problem could have occurred on other sections of the board. Coating cannot overcome poor production set-up or control when potentially corrosive materials are still on the surface of the board.

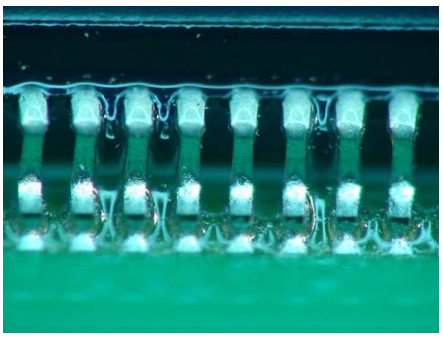
Satisfactory Coating

Coating inspection under UV light shows satisfactory coverage on the surface of the BGA and board assembly. However, there is little penetration under the BGA with spray coating. Dip coating will help penetration under the area array component if that is what is required. However, a slower entry speed and dwell time will be required in the tank. Time must be allowed for the coating to penetrate fully and displace any air.

Care needs to be taken with the choice of coating materials if coating under these parts is actually required. Selected coatings can have a high expansion rate during thermal cycling and could lead to solder joint damage.

Coating Bubbles

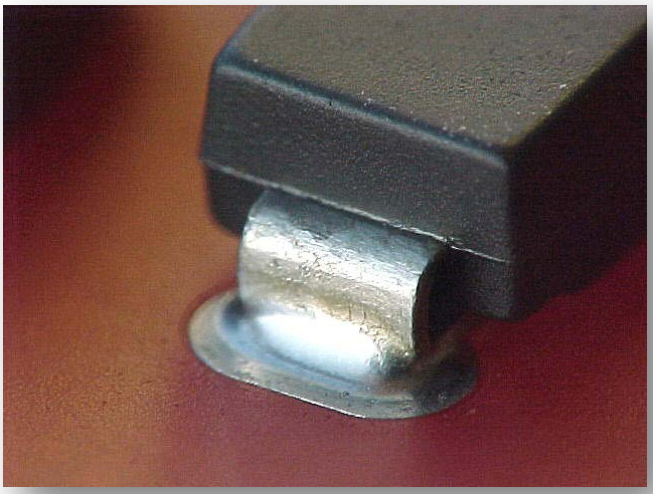




Bubbles in the coating surface are not uncommon, the criteria is covered in different industry specifications, the key issue is where the bubbles are positioned. If they are between terminations, tracks and solder joints they can potentially allow a reduction in insulation resistance and would be unacceptable.

If, however, they are on top of large components or on the surface of the solder mask not above tracking they should not be an issue. However, many people and specifications state this to be a rejected regardless of the area that is affected and if the assembly is fit for purpose. Some companies actually place a size criteria on the bubbles based on their location. It's a pain to measure these but also a pain to strip and recoat! Bubbles are most commonly related to hold and cure time with the inability of solvent to escape or trapped air still to be displaced. If this occurs on a stable dip process look at what process parameters have changed, something must have changed!!

Satisfactory Coating



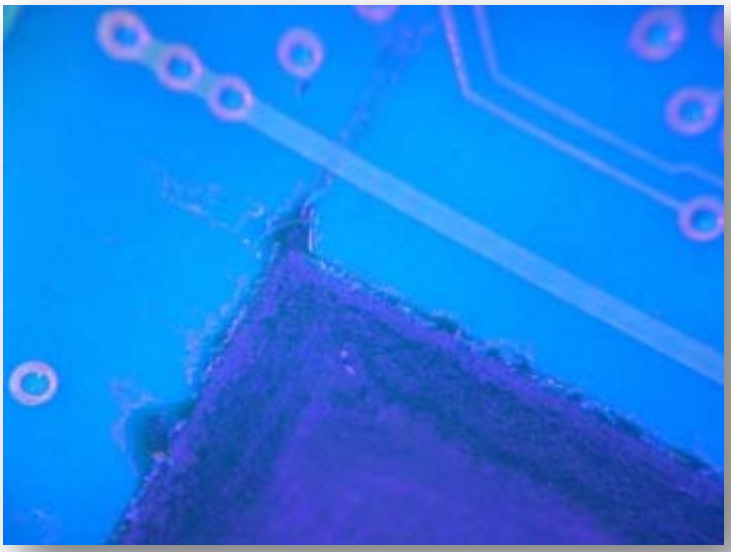
This board assembly has a Parylene coating which is around 15um in thickness on all surfaces of the board and components. Based on the process the coverage and thickness is virtually guaranteed. It is however difficult to see the coating due to the process and the type of material used. There is generally no UV trace in this material, however some suppliers are now able to provide it on selected types

Coating Bubbles/Incomplete Coverage

Coating bubbles or voids, regardless of the name are not acceptable as clearly the coating does not provide any protection between the two termination points on this SOIC. This and other positions like it may be down to local contamination in specific areas on the solder mask preventing the surface wetting during dip coating. It may also be the case that this product has been reworked leaving flux on the surface of the board.

The coating has wet the component terminations, the component body but not the surface of the mask. If contamination of the surface is to blame there should be other areas like this one example.

