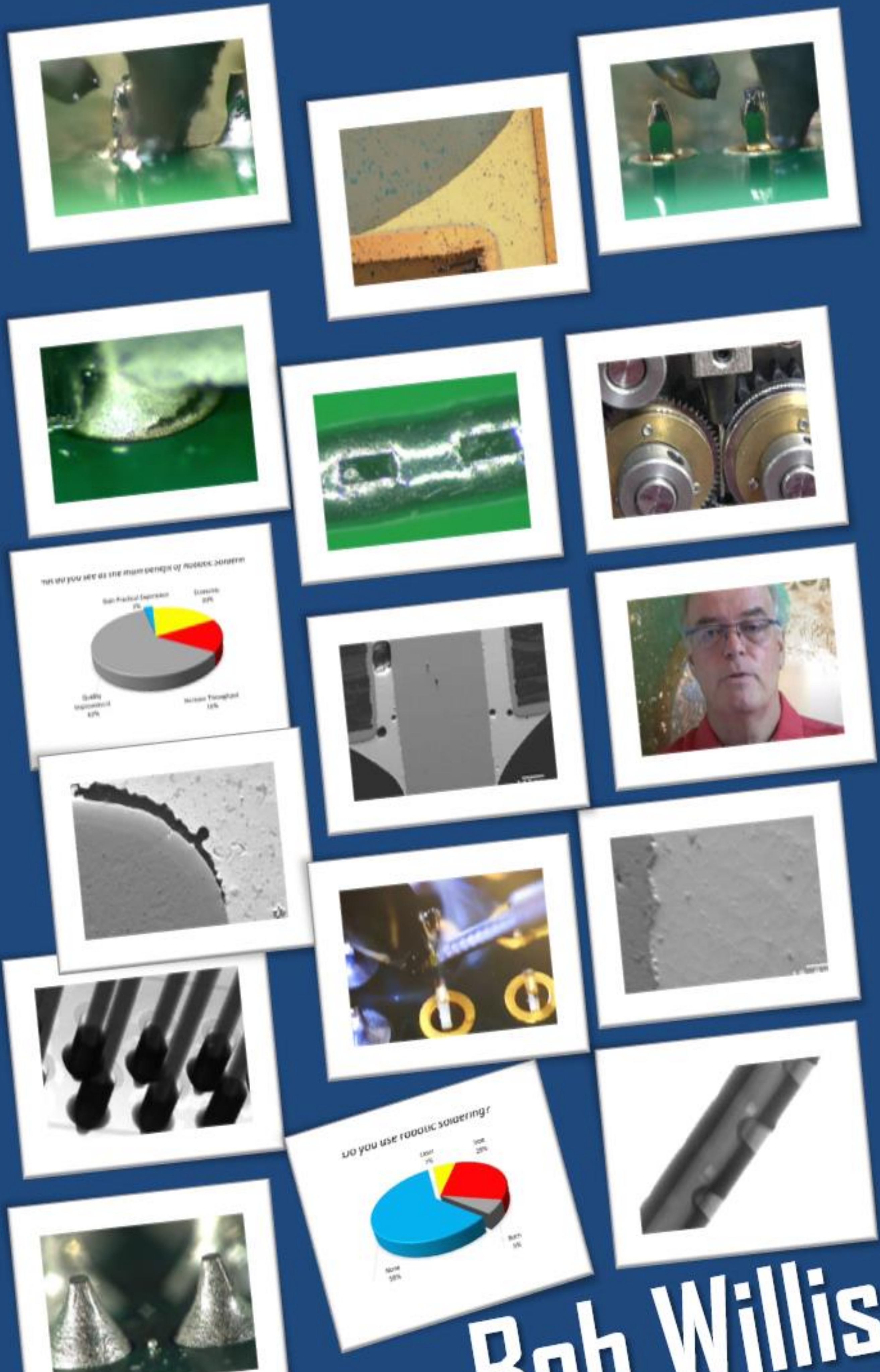
Robotic Soldering – Inspection & Process Defect Guide



Bob Willis

Contents

Bob Willis' New Soldering eBook
Foreword from Keith Bryant & Pete Starkey


I am greatly honoured to be chosen as ONE of the people to write the foreword to Bob's latest book

I have known Bob since the early days of Surface Mount, before 'lead-free' QFPs, balled devices and many other technologies which the industry and I became familiar with, thanks largely to Bobs expertise and hard work. I have had the pleasure in working closely with him on projects including PIHR, so I can say with confidence he is a guru in our industry and shares his vast knowledge with all who ask. He puts 110% into everything he does, so I know that, like his many other works, this will be a great read, full of useful information, but more importantly a full dive into the technology, explained in an easy-to-follow style.

Keith Bryant

If you want to know about soldering, ask Bob Willis!

Robotic soldering is not new – the earliest reference I found dated back to 1979 – and there is no shortage of information on the subject. But if you want an objective guide, written from a practical hands-on point of view by an acclaimed industry expert whose expertise in the technology is matched by his ability to communicate effectively, then you will find Bob's latest eBook an invaluable resource.

The Robotic Soldering, Inspection & Defect Guide covers all aspects of the technology, with a comprehensive introduction to available processes, solders, fluxes, pastes and design considerations, plus some tricks of the trade in tooling for assembly, and extends to more than 80 pages.

Bob also discusses inspection methods and the causes and cures of process defects, pointing you in the right direction to resolve whatever may be the issue, all with the help of multimedia and video links.

Not only is the book free to the user, Bob has encouraged the support of sponsors whose financial contributions all go to his nominated charities.

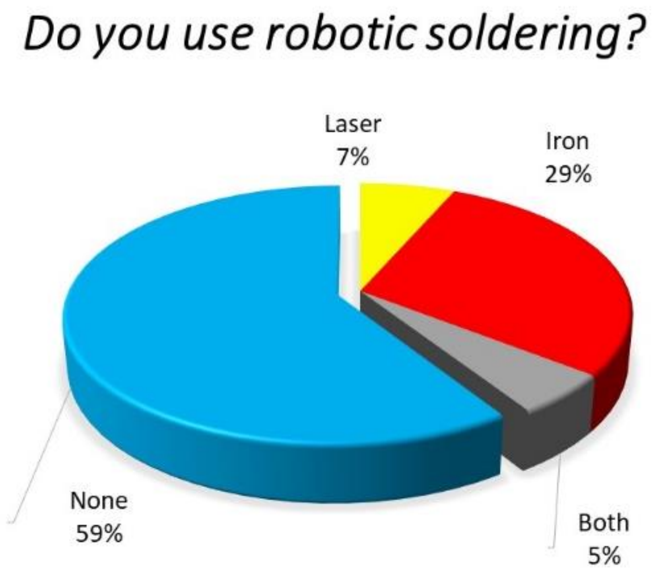
Bob Willis has operated a very successful training and consultancy business over the last 35 years and has created one of the largest collections of interactive training material in the industry. I am proud to have cooperated with Bob for many of those years and to have served with him on the Council of the Institute of Circuit Technology and the Technical Committee of SMART Group (latterly the European chapter of SMTA).

I recommend that you take advantage of Bob's generosity in sharing his extensive knowledge and experience

Pete Starkey

Introduction

There have been lots of discussion on the potential use of soldering robots for Printed Circuit Board (PCB) assembly but in all honesty, they have already been used widely over the past 10 years. I was recently very interested to see robots being used to load and unload full size process panels on a vertical solder levelling systems in a PCB manufacturing facility. That's not a fun job, hot and hard work, so why not let a robot take the strain. A recent study confirmed an increase of 140,000 new installations in China during 2019. In Europe the leading country in terms of new installations was Germany with 28,000 which was nearly double Italy (next highest) for the same year. Now of course these are not all for electronic assembly but it is an indication of the appetite for new technology and the increasing number of suppliers.



The use of Low Temperature Solders (LTS) is also not new; however, its widespread use in PCB assembly has been limited. Tin/Bismuth/Silver was considered, then dismissed during the early introduction of lead-free as the alloy options were more brittle, so Sn/Ag/Cu (SAC) became the go to alloy avoiding lead but at much higher process temperatures and significant cost. Japan did use LTS materials in many consumer and office based products during their very successful introduction of lead-free technology between 2000-2004. Suppliers have been working hard to find the best combination of SnBi and other elements to balance the reliability of the final joints, its peak reflow temperature and material cost. Some suppliers are moving to more reliable versions of the LTS products but at high soldering temperatures.

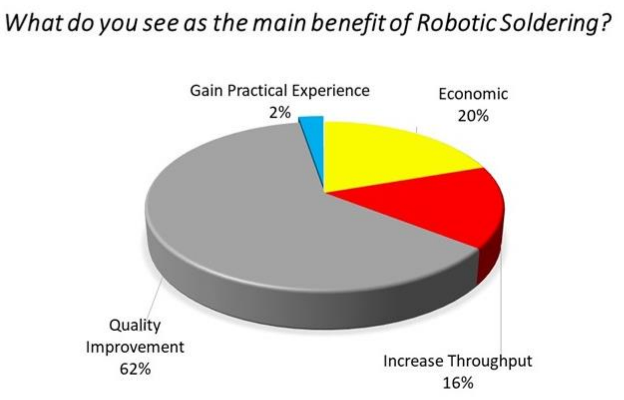
So what are the benefits of LTS to our printed circuit board industry and designers; well potentially less issues with moisture as soldering temperatures can be reduced. Less impact on solderability of selected surface finishes like Tin and Organic Solderability Preservative (OSP) during multi step soldering and less delamination and warpage issues.

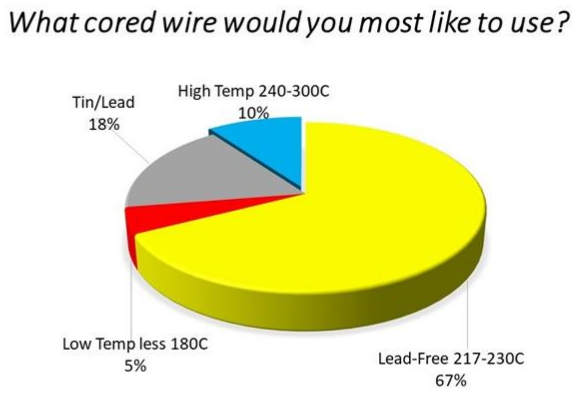
The iNEMI 2017 Roadmap indicated the share of the solder paste market for LTS would be 12% in 2022 and 20% by volume in 2028. They also demonstrated the cost advantage in assembly from energy savings. A number of prominent producers have also highlighted the potential for product cost saving.

On a personal level I have had the opportunity to work with high temperature and low temperature alloys with both robotic contact and laser soldering with work contributing to National Physical Laboratory (NPL) report "Practical Guide to Soldering PCBs with High Temperature Solder Alloys". This work in 2015 with Dr Chris Hunt provided a greater interest to organise other events and continue further research into the use of robotic soldering, the benefits and understanding of the issues that users would have to face in its implementation. This has helped create the opportunity for the ***Robotic Soldering Experience***, a multi company project to demonstrate the practical use of this new soldering technology. Despite all of the efforts to run the workshop in 2020 COVID made it impossible to hold the event. I will always appreciate the support from **Claire & Rob Saunders** of **WNIE.online** in helping to launch this production feature, however **COVID 1 - Robotic Soldering Experience 0**

My own surveys conducted during online webinars in 2021/22 indicated that the main reason for using robots was for quality improvements rather than economic savings which most people would not necessarily assume.

Claire and the team have been very supportive and generous to this charity project





Robotic Soldering Process Steps

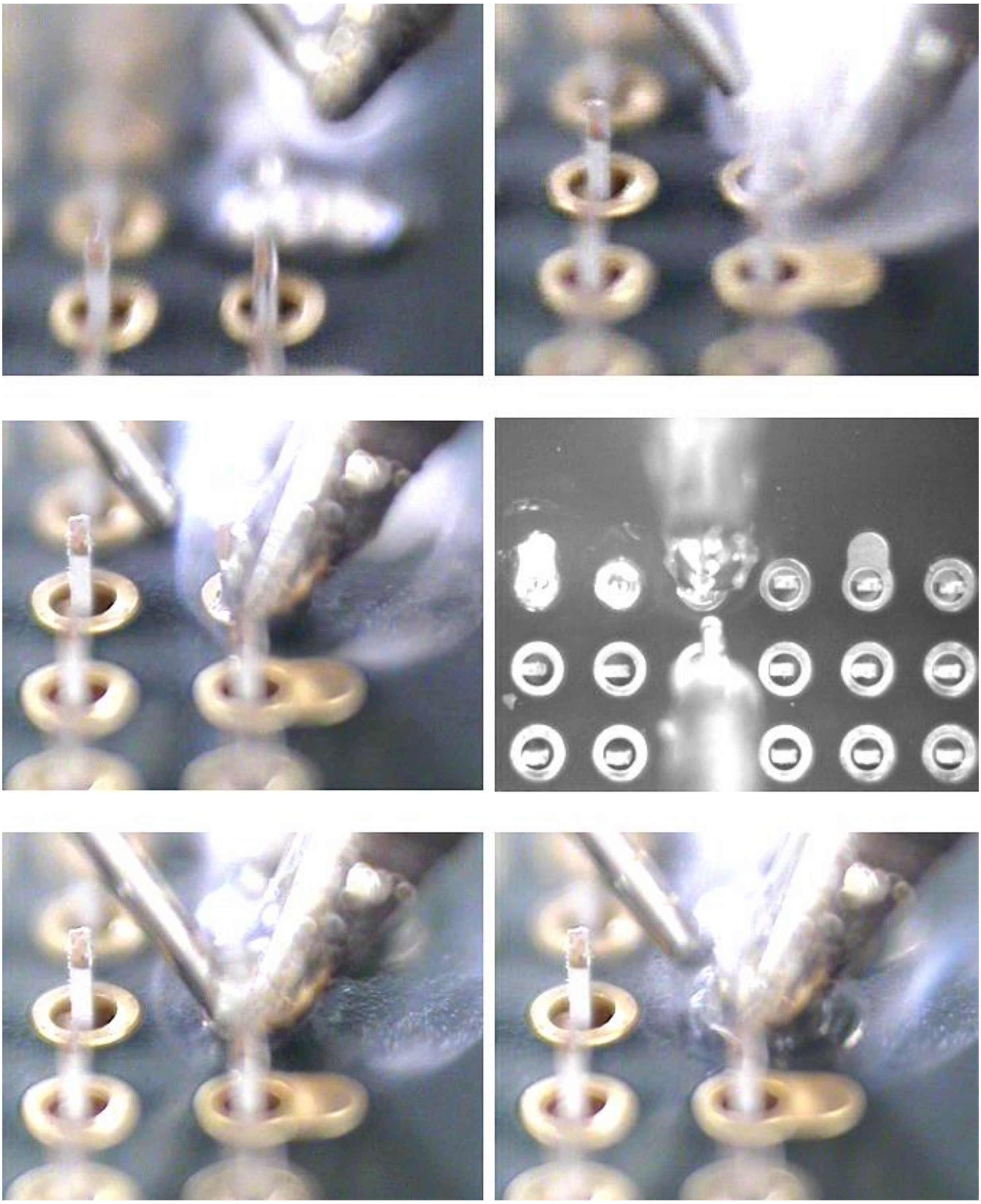
In traditional printed board assembly incorporating surface mount and through hole there will be mixed assembly/soldering process. This may include solder paste printing/jetting, component placement then reflow soldering on side one with the possibility of repeating this on the second side. If through hole terminations require soldering then either manual hand soldering, automated wave, selective or intrusive reflow would be considered. This will all depend on volume, the component type and the layout of the printed circuit board. In recent years other factors have come into play like limited availability of trained staff, energy and running costs of selective or wave soldering. This is not to mention the floor space taken up with equipment.

If loose components are to be inserted into the board, and subsequently soldered using a robotic system, parts must be retained in place consistently. The body must be held flat to the board. Tall electrical and mechanical parts should also be held flat to the board surface when the board is inverted for soldering. If components do move their lead position will vary and may cause problems during soldering. The component will often not be held flush with the board surface and will exceed the tolerances in IPC standards like IPC 610 “Acceptability of Electronic Assemblies” even if the quality of the joints is satisfactory. With correct engineering and setup there is no reason why the requirements or IPC 610 level 3 cannot be exceeded with robotic soldering systems.

Typically if component leads are not locked in place an assembly/soldering pallet will be required. This holds the board edges and then a top hat plate can be placed over the board to hold parts in place. Sometimes the pallet is also the tooling plate used for alignment of the board during soldering. This plate may also act as a carrier that passes through the automated assembly line for other process steps. It should hold the board flat and not allow any bounce or deflection when a soldering iron tip contacts the board surface or component termination. If the soldering system has visual alignment cameras to check PCB fiducial marks then any top hat plates must not obscure the marks from the camera when viewed from the topside.

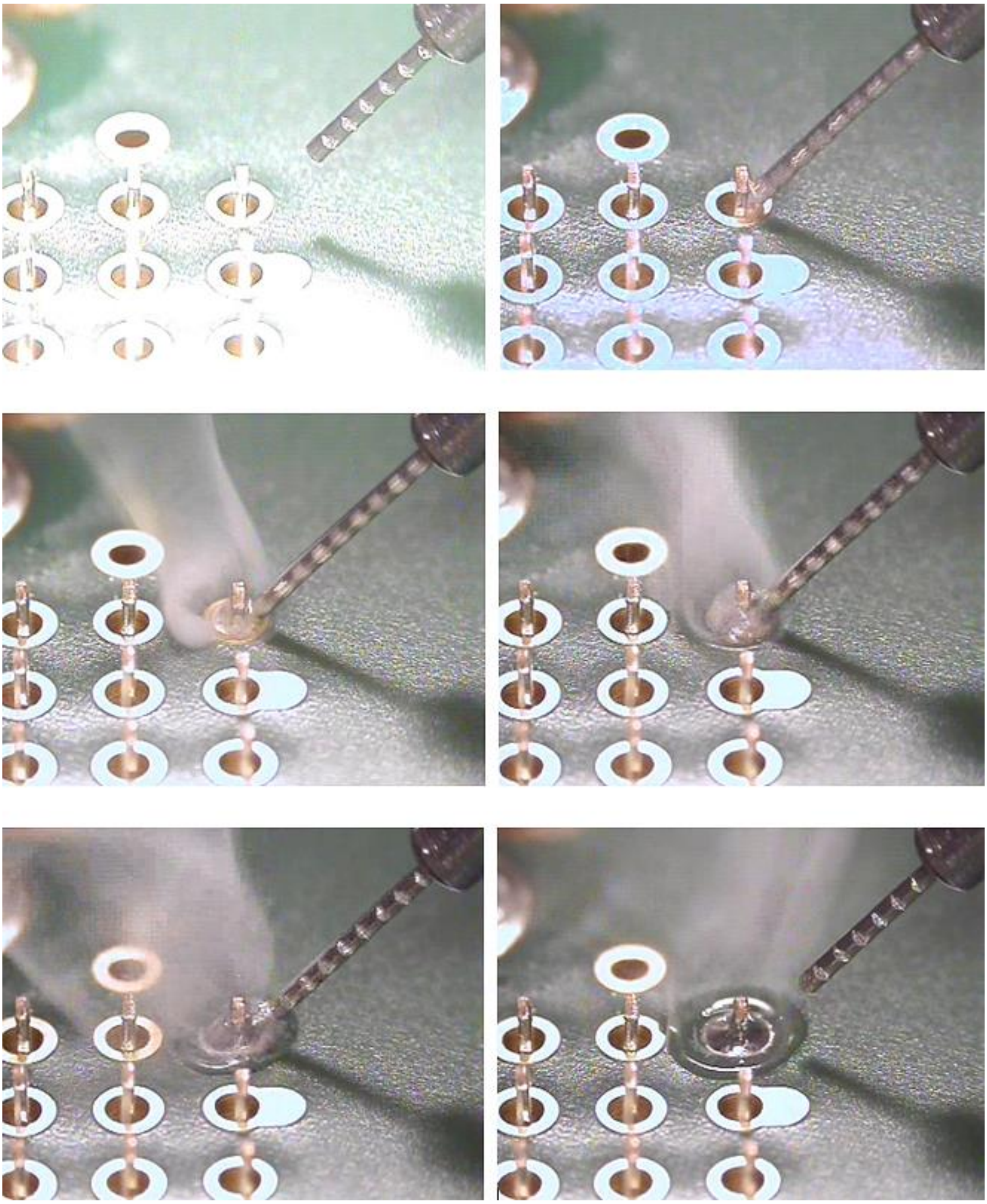
If after all of the soldering steps have been programmed into the robot, then soldering can proceed. Programming the robot movement would traditionally be undertaken with a pendent keyboard. Alternatively design data can be fed into the machine, or a master board image used as a reference for manual teaching.

Contact Soldering Option



Images above show the steps with a robotic iron tip conducting the soldering process. These steps can vary based on different machine types but fundamentally are the same steps to achieve satisfactory joints. Step/image 1 Tin the tip with cored wire, 2 Move tip to joint area, 3+4 pre-heat joint area, 5 feed solder wire, 6 retract the cored wire then iron tip and move on to the next joint.

Laser Soldering Option



Images above show steps with robotic laser soldering and were taken from videos recorded during the author's trials. These steps may vary slightly based on different machine types but fundamentally are the same to achieve satisfactory soldering. Step/image 1 pre-heat pad & lead, 2 pre-heat and feed the solder wire, 3-4 reflow and continue feeding the wire, 5-6 retract the solder wire then move to the next joint.


Laser
Vertical
Multiple-Joint
Desktop
Horizontal
Multiple-joint
In-line
Robotic Soldering and Automation
JAPAN UNIX
GLOBAL NETWORK
HQ : JAPAN
CHINA | S. KOREA | MALAYSIA | TAIWAN
MEXICO | U.S.A. | INDIA
Care for tomorrow
Always master soldering, with cutting-edge solutions from Japan Unix – today, tomorrow, and under any circumstances.
JAPAN UNIX CO., LTD.
robot

Examples of soldering set point parameters used in one of our trials with a 96 way connector test board are illustrated below. These were determined in discussion with equipment suppliers to achieve satisfactory filling of the plated through holes with their equipment. The machine parameters are listed below for example only. It is very important to define the process parameters used for soldering and determine their repeatability during production trials. It is inevitable that the design of the board and the inner layer connection points will impact the solder fill due to the thermal drain. It is often the case that adding a preheat system will improve hole fill and, with it, faster processing on higher mass boards. Often solder joints are part of a prefabricated unit not just a printed board and can also impact heating and soldering times. Through hole fill is the key parameter in terms of inspection and in process quality control but inspection of the board, copper plating, laminate, pad and solder mask for damage are also important.