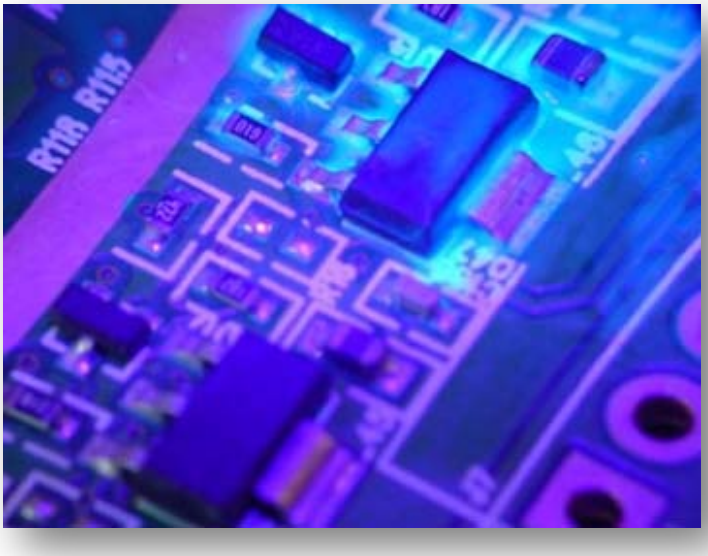
Incomplete Coating Removal



Coating rework can be difficult and time consuming so every effort must be made to make sure the process is well controlled and a first off is always inspected prior to working on a long batch of work. In the example a selected area of the board is being reworked with a solvent stripper and masking tape is being used to define the area.

After stripping and inspection under UV light it is clear that the coating has not been fully removed which may be acceptable or unacceptable for a given application. There are other coating removal methods which may be faster and more successful like abrasive jets which provide a cleaner option. Selecting the right masking tape is important to reduce the amount of solvent seeping under the mask. It should be removed as soon as possible to prevent further attack of the main coating.

Coating Removal for Rework



Coating inspection is being undertaken under UV light after chemical stripping. Rework is being conducted on the board assembly prior to surface mount component removal. It is often easy to solder through a coated board with selected materials. However, the coating will char, leave residues, require cleaning and need to be conducted under suitable extraction. If the coating is too thick contact heating is less effective than the preferred technique.

The example board shows one area which has been cleaned and the top area is not fully stripped. It is virtually impossible to remove coating from under components. This can make component removal more difficult just like using adhesive. The trick is to raise the temperature of the board assembly above ambient even when using contact tools where you can apply a small twisting action to the body of the part.

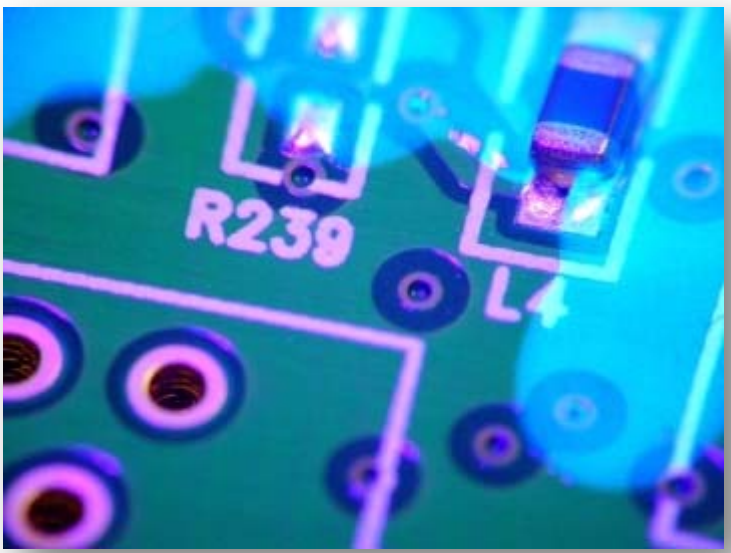
Unsatisfactory Masking

Masking board assemblies prior to coating is unfortunately a common practice during conformal coating. More consideration at the design stage would reduce the need for masking, which is often difficult and always time consuming.

Incomplete Coating Coverage

The conformal coating has not encapsulated the side of these two resistors which were manually sprayed. Manual spray fluxing can give a satisfactory coating performance, although it will vary in thickness. When manually spray fluxing boards apply the coating from one direction at approximately 45deg, and then repeat the spray coating in three additional steps after first rotating the boards through 90deg. Try to avoid applying excessive coating in any one step. During manual spray fluxing it is good practice to check the area of coverage on a test surface then inspect under UV light. This provides a simple method of looking at the spray width, consistent coverage and some comprehension of the amount of coating applied with any one spray stroke