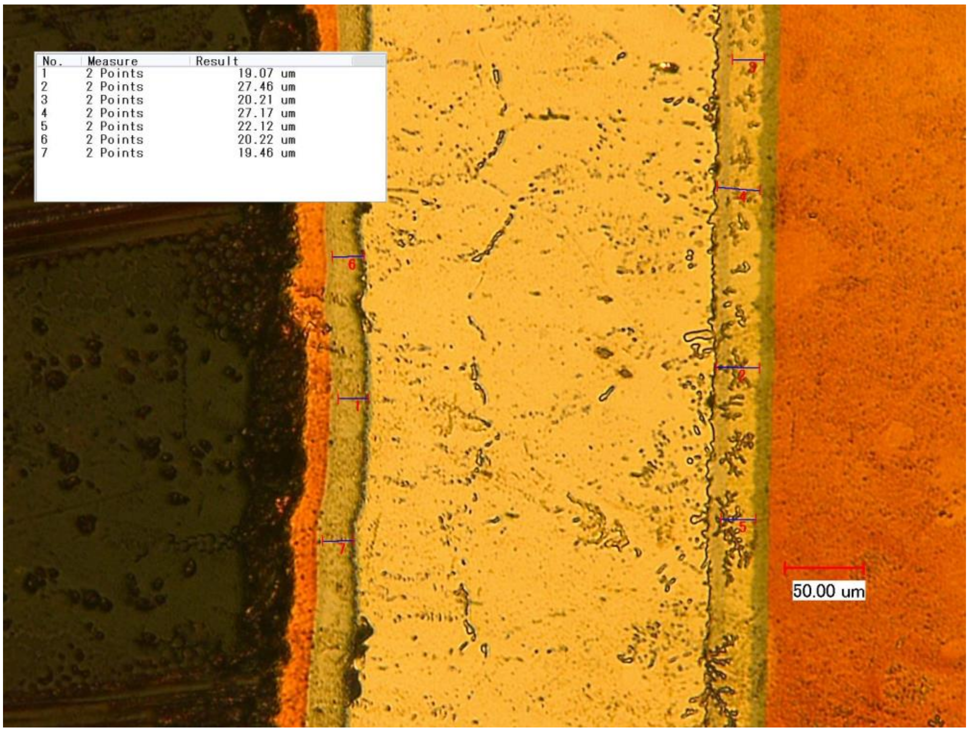
After initial trials and comparison of results the parameters can be modified. If solder hole fill is impacted by the design of the board the parameters can be changed. If all of the joints show 100% fill and full wetting temperature and hold times could be reduced to find the sweet spot. There is no point in just using too high temperature just because it works first time on your boards.

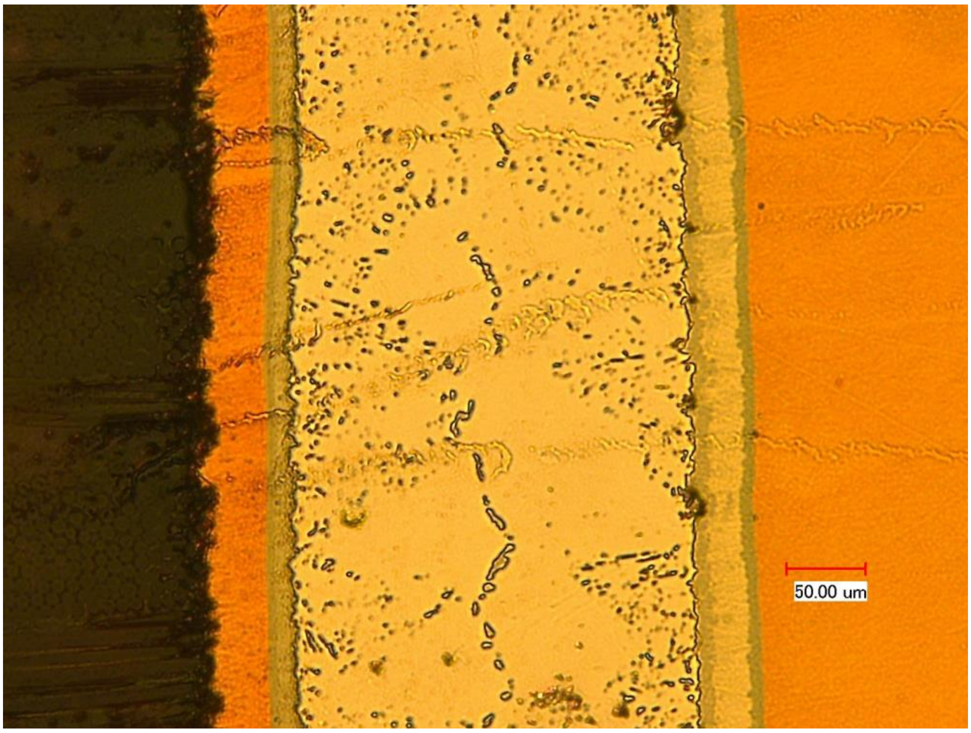
Contact Tip Soldering (Example Only)

Soldering iron tip size 2 mm
Solder iron temperature setpoint 385^{o}C
Pre solder feed 0.4s
Pre heating 0.8s
Solder feed 0.4s
Solder wire feed speed 75%
Solder wire pull back 0.1 mm

Laser Soldering (Example Only)

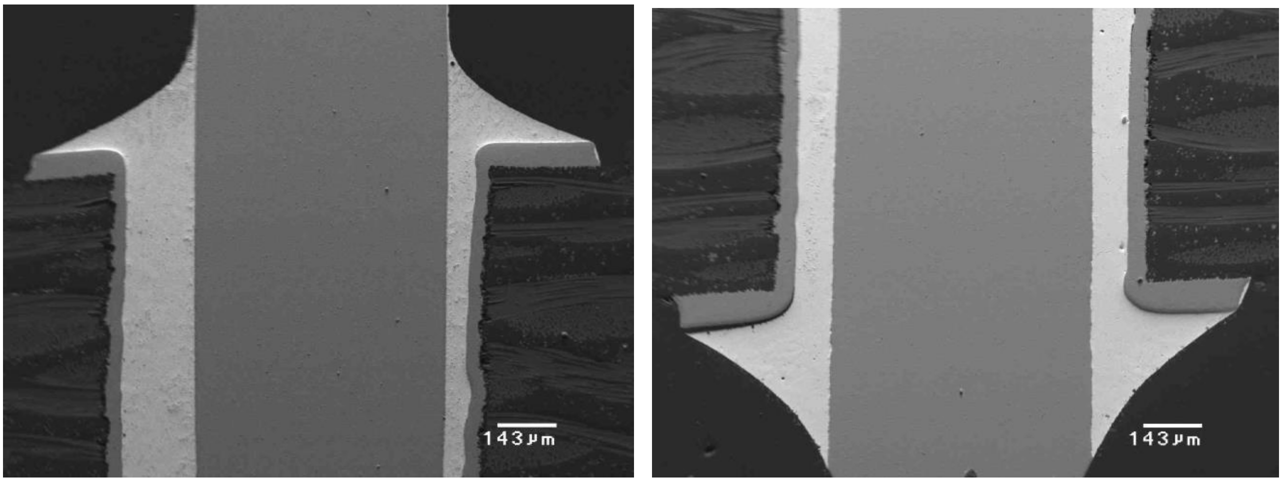
60 watt single emitter 940nm wavelength
Minimum light diameter 0.5 – 0.8 mm
Preheat 0.0s with 0 watt
Solder feed 0.6s with 25 watt
Hold time 0.2s with 25 watt
Solder wire pullback 0.1s
Solder wire speed 100%



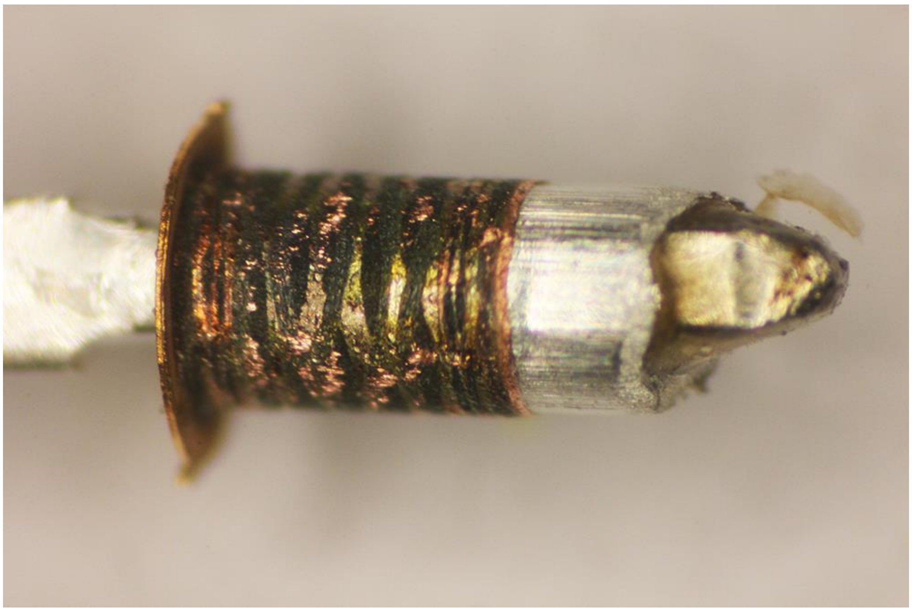


Two examples of successful solder joints examined during trials using microsections. This is the most common technique to evaluate solder joint wetting to the plated through hole and pin, solder joint intermetallic thickness and copper through hole adhesion/pull away can be assessed.

Microsections can be further examined using Scanning Electron Microscope (SEM) for further detail or to determine the individual elements of the joint before and after environmental ageing. In addition solder joints can be assessed by mechanical pull test to compare the impact of ageing on different material combinations. There are no international standards but it is effective as a comparative test technique.

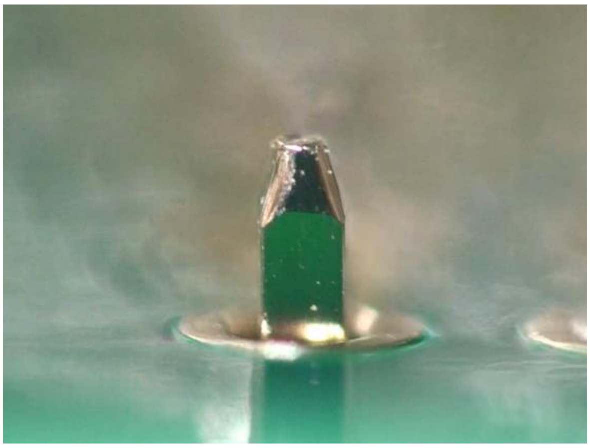
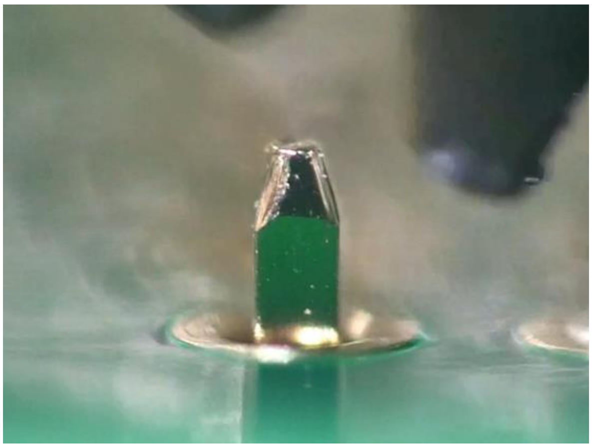


Microsections examined under SEM with lead-free fillet lifting visible on the solder side of the board. This is quite common and not unique to robotic soldering; it is seen on wave, reflow and selective soldering.



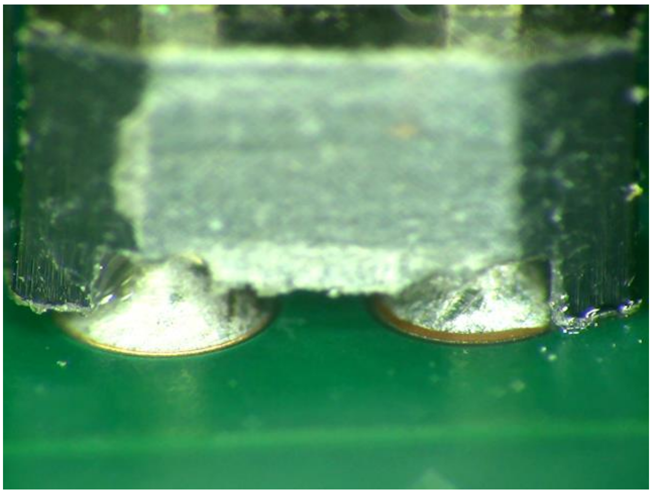
Individual termination/solder joint after pull test. The barrel of the copper plated through hole has been pulled out of the PCB laminate. The solder fill in the 1.6mm thick board had in excess of 75% fill with tin/silver/copper.