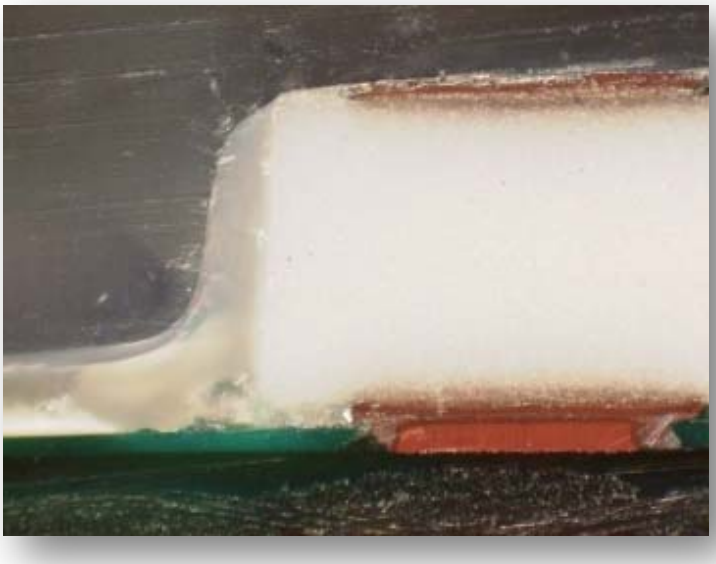
Poor Coating Application



Coating on the board assembly should have had a straight edge; it is not clear why the uneven application line has happened in production. The coating was applied using an automatic spray system and previous example boards were satisfactory. Running a blank test board or test cards prior to coating fully populated boards should be standard practice before running a production batch.

The cost of contract assembly is often all about the masking/demasking process and not the cost of coating or materials. Making sure that every point that does not require coating has been protected can be made easier. For manual spray coating boards are often placed on a rotary table and in a UV coating booth. If all the boards are located in lines and the masking tape used has a UV trace or naturally fluoresces it can make inspection for missing masking very simple before spraying unprotected surfaces. Missing dots are then fairly easy to spot; can you see the missing dots?

Conformal Coating Coverage

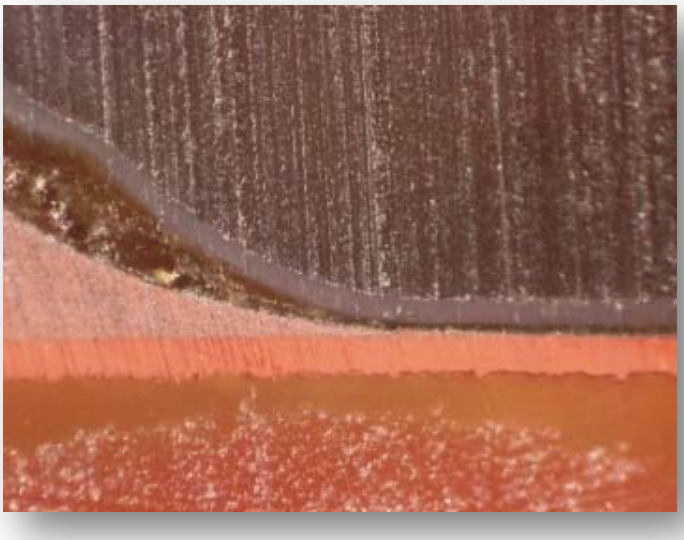


It is difficult to measure the thickness of coatings as they vary by process, material and the surface they cover. With the exception of Parylene, which has a defined coating thickness, the following is generally true.

* Flat surface may have a consistent coating thickness
* Sharp corners have a thin coating
* Thickness can build up between adjacent surfaces particularly on dip coating

The image shows a microsection of a chip component on the surface of a printed board after coating and cure. Epoxy resin has been used to hold the sample for grinding. The coating thickness on the side of the chip is thick and on the sharp edge of the ceramic part the thickness is significantly reduced. This is no different than any other example you may examine. Changes in the process parameters and materials could increase or decrease the coating thickness on the board assembly.

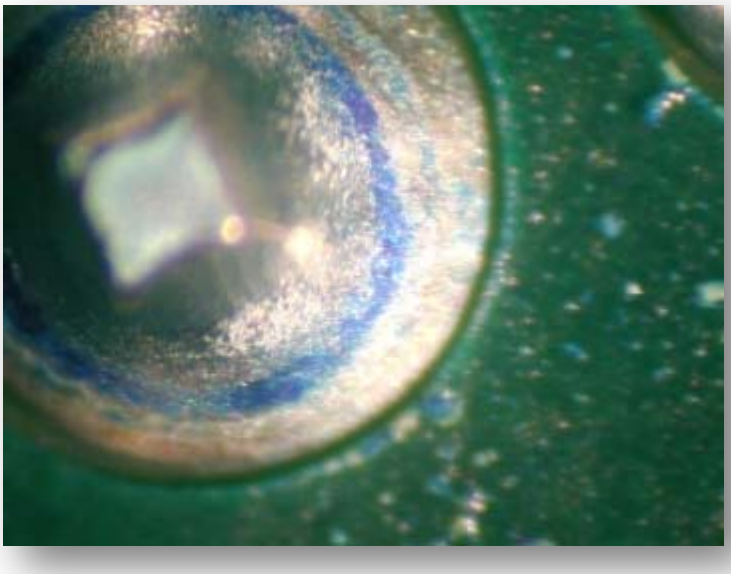
Conformal Coating Coverage



Coating coverage on this microsection shows the potential performance advantages of Parylene coating. The image shows the coating which is between 20-30um directly over the surface of the board, solder joint and reflow paste flux residues. The flux residues are the brown material appearing to lift the coating, however, the coating is actually just following the contour of the residue.

There are very few technical research reports on flux residues on the surface of boards after conformal coating. Often people focus on the flux and its possible corrosion under the coating; however, the thickness of any inert residues can expand and move under the coating possibly leaving voids. This in turn allows moisture to form in the voids and remain in place, which can then cause failure. This is more difficult to define, reactive flux residues are easier to research and monitor.

Flux Residue Testing

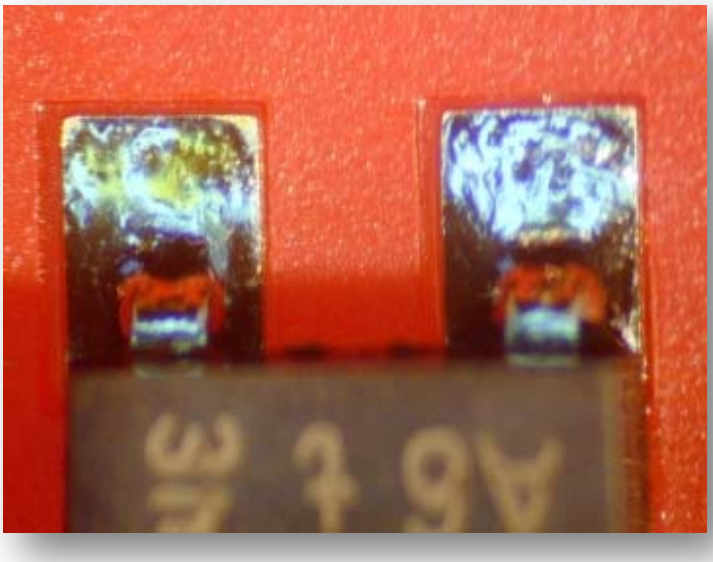


Traditionally contamination on the surface of a board assembly is tested using a contamination measuring system like ionic measurement or surface insulation resistance. In recent years the use of a more sophisticated techniques like ion chromatography have been used to quantify the contamination type.

A simpler fluid indicator test designed by Zestron is available, it tests for activators and resin on the surface of joints either before or after a cleaning operation and prior to coating.

Basically the test fluid is placed on sample areas of the board assembly or on selected joints. The fluid must be left on the surface of the test feature for a fixed period of time for a reaction to take place. There are two different fluids, one for activators and resin systems. In the example a through hole termination produced by selective soldering has been tested for flux activators. The change in colour to a blue deposit shows a positive result indicating there are activators still present on the solder joint surface. Blue residues can also be seen around the joint on the solder mask. A company would need to establish their own criteria between this test method, cleanliness results, SIR reliability data and possibly product exposure to specific operating environments.

Flux/Resin Residue Testing

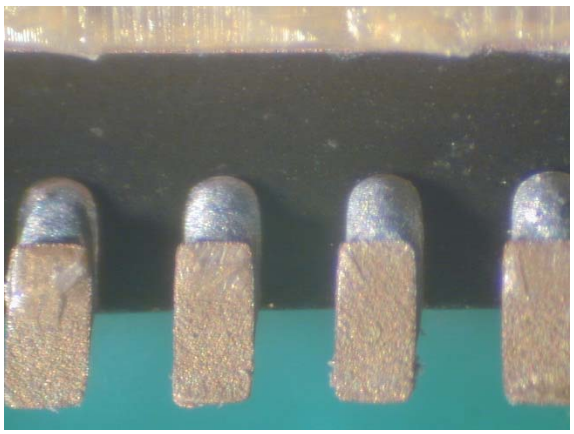


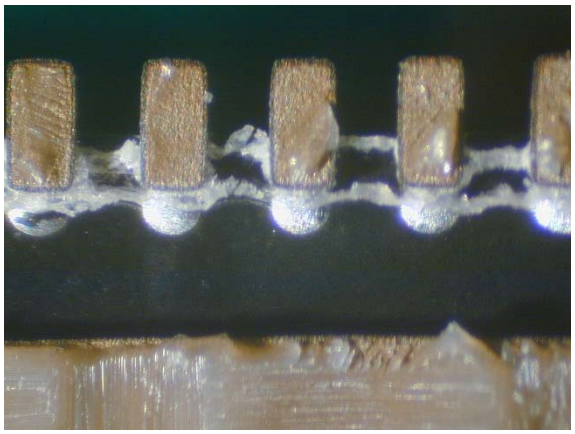
Traditionally contamination on the surface of a board assembly is tested using a contamination measuring system which is defined in IPC standards. A simple fluid test indicator designed by Zestron is available, it tests for activators and resin on the surface of joints either before or after a cleaning operation and prior to coating. Basically the test fluid is placed on sample areas of the board assembly or on selected joints. The fluid must be left on the surface of the test feature for a fixed period of time for a reaction to take place. There are two different fluids, one for activators and resin systems.