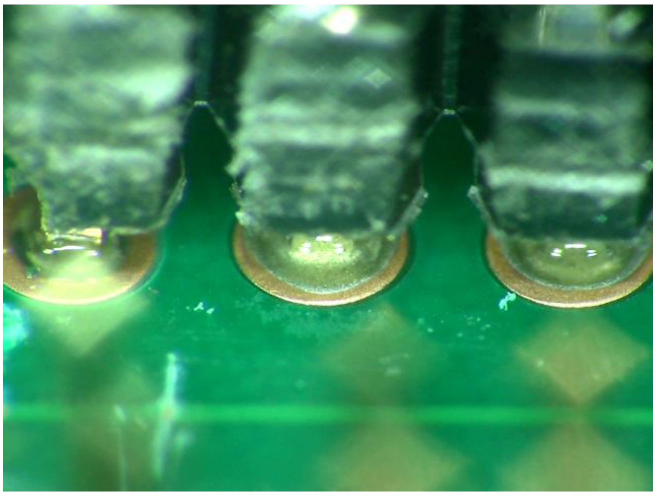
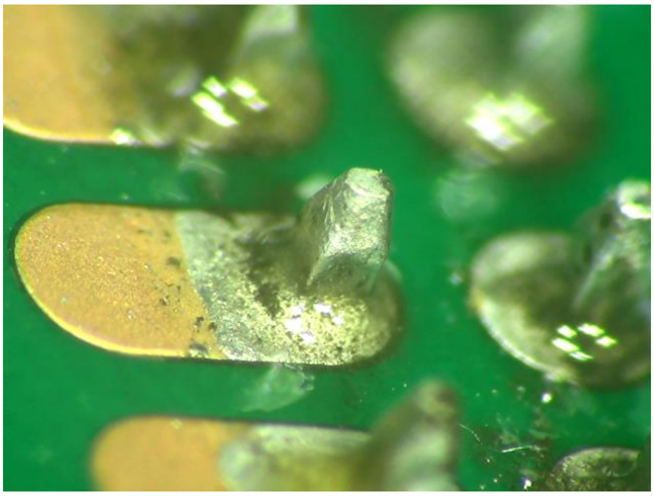
The following images show a successful soldering sequence conducted by the author in the UK with Low Temperature Solder (LTS) cored wire. Apart from changes to soldering temperatures and the need to consider handling of the brittle wire the results were very successful. Although LTS may not be a first choice for robotic soldering it can work successfully as an alternative to high temperature alloys. Sample joints were inspected by X-ray, microsection and thermal cycling to confirm the quality of the joints.



The following images are from our sample test board for the **Robotic Soldering Experience Project** successfully soldering with LTS wire. The board was 1.6mm thick with two different surface finishes, gold over nickel and copper Organic Solderability Preservative OSP. The holes featured a range of different thermal break connections to inner layers. They were produced with the same process sequence illustrated in the previous page. The set point temperature of 265^{o}C and an average dwell time of 2.5 seconds.

Robotic Soldering Sequence

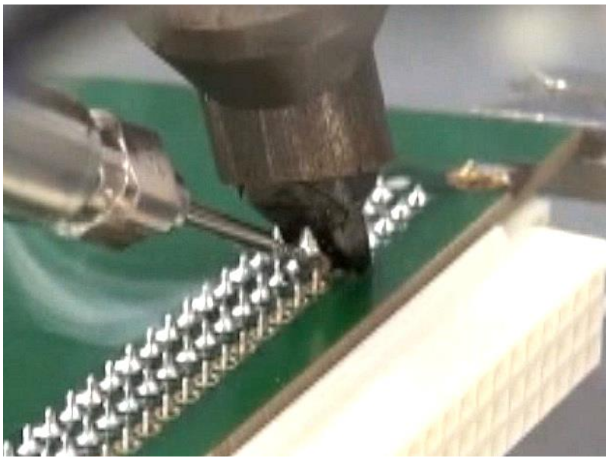
There are different options for the programming steps for the robot. First the soldering iron tip is cleaned prior to soldering. The tip is then tinned before approaching the point of soldering and placed on the joint area contacting the pad and potentially the pin. As the tip has been tinned heat transfer takes place more quickly.

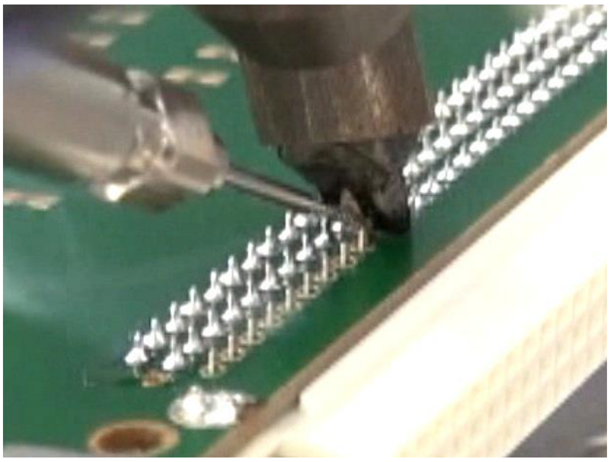
Visible wetting speed will depend on the PCB surface finish, if the board is solder levelled then the apparent speed of wetting will be very fast due to the thin layer of solder on the pad surface. In the case of gold, tin or copper OSP it will appear to be slower. This can also be true for the component pins but predominantly they are tin plated or a gold flash over copper or nickel above the base layer of the pin. The next step is to add the required amount of solder to the joint. There is a delay or hold to allow the solder to initially wet the pad/pin and fill the plated through hole.

It is important during setup to closely observe this while the solder is wicking into the hole. The solder wire feed then stops and is pulled back from the joint surface. The next step is the iron tip moves to the next joint to repeat the sequence or moves to the tip cleaner station before re-tinning then to the hold position or the next joint. The frequency of tip cleaning or re tinning is dependent on a number of factors and will be adjusted by the engineer or operator based on the experience gained on the product, the cored wire flux volume being used, iron temperature and possible nitrogen.

The other process options are adding slightly more solder directly to the tip rather than just tinning the tip and adding the remaining solder to the joint area. This is the same as an operator would normally do to see the joint area is up to temperature. In the second case of adding more solder direct to the tip before contacting the joint can result in more solder balls. However, if the cored flux formulation allows for this with less volatility the process can be much quicker. A case for careful material evaluations with your supplier.

There are many different tip options with robotic soldering just like with manual soldering. The larger the surface area in contact with the pad and pin the faster the soldering operation and the possibility of decreasing the tip temperature. The majority of tips do have a flat face that contacts the joint area; however split or forked tips are very effective for a faster process sequence. Split or forked tips allow drag soldering along a row of vertical pins as illustrated below.





One of the author's test boards and a 96 pin connector being drag soldered in Japan with Sn/Cu/Ni cored wire using a robot from Japan UNIX. Drag or leapfrog soldering is becoming more common as it is a faster process than the standard point to point.

Alternatively a forked or half moon tip can be lowered vertically on to each vertical pin in turn with the solder alloy wetting between two points of the fork, or half the circumference of the pin contacting the length of the pin and the pad surface with melted solder. Cored solder wire is added to the fork tips during soldering to replace the volume which capillaries into the plated through hole. This process can be faster than soldering individual pins with horizontal movement but equal to drag soldering. However, it all comes down to the thermal demand of the board design and the component mass. It is an engineer's job to adapt the program to achieve successful soldering along with carefully selecting the best cored wire alloy.

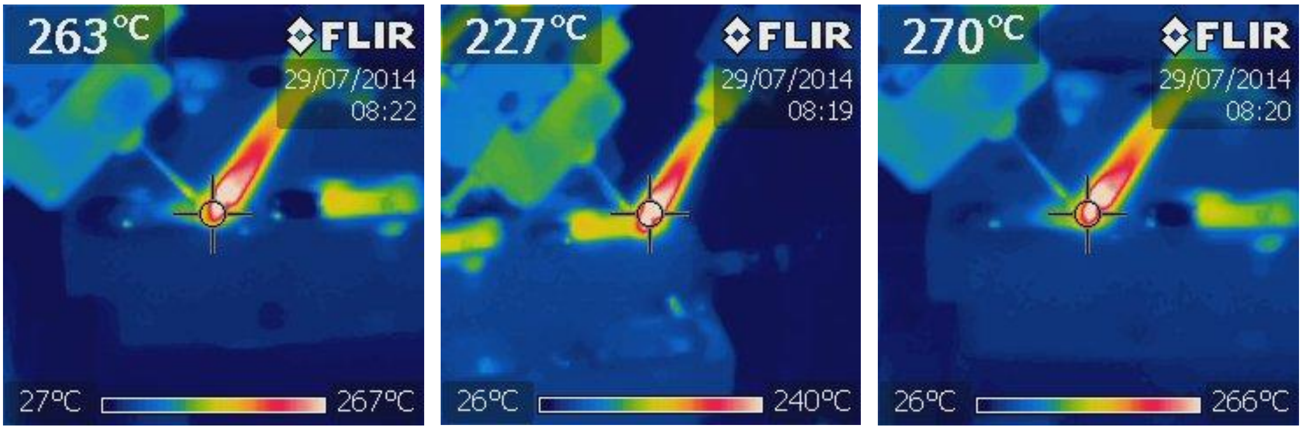
Forked tip soldering iron is positioned close to the pin and pad and is then lowered vertically to the pad surface. With greater surface contact between solder and forked tip the total process cycle time can be quicker.

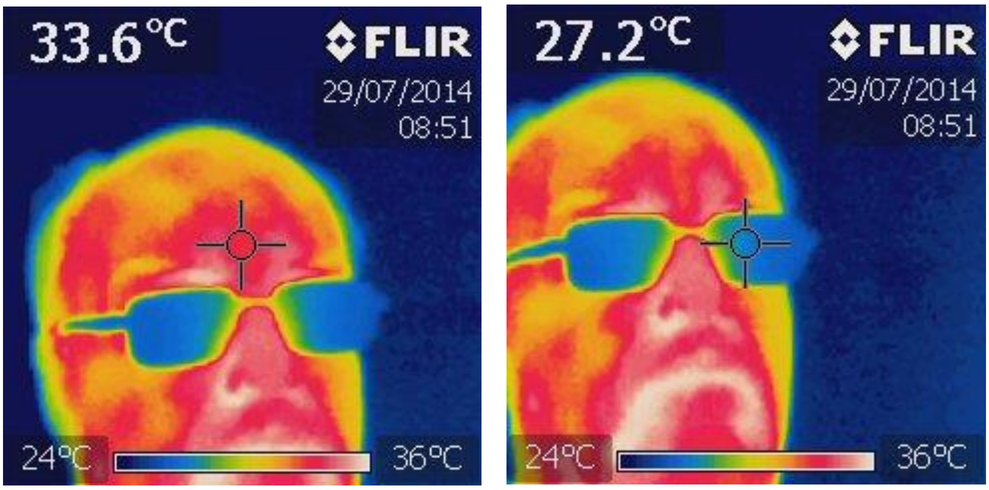
As with any assembly process it is necessary to ensure the process is checked regularly and before starting the soldering operation. Regular in process checks should take place as part of a **Process Control Monitoring System (PCMS).** Basically a process control engineer looks at different processes in manufacture. Checking examples of product and the process settings. If there is an issue, then engineering have to examine and correct the process within a set time otherwise the process is halted. Initial or periodic inspection steps on a robotic soldering work cell may include some of the following steps:

Run the soldering programme with solder wire feed turned off on bare test board
Check correct process parameters/programme are loaded for the PCB design
Correct size soldering iron tips are in good condition – Check joint/tip count rate
Correct solder wire alloy/size is loaded
Solder wire feed is working – Check wire indenting depth to the flux core
Inspect solder joints after first off production board then random samples hourly
Check nitrogen supply temperature around the point of soldering
Tip cleaning system is set-up and working – Moisten sponge if being used
If used check AOI programme for the PCB design currently running
Examine the AOI results for bottom side PTH joints – Reject rate and false calls
Random off line X-ray check of topside joints – Minimum 75% solder rise in PTH

Checking soldering iron temperature is possible directly on the tip of the iron with a thermocouple as a simple standard calibration/verification. In the past we have mounted thermocouples into through holes then soldered terminations and plated through holes. This shows the impact of heating, dwell and cooling in real time but also the benefit on preheating adjacent joints, slowly the temperature does rise in adjacent areas of the board. A temperature profiler normally used for wave and reflow soldering can collect all of the data from a row of terminations. In the past this has been useful to see if soldering all joints in sequence is best or to skip a couple of thermally demanding positions then returning to these joint positions is a benefit. It may be better than just increasing temperature or dwell time on selected joints.