In the example SOT23 terminations produced by reflow soldering have been tested for resin. The change in colour to a brown deposit shows a positive result indicating there is contamination still present on the joint surface. A company would need to establish their own criteria between this test method, adhesion, cleanliness results, SIR reliability data and possibly product exposure to specific operating environments.

Cleaning Under QFN

Successful cleaning under low stand-off height QFN packages can be achieved as shown on the first image below; poor flux and cleaner compatibility is shown on the second. Both were cleaned in a batch process using different process recipes. The sample parts were removed from the surface of the board mechanically to show the process performance.

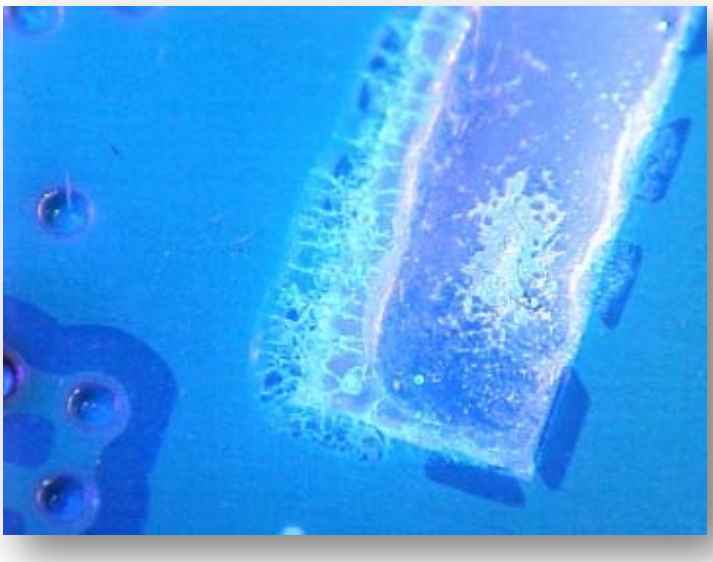




Modification to the design rules for a component can also increase the reliability of the cleaning process and, if marginal, also reduce the risk of failure, Increasing the distance between the termination pad and centre pad, and eliminating solder mask under the package can also help cleaning performance.

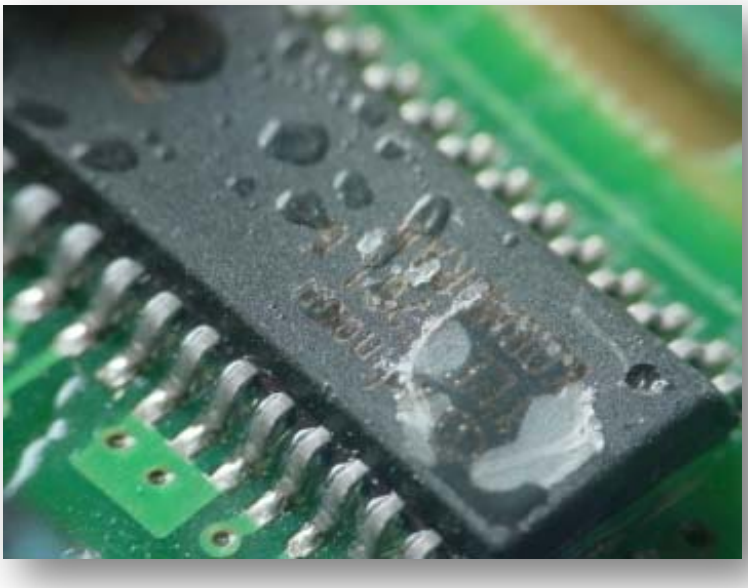
The key to evaluating residue removal is the solubility of the residue in the cleaning material before undertaking the actual cleaning trials. When this is established then the correct cleaning process can be defined to allow full penetration under the components of the cleaning chemistry before confirming that it can also be removed during the rinsing phase of the process.

Conformal Coating Lift



Coating is lifting around the edge of the pad. A reaction has taken place with the flux under the coating. Inspection showed this to be visible under all of the termination points after reflow soldering so it is assumed to be related to the residues left on the joint areas

Poor Conformal Coat Wetting



The coating has not wetted the surface of this plastic IC which is not uncommon. The Small Outline Integrated Circuit (SOIC) may have some contamination on the surface of the plastic. Mould release agents are often suggested as the main cause of de-wetting on ICs and it is difficult to eliminate this unless the product is going to go through a defined cleaning process.

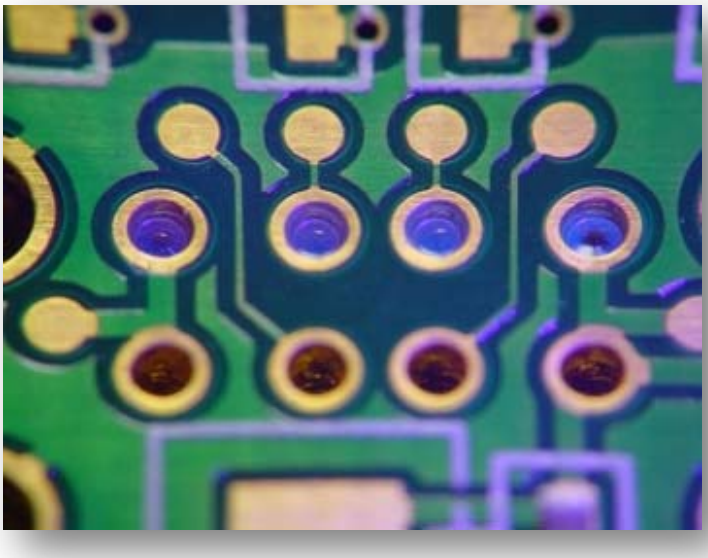
It is suggested that minor de-wetting is acceptable provided all other inspection criteria are met. The example shown would not be considered minor. The main coating requirements are all terminations, adjacent pads, and exposed terminations be coated to a defined minimum thickness with no bubbles between the terminations with minimum design clearances. This does, however, contradict existing quality standards that have not been reviewed for many years.

Excessive Coating Thickness

The conformal coating applied in this example is excessive which has probably lead to the bubbles forming during cure. The excessive thickness allows the outer surface to dry and form a skin which does not allow further solvent in the bulk of the coating to evaporate allowing the bubbles to form.

If bubbles are only visible on the top surface of the Quad Flat Package (QFP) they should be considered a cosmetic defect, not requiring rework. Modifications should be made to process parameters rather than reworking the product. There is also some pull back of the coating on the edges of the component, possibly due to the thick deposit on the top of the device.

Conformal Coating Penetration



Coating has penetrated through holes to the opposite side of the board due to the process parameters or the viscosity of the material. This may not be an issue but if this contamination could cause a problem in manufacture the holes should be masked, this could be a permanent mask sealing the holes to eliminate removal.

Changing the process parameter on the spray system could be considered along with the solids content of the coating material or the hold time prior to full curing. A short cure time may not allow the coating to tack dry. If the coating thickness is excessive the surface will flash dry leaving the liquid coating to drain through the holes during the hold period. The best solution is masking to guarantee the process reliability.

Coating Overspray

Coating overspray has been detected during inspection under UV light. The outer case of this connector shows evidence of the coating which may be due to incorrect programming of the robotic spray system. Although the connector could be masked this adds to the time and cost in manufacture.

Even if the engineer's programming is correct, changes or fluctuation in process parameters may be the root cause of the problem. However, variations in the positioning of the product on the equipment may be a contributing factor to the contamination. Although using a standard blank board can be used for machine set-up and optimisation, an assembled board must be used for machine program confirmation.

Text Books on Coating, Cleaning & Contamination Testing

Handbook for Critical Cleaning – Cleaning Agents and Systems
Barbara & Edward Kanegsberg – CRC
Cleaning Printed Wiring Assemblies in Today's Environment Les Hymes – VNR
Cleaning & Contamination of Electronic Components & Assemblies Brian Ellis
Handbook of Aqueous Cleaning Technology for Electronic Assemblies
F R Cala & A E Winston – Electrochemical Publications
After CFCs – Options for Cleaning Electronic Assemblies
Dr Colin Lea - Electrochemical Publications
Coating Materials for Electronic Applications James J Licari – Noyes Publications
Conformal Coatings for Electronics Applications Manfred Suppa – Werner Peters
Contamination Effects on Electronic Products Carl J Tautscher – Dekker
Aqueous Cleaning Handbook Malcolm C McLaughlin – Morris Lee Publishing

NPL Cleaning Related Reports and Good Practice Guides

Here is a list of the cleaning related reports available free of charge to engineers to download to assist them with their cleaning process development. The project reports can be downloaded direct from the database.

An Assessment of the Suitability of Current PCB Laminates to Withstand Lead-free Reflow Profiles
Development of a New Surface Insulation Resistance (SIR) Test Method
Development of Surface Insulation Resistance Measurements for Electronic Assemblies
Test Procedure for Process Validation With Surface Insulation Resistance
Effect of Test Voltage, Test Pattern and Board Finish on SIR Measurements for Various Fluxes
The Role of Permeability and Ion Transport in Conformal Coating Protection
Evaluation of the Ability of Conformal Coatings to Inhibit Tin Whiskering
Measuring Anionic Contamination of Printed Circuit Boards & Assemblies using Ion Chromatography
Preliminary Measurements of Solder Flux Residues in an AC Environment
Protection Performance of Conformal Coatings in Harsh Environments
Susceptibility of Lead-Free Systems to Electrochemical Migration
Test Method for Conformal Coating Protection Performance of Assembly for Harsh Environments
Test Method for Measurement of the Propensity for Conformal Coatings to Inhibit Tin Whiskering

IPC Cleaning Standards and User Guides

The following are a list of IPC documents on cleaning processes and assessment of the cleaning process performance and can be purchased direct from the IPC via their website www.ipc.org or via any of their worldwide distributors.