Non-contact temperature measurement is simpler, quicker and more fun to use. Using a handheld IR camera can check the soldering iron tip or board before and after contact. We have also used it for checking the impact of preheating the board area with nitrogen flow from around the soldering iron tip. Changing temperature and gas flow rates to impact or improve the cycle time can be an additional benefit. Images below on a customer site show thermal measurement around a soldering iron with different flow rates of nitrogen from the collar surrounding the iron tip.

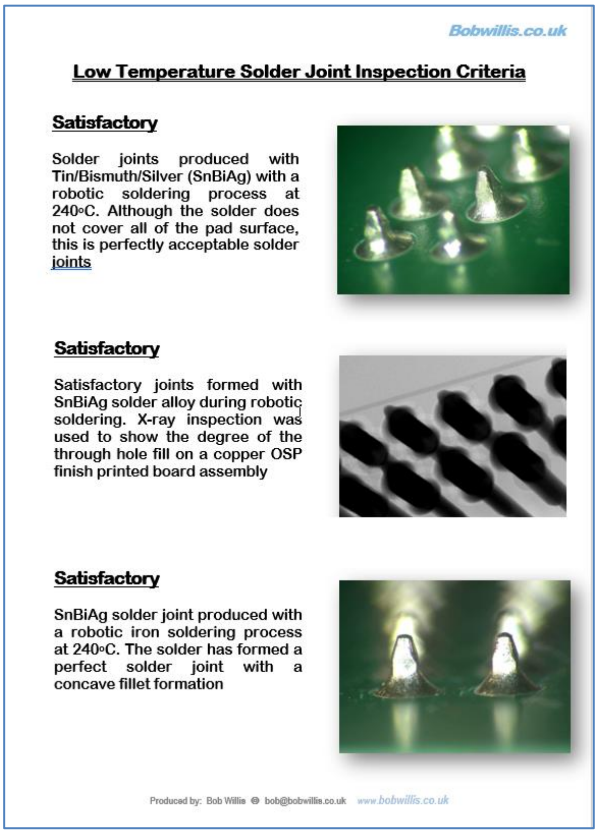




It may have been annoying for my customer, but I had to monitor my temperature changes while I was working very hard onsite with his robotic soldering process!

“Remember If you are considering robotic soldering as a future direction, why not upskill one of your production team to work with an engineer during equipment selection and installation. The production staff member should be highly skilled in hand soldering and solder joint inspection.

Once installed and running your staff member can run the machine and then progress with more machines to form a soldering cell within the company. Hopefully if they like their new role they can also train others in the future. It is more than likely the engineer will have found new production toys to play with“ – Bob Willis



Bobwillis.co.uk

Low Temperature Solder Joint Inspection Criteria

Satisfactory

Solder joints produced with Tin/Bismuth/Silver (SnBiAg) with a robotic soldering process at 240°C. Although the solder does not cover all of the pad surface, this is perfectly acceptable solder joints

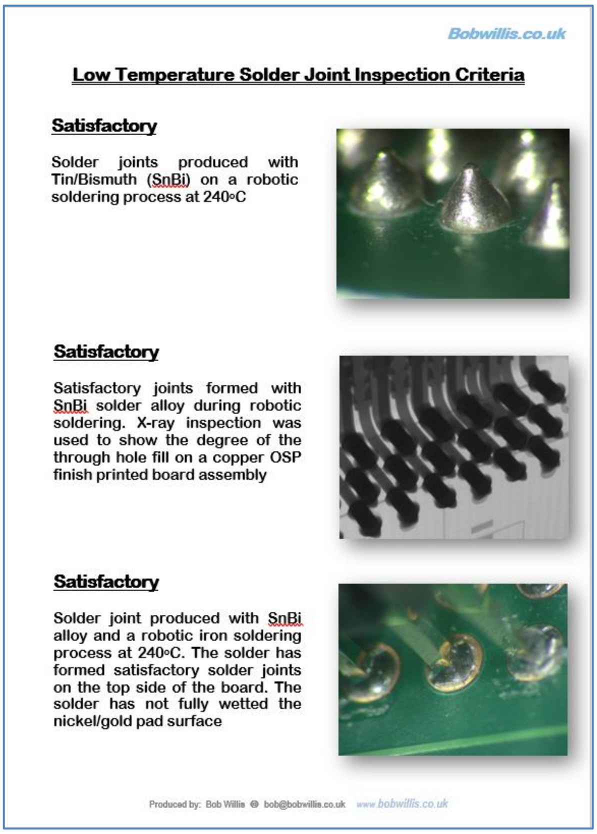
Satisfactory

Satisfactory joints formed with SnBiAg solder alloy during robotic soldering. X-ray inspection was used to show the degree of the through hole fill on a copper OSP finish printed board assembly

Satisfactory

SnBiAg solder joint produced with a robotic iron soldering process at 240°C. The solder has formed a perfect solder joint with a concave fillet formation

Produced by: Bob Willis © bob@bobwillis.co.uk www.bobwillis.co.uk



Bobwillis.co.uk

Low Temperature Solder Joint Inspection Criteria

Satisfactory

Solder joints produced with Tin/Bismuth (SnBi) on a robotic soldering process at 240°C

Satisfactory

Satisfactory joints formed with SnBi solder alloy during robotic soldering. X-ray inspection was used to show the degree of the through hole fill on a copper OSP finish printed board assembly

Satisfactory

Solder joint produced with SnBi alloy and a robotic iron soldering process at 240°C. The solder has formed satisfactory solder joints on the top side of the board. The solder has not fully wetted the nickel/gold pad surface

Produced by: Bob Willis © bob@bobwillis.co.uk www.bobwillis.co.uk

Bobwillis.co.uk

High Temperature Solder Joint Inspection Criteria

Satisfactory

Solder joints produced with tin/copper on a laser soldering process. Although the solder does not cover all of the pad, this is perfectly acceptable. There is variation in the solder volume per joint which is not uncommon with automated wire feed process

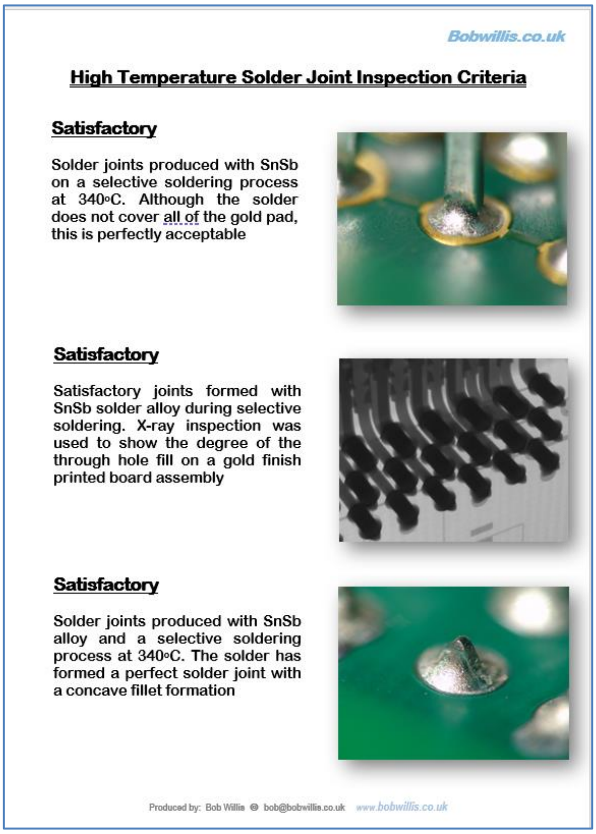
Satisfactory

Satisfactory joints formed with tin/copper with laser soldering. X-ray inspection was used to show the degree of the through hole fill on the printed board assembly

Satisfactory

Solder joints produced with tin/copper with laser soldering process. The solder has formed a perfect solder joint with a concave fillet formation. There is excess flux present on this example but if cleaning is used this would not be a problem

Produced by: Bob Willis © bob@bobwillis.co.uk www.bobwillis.co.uk



Bobwillis.co.uk

High Temperature Solder Joint Inspection Criteria

Satisfactory

Solder joints produced with SnSb on a selective soldering process at 340°C. Although the solder does not cover all of the gold pad, this is perfectly acceptable

Satisfactory

Satisfactory joints formed with SnSb solder alloy during selective soldering. X-ray inspection was used to show the degree of the through hole fill on a gold finish printed board assembly

Satisfactory

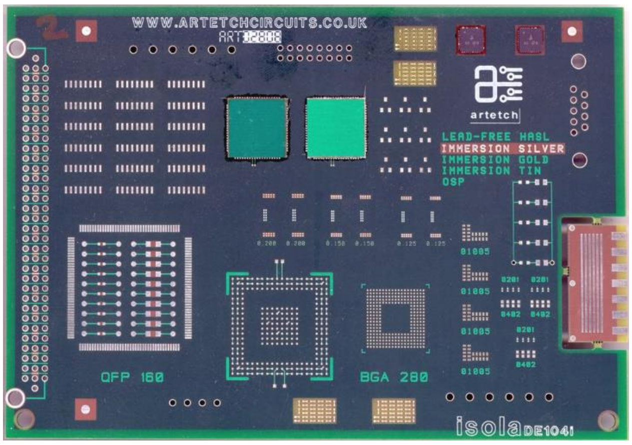
Solder joints produced with SnSb alloy and a selective soldering process at 340°C. The solder has formed a perfect solder joint with a concave fillet formation

Produced by: Bob Willis © bob@bobwillis.co.uk www.bobwillis.co.uk

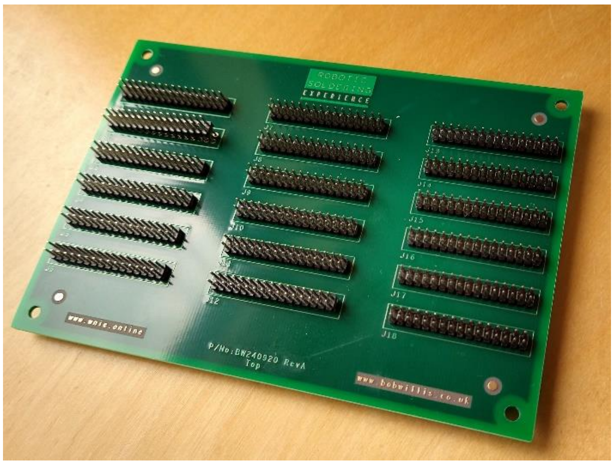
Our robotic soldering inspection wall chart sets are all available FREE to download via the smta.org website as part of our training media package sponsorship

PCB Design Considerations

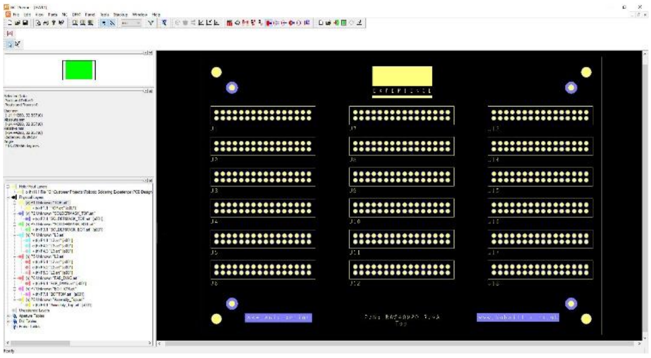
During our process trials, excluding work with customers, we have used two different multilayer board designs. These were produced from different laminate types, four and six copper layers with different surface finishes to examine wetting and join formation before and after temperature cycling.

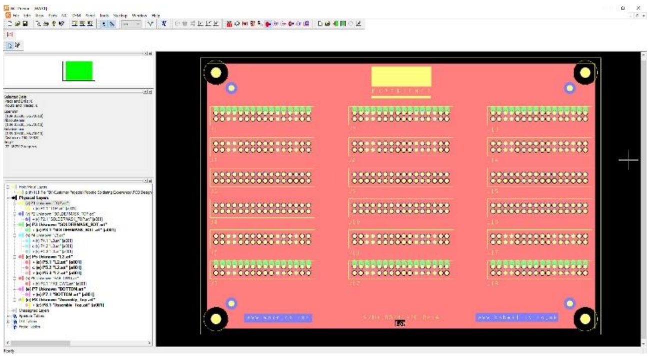


Our first board design above has been used for many other projects prior to robotic soldering. It featured 96, 16 and 6 pin post header connectors

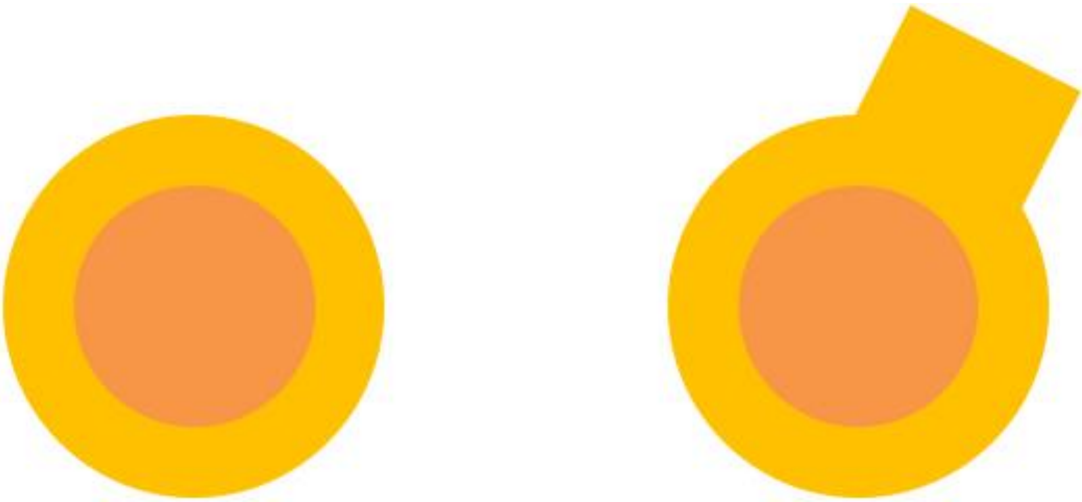


*The second design above was produced specifically for the **Robotic Soldering Experience Project** struck down by COVID. Gold, OSP copper and tin finished PCBs were used for the surface finish. This board design featured double row post header connectors with 32 pins. If you need test boards Gerber design files are available to assist your process evaluation.*

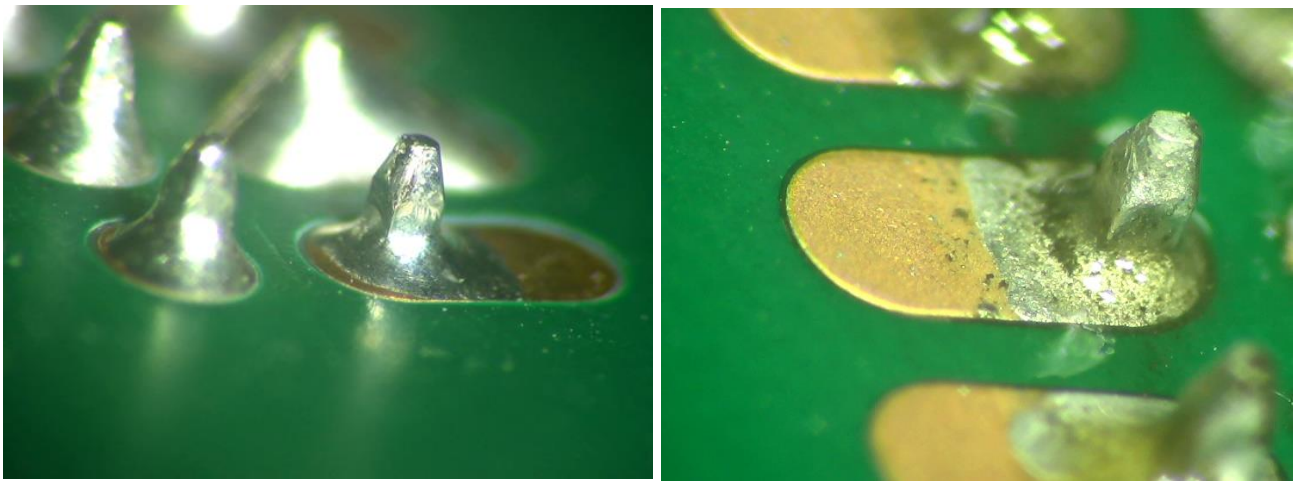




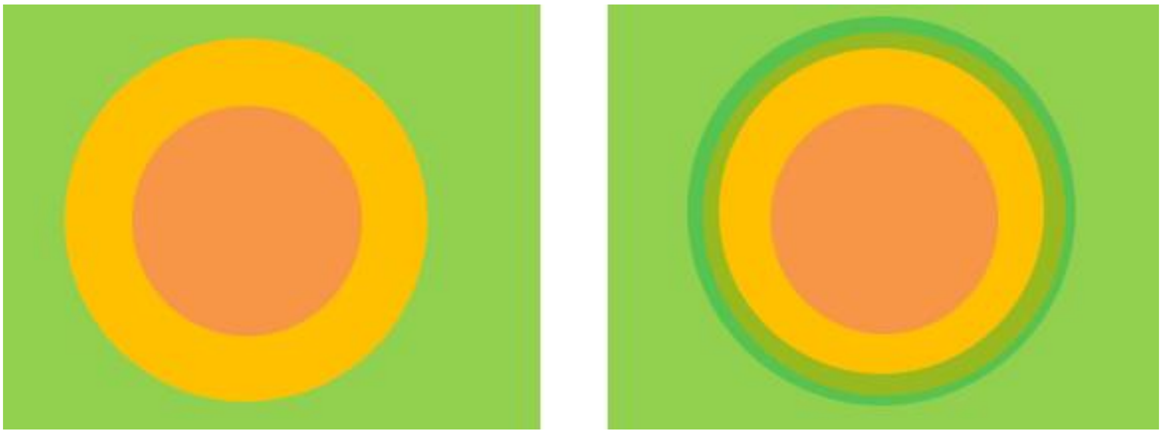
Selected pads on the bottom side of the board can incorporate teardrops. This increases the surface area for soldering iron tip contact. Some early and lower cost robotic soldering systems benefitted from larger landing areas and also avoided board or solder mask damage. There is also no necessity for the pad extension to be completely covered with solder provided the hole and pin are successfully soldered, ideally with 100% hole fill and positive solder fillets. This exceeds IPC requirements of IPC610 class 3.



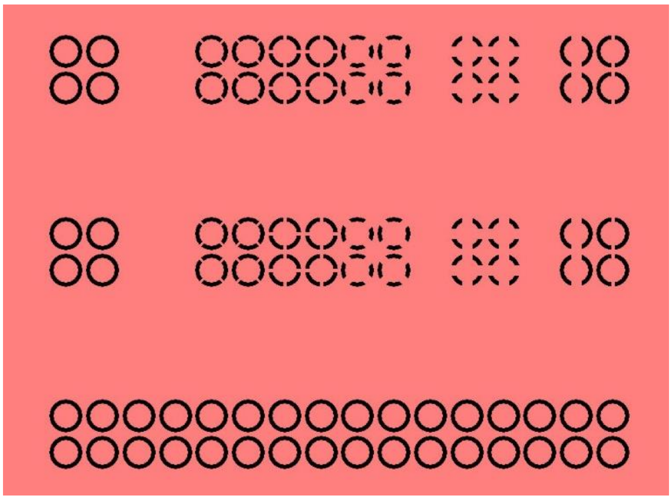
Example of pads on the solder side of the board. First a standard pad, the second with a pad extension to aid the robotic soldering and assembly process. The tab provides a larger area to land on to with less precise equipment.



Large pad extensions are shown above with perfect solder joints but with the original surface finish copper OSP visible. These are not considered defects, the teardrop pad design may be a little large but has no impact on the quality or reliability of the solder joints. There should be no reason to just add solder to cover the ends of the pads.



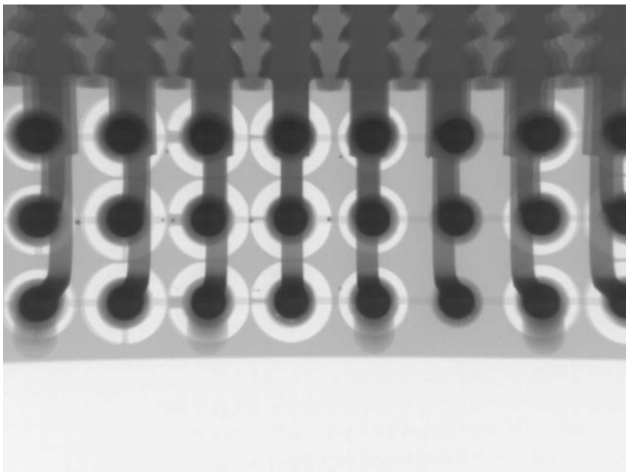
Solder mask design should remain standard with a clearance between the pad and the mask edge inline with your PCB supplier's capability. Alternatively, if there is a benefit, solder mask defined pads can be used. Not all suppliers are capable of maintaining the accurate location of the solder mask over the pads. Much of the error comes with the selection of larger PCB process panel sizes. Back when lead-free solder was being introduced for selective, wave and intrusive reflow one producer suggested that solder mask defined pads prevented solder fillet lifting. Our test results did show the solder mask did make it more difficult to see the solder fillet lifted from the surface of the pad. It's not quite the same thing!!

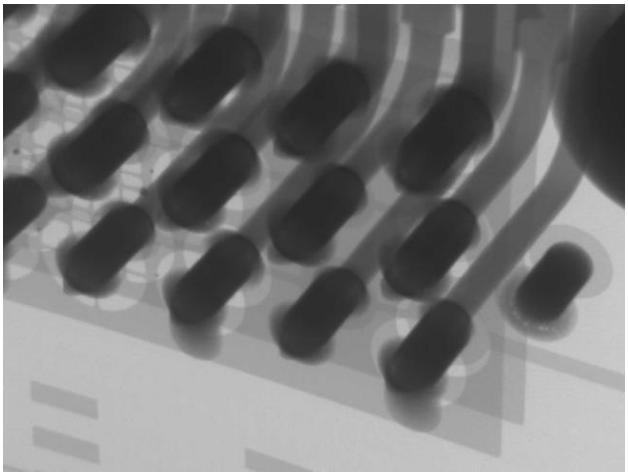


Inner layer connection points were varied on our test boards to impact the thermal soldering demand in trials. Different thermal breaks and inner layer openings were incorporated into the design and illustrated by the CAD layout.