IPC-HDBK-830A Guidelines for Design, Selection and Application of Conformal Coatings
IPC CH-65B Guidelines for Cleaning of Printed Boards and Assemblies
IPC 5704 Cleanliness Requirements for Unpopulated Printed Boards
IPC 7526D Stencil and Misprinted Board Cleaning Handbook
IPC 9201A Surface Insulation Resistance Handbook
IPC-5701 Users Guide for Cleanliness of Unpopulated Printed Boards
IPC-5703 Cleanliness Guidelines for Printed Board Fabricators
IPC 5702 Guidelines for OEMs in Determining Acceptable Levels of Cleanliness of Unpopulated PCBs
IPC-7711B/7721B Rework of Electronic Assemblies/Repair & Modification of Printed Boards & Electronic Assemblies

Listing compiled by the author for the "Cleaning & Contamination Testing Centre" special feature areas organised at IPC APEX, Productronica and New Exhibition exhibitions in 2013-2014

Conformal Coating Training Material & Services

Over the last few years we have created a number of products and services for conformal coating training and education which are commercially available worldwide. These products and services are based on practical experience and evaluation projects conducted by the author. Any of these products or services can be obtained by contacting info@smartgroup.org or in the case of CD-ROM and poster sets they can be obtained from other industry groups. SMART provides onsite training and online webinars on coating and defect prevention

Conformal coating CD training products, photo CD and inspection charts shown above can be obtained from the following organisations or websites. SMART Group can also run theory and hands on workshops or online webinars for your company:

SMART Surface Mount & Related Technologies Association
www.smartgroup.org

IPC - Association Connecting Electronics Industries
www.ipc.org

SMTA Surface Mount Technology Association
www.smta.org

Electronics.ca Publications publishing company
www.electronics.ca

Cleaning Printed Circuit Assemblies, Design & Process Control Workshop

(The workshop can also be run as a shorter online webinar)

Workshop Introduction:

The majority of the industry worldwide have been running no clean processes for many years and have often, understandably, neglected important issues like design for cleaning, selecting compatible components and compatibility between cleaning materials and flux residues. With increased miniaturisation and the demands of modern circuits cleaning has come back into fashion. Conformal coating is another process which has demanded special levels of surface cleanliness to guarantee coating adhesion and long term reliability. Although there are high reliability produces that use coating with no clean others want that extra confidence. Each company will receive a FREE set of coating/cleaning inspection and quality control wall charts which also cover defects seen during assembly.

Workshop topics include:

PCB Design for cleaning
Testing component compatibility
Flux compatibility with cleaning solvents
Solubility of soldering residues
Cost of process chemistry and equipment
Inline or batch cleaning options
Process capability, simple shop floor analysis
Water, semi aqueous or solvent?
Determining cleanliness standards
Ionic, SIR and visual inspection methods
Environmental requirements of cleaning
Cleaning no clean flux residues

Conformal Coating Applications, Inspection, Rework & Quality Control Workshop

(The workshop can also be run as a shorter online webinar)

The use of Conformal coating has provided benefits to industry for many years either in the high reliability market sector or where products have to deal with extreme environmental conditions or simply in use in consumer applications. The use of coatings is seen in different industries like telecommunications, automotive and consumer products and these industries have benefited from the use of selective coating but for different reasons.

This workshop will provide a simple guide to the use of coatings, their application and process, product benefits, inspection and quality control. A practical session will also allow delegates to examine coated boards using different materials and inspect the coating application.

Workshop Topics Included:

Why Conformal Coat
Clean or No Clean
Coating Material Options
Coating Process Options
Cost of coating assemblies
SIR and cleanliness testing
Cleanliness testing methods
Reliability of Coating
Testing & Evaluation of Coatings
Correct design for coating
Masking options and methods
Inspection & Quality Control of Coating
In-house or Contracting Services
Inspection of coatings & methods
Rework & repair of board assemblies

For further information or to book a workshop or webinar contact info@smartgroup.org

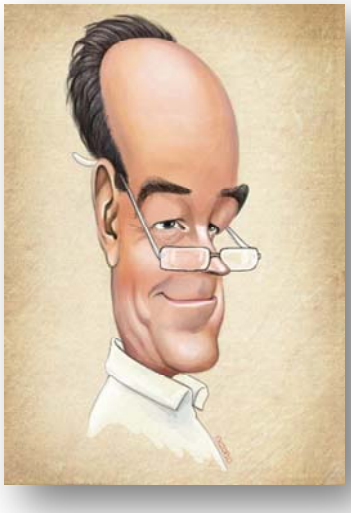
Author's Profile

Bob Willis currently operates a training and consultancy business based in UK and has created one of the largest collections of interactive training material in the industry. He is a member of the SMART Group Technical Committee. Over the years Bob has been Chairman and Technical Director of the SMART Group and holds 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, SMART, IPC and recently EIPC. Although a specialist for companies implementing lead-free manufacture Bob has provided worldwide consultancy in most areas of electronic manufacture over the last 30 years. 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 Reflow and Wave Soldering Workshops in Europe for one of the largest suppliers of capital equipment. 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. This has earned him the *SOLDERTEC/Tin Technology Global Lead-Free Award* for his contribution to the industry. He has also been presented with the *SMTA International Leadership Award* and *IPC Committee Award* for contribution to their standards activity