To improve soldering/desoldering considered the following design features:

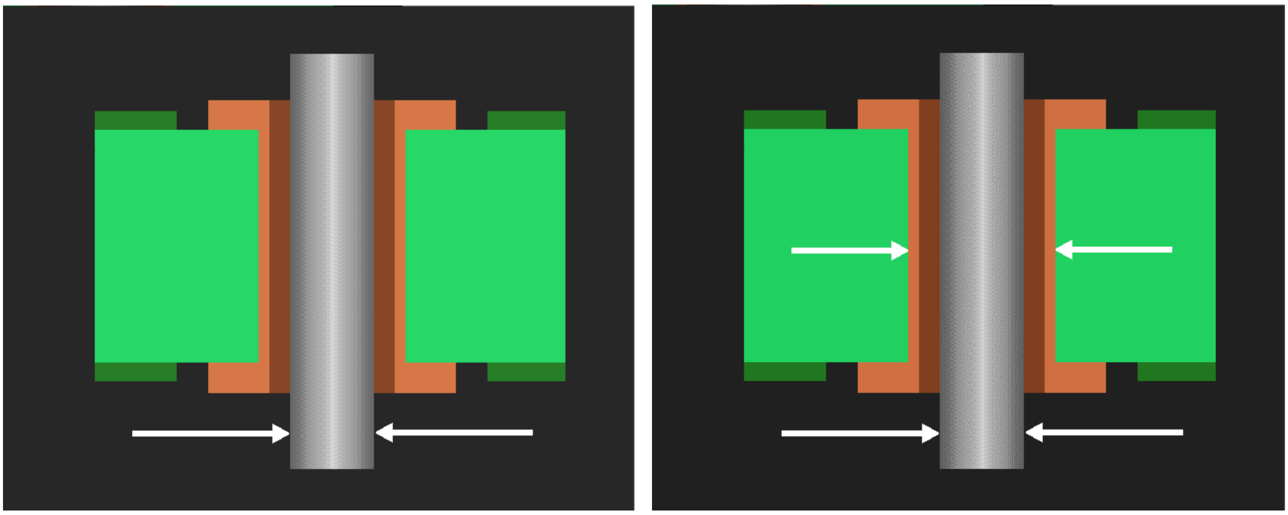
Reduce the width or number of the copper tracks between the hole and inner copper plane. If possible increasing the laminate gap between inner layer pad and copper plane will make soldering and desoldering, if necessary, easier for staff.



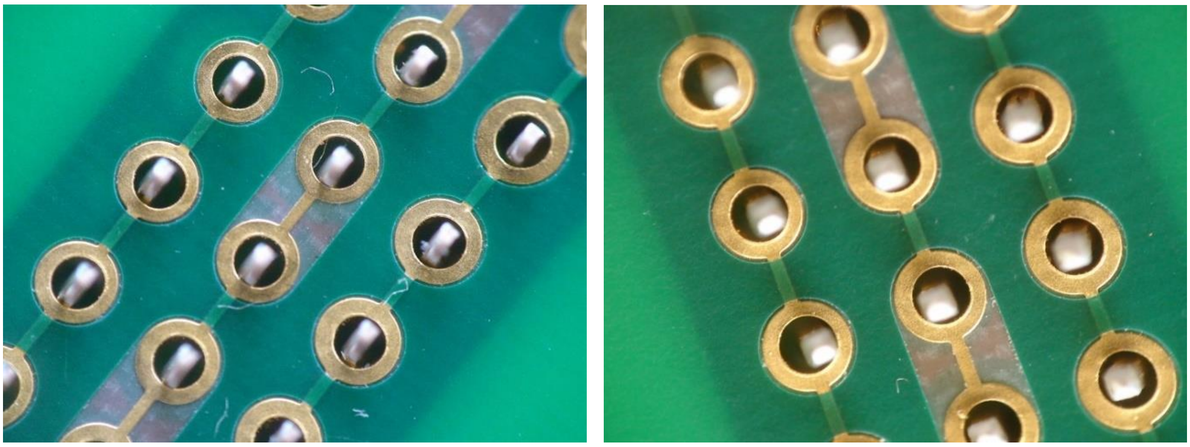


The X-ray images above show the thermal breaks between copper planes and plated through holes. There are also a few solder balls visible in the second image, but we will discuss this later.

The first important things with through hole soldering is the lead to hole ratio; in this case it is based on the pin size. The finished hole size required on the PCB after plating and surface finishes like nickel/gold would be the pin size plus 0.010” 0.25mm to make manual or automated through hole assembly easy to achieve. To be honest we have used this basic design rule for selective, intrusive, wave and manual soldering with great success over many years. Comparing design rules with many component producers there is considerable variation. **Designers measure pins don’t rely on suppliers pdfs.**



Illustrated above are the important points. The pin size, corner to corner if the pin is square or oblong or diameter if round should be the finished hole size criteria/specification for the PCB supplier and included with the PCB design documentation package



First example the pin is oblong, largest measurement for the pin is across the corners. The second example above shows the same connector but with square pins. When manually soldering, the difference in the hole fill can be adjusted by the operator feeding in more or less solder wire. ***With a robot you need to teach it!***

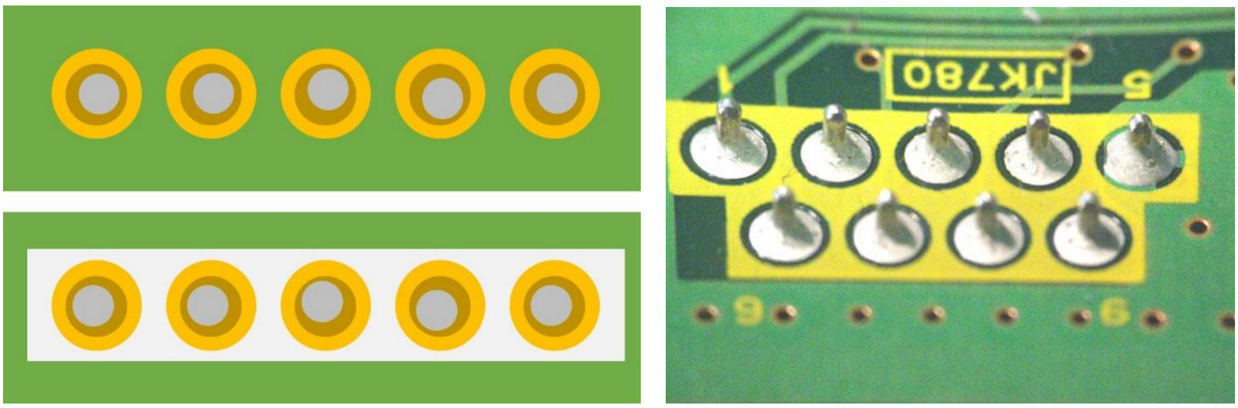
Selective and wave soldering can tolerate some variation with little impact on the capillary fill into the hole. In the case of robotic tip or laser the system computer needs to be programmed with the solder wire size and feed rate to achieve the desired through hole fill. If the connector pin or hole size changes there may be too little or excess solder. Some systems, however, are able to automatically calculate the volume size requirement then adjust the solder feed. This is not on the fly but part of the machine programming.

Legend

A final option in the design could be legend ink often avoided due to density of modern surface mount products and PCB cost. In this case we are not talking about the normal nomenclature for component numbers and identifiers. Yellow or white ink can be used around through hole pads as shown below and provides a better view of possible soldering defects due to the lighter background colour compared with green solder mask.

Obviously, you are not going to add cost to the circuit board specifically for this application but if you are already using legend it provides a useful feature. If you inspect boards after soldering it also provides a clear view of the volume of flux residue left, any solder balls or very fine whisker shorts from drag soldering can be easily spotted. This design trick was used for many years in high volume Japanese products to quickly spot shorts after soldering. Early AOI systems benefitted from the higher contrast of the shorts and the background colour. The author also used this feature to help video solder short formation and the use of hot air knife solder short removal on wave soldering on Hollis, Electrovert and Soltec machines

I seem to remember it was on a Amstrad computer board, a factory in Wales

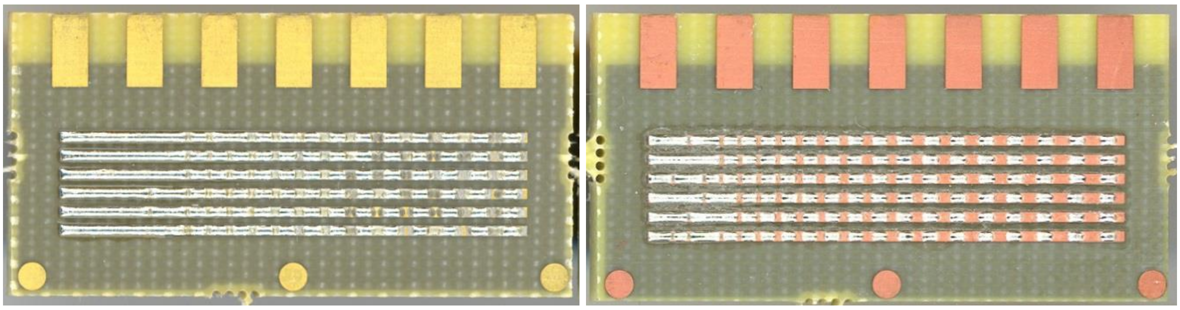


Two examples of use of legend ink to help inspection easily see solder shorts. It would also help to illustrate the amount of flux residues or solder ball spitting

Process and procurement engineers should understand the rate that solderability changes on a PCB surface pad, there are two ways the solder joint formation is impacted by a surface effect normally associated with oxide formation and the effect between the coating and the base material. Normally temperature or stronger fluxes can overcome surface reactions but not between the base materials you are soldering to and the coating. Both of these effects occur when boards go through a moisture pre-bake, first reflow process, adhesive curing, double sided reflow or any other heating step before manual or robotic soldering.

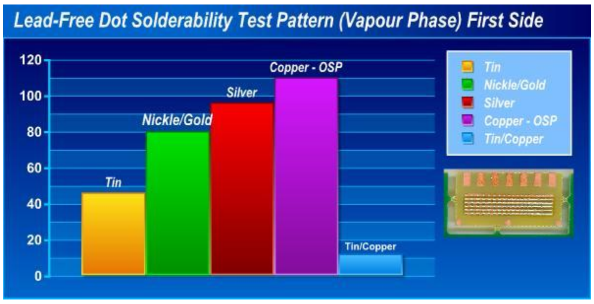
There are also situations where boards are built to a certain level then customised at a later stage with the addition of other through hole components. This is a great opportunity for robotic soldering but you must remember the changes in solderability on the board surface over time. Never blame the machine

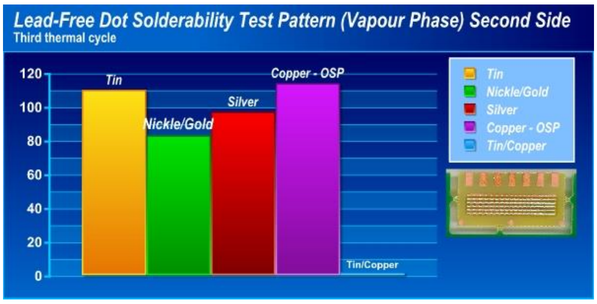
Design and process engineers should consider the use of wetting test coupons into the waste areas of boards so that in production solderability can be assessed and any changes noted. These can also be used by the supplier for verification of paste wetting after fabrication or by the assembly company at goods receipt on a sample board.