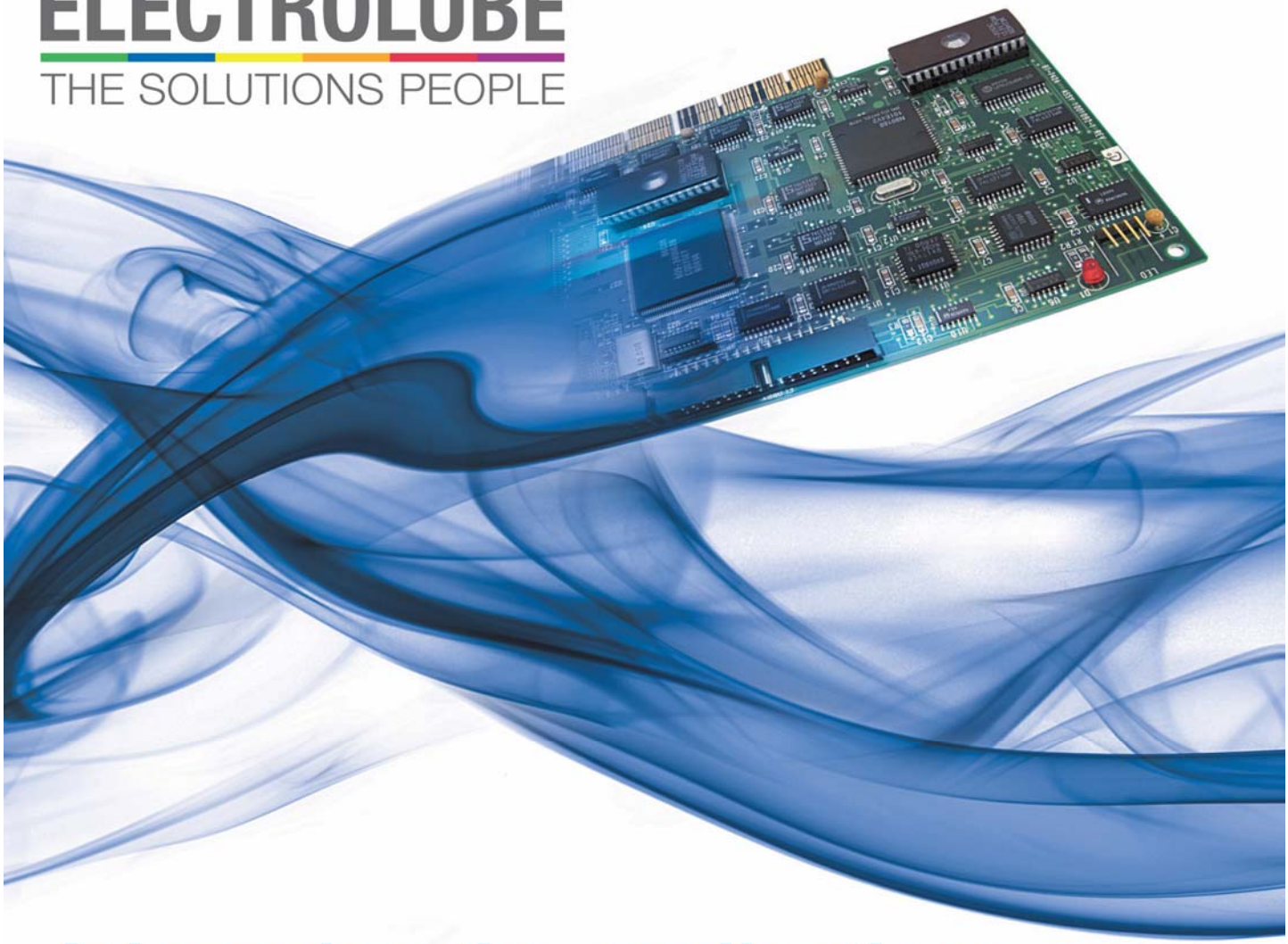
He has also run training workshops with research groups like *ITTF, SINTEF, NPL & IVF* in Europe. Bob has organised and run lead-free production lines at international exhibitions *Productronica*, *Hanover Fair*. *Nepcon Electronics* in Germany and England plus IPC APEX and SMTA International in USA 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 chip and through hole intrusive reflow connectors. In September 2016 Bob was presented with *Best Speaker at SMTA International Conference* 2016 in Chicago

He has 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. Following his time with GEC he became Technical Director of an electronics contract manufacturing company where he formed a successful training and consultancy division



ELECTROLUBE

THE SOLUTIONS PEOPLE



Advancing the applications of conformal coatings

Our knowledge of the wider industry, combined with our ever advancing approaches to research and development, have created a range of Conformal Coatings with the necessary high performance and reliability required for the military, aerospace and commercial industries.

Although every electro-chemical solution is in our spectrum, our specialist approach to problems has established us as a key provider and developer to the technical electrical industry, both commercial and domestic.

Isn't it time you discovered out how Electrolube can serve you?
Simply call, or visit our website.

Tel: +44 (0)1530 419600

www.electrolube.com

Scan the code to discover our full spectrum of superior Conformal Coatings and electro-chemical solutions.

Electronic & General Purpose Cleaning	Conformal Coatings	Encapsulation Resins	Thermal Management Solutions	Contact Lubricants	Maintenance & Service Aids