The images above show the dot test patterns after reflow on gold (left) and OSP (right). The test pattern does feature the solder paste dot pattern and wetting balance measurement pads

The wetting test pattern is used to assess changes in finish, processing & storage. This test coupon is used in the industry and has very good correlation with the wetting balance, which is the industry standard for solderability assessment. The test pattern has been regularly used by the author on many projects and is a very cost effective and simple way of assessing wetting. The test pattern design files and a guide to its use are freely available from the author or reference the Author's YouTube site



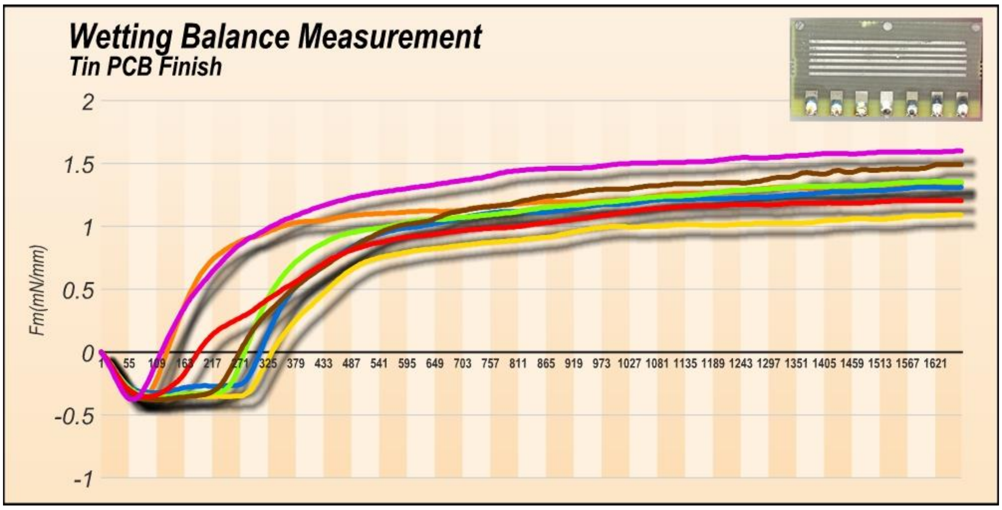


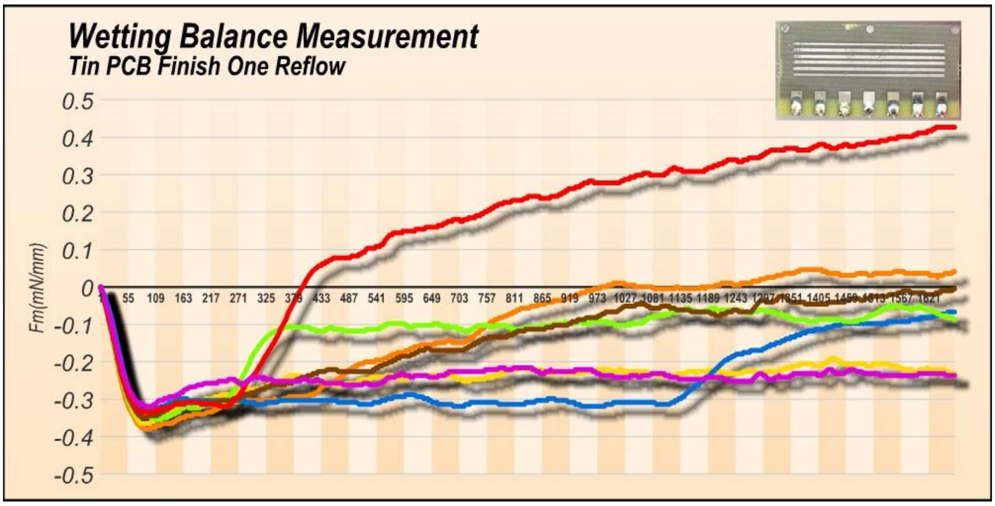
The graphs above show the change in the wetting score after one and two reflow operations. Low scores on the graph are good and based on the solder paste wetting along the tracks. The test method works well for most surface finishes with tin/lead, SAC or low temperature pastes

Here is a general guide to the solderability life of different PCB finishes based on the correct processing, packing, coating thickness and assembly process conditions etc:

Tin/Lead reflowed	**12 months minimum**
Hot air levelled	**12 months minimum**
Lead-Free HASL	**12 months**
Nickel/Gold	**12 months minimum**
Silver	**<12 months**
Immersion tin	**< 6 months**
Copper protective (OSP)	**< 12 months**

Even if the boards are packed in a gas tight condition this may protect the surfaces from oxide formation but not control the natural ageing of the base coating or base metal. Every engineer should understand that there is a limit and you should work within the limits of the coatings. It is poor practice to have to store boards for long periods of time as soldering yields will suffer. It should be remembered that the distance and time boards are in transit from the supplier will have an impact on wetting. Understand the method of transport and the conditions in terms of temperature and environmental contamination the boards may be subjected too. Finally, we all use standard terms of tin, silver, OSP etc. and often quote an IPC standard. We often do not state the supplier's brand. If you have multiple PCB suppliers it is very important that you also know what chemistry supplier you are looking at as, just like the finish categories, there are differences between vendors so know what vendor's product you are using and specifying. You need to know this detail for your onsite PCB supplier audits?

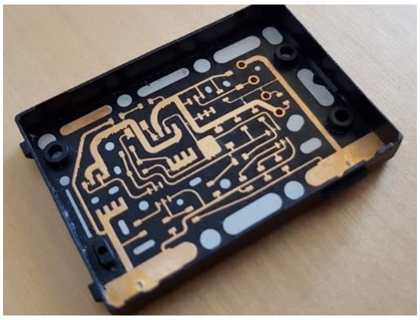
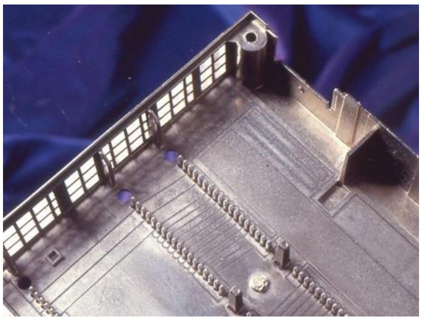




Two graphs above show the change in wetting after one reflow in air and a hold time before second side assembly. Some finishes just can't survive multiple reflows and hold before final assembly. It is very important to compare your actual factory build times with your chosen surface finish. If you don't consider this issue it will impact on your robotic soldering process

3D Plastic Injection Boards

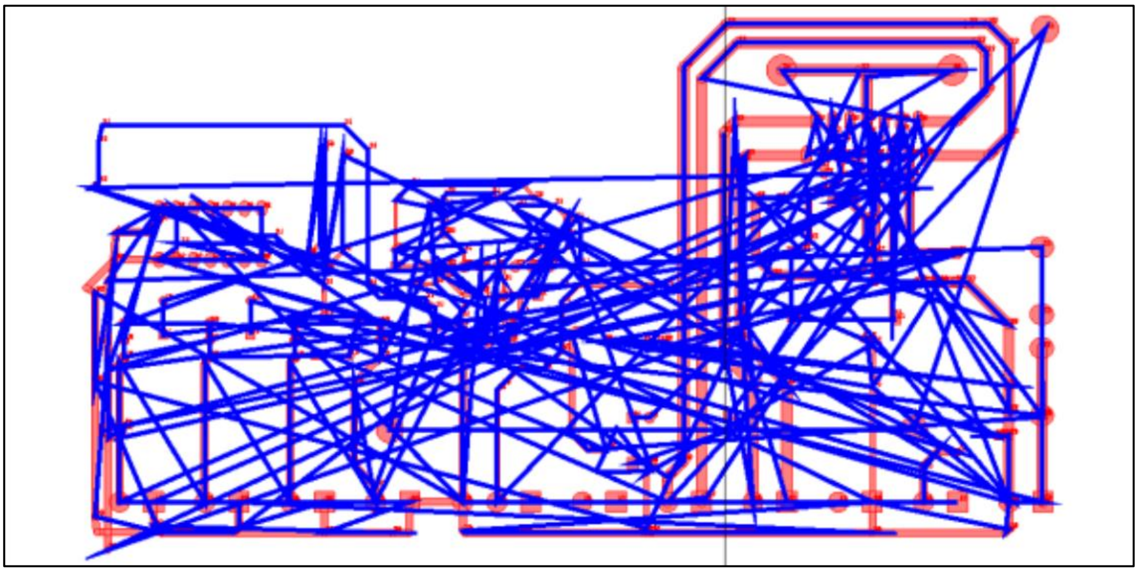
It is worth mentioning that substrates do not have to be flat. Potentially robotic soldering, laser or contact can be beneficial when used with 3D substrates. The author first had a fascination with plastic boards back 1984 when engineers were all talking about cable TV plus all of the other street infrastructure needed for future communication. I remember that my first experiments in plating on to plastic was not very successful. The electroless copper to plastic adhesion was poor due to poor surface preparation. The key was opening up the surface of the plastic surface to allow the copper to key. Otherwise all of the copper and any further surface plating just fell off, much to the annoyance of our plating chemist.

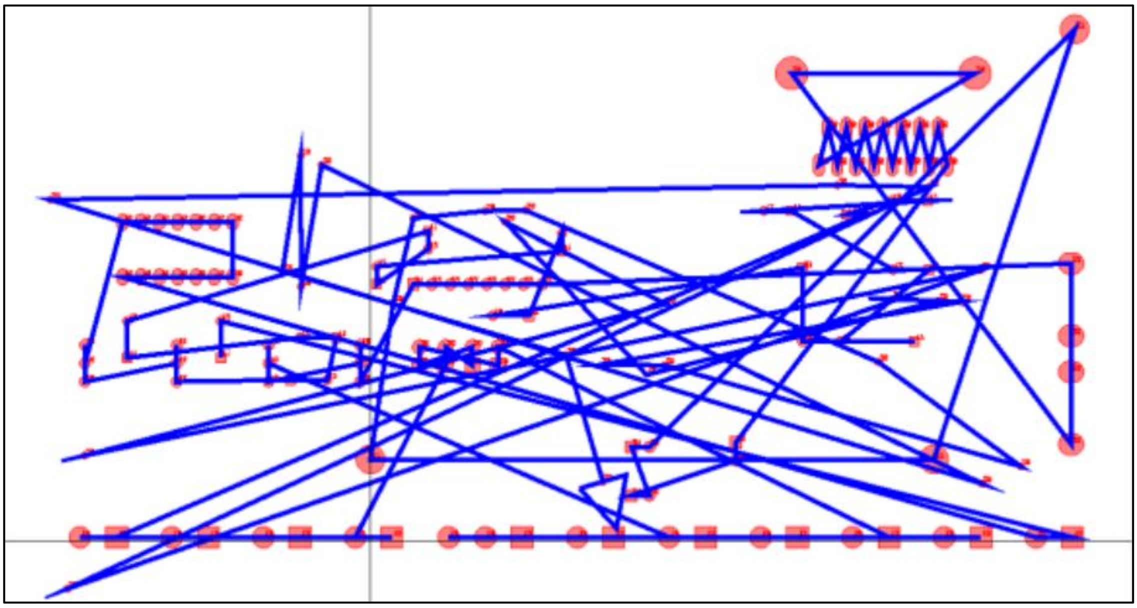

*The examples above are just a few images of injection moulded boards produced from Polyamides PA and Polybutylene PBT. If as a design or process engineer, you also have an interest in the technology we believe there is only one book on the subject. It is a great insight into the technology and written by **Jorg Franke - Three Dimensional Moulded Interconnect Devices (3D-MID)***

With correct tooling pallets flexible and semi flexible substrates can be easily soldered using well designed pallets. The pallets can be used as a location points/housing for the components, the flexibles mounted to the pins and fixed to the pallet.

Just like selective soldering systems most robotic equipment has software that can take the data from design files and plot the pad position for the soldering process. Other systems can work from a bare board capturing the pad locations and through hole positions and from that define an initial path for the robotic head. It is very important to understand from different suppliers the method of capturing this information. Back in the day the only method was teaching the system point by point. Then adding specific soldering parameters, tip movement, solder feed etc manually.



When the basic information on pad locations is defined, as illustrated above, the software needs to optimise movement for production. Again engineers familiar with selective soldering systems will be well aware with these steps. The new optimised path is shown below. As previously mentioned there may be times when the next position to be soldered may not be optimal for speed but necessary to aid heating of high mass connections.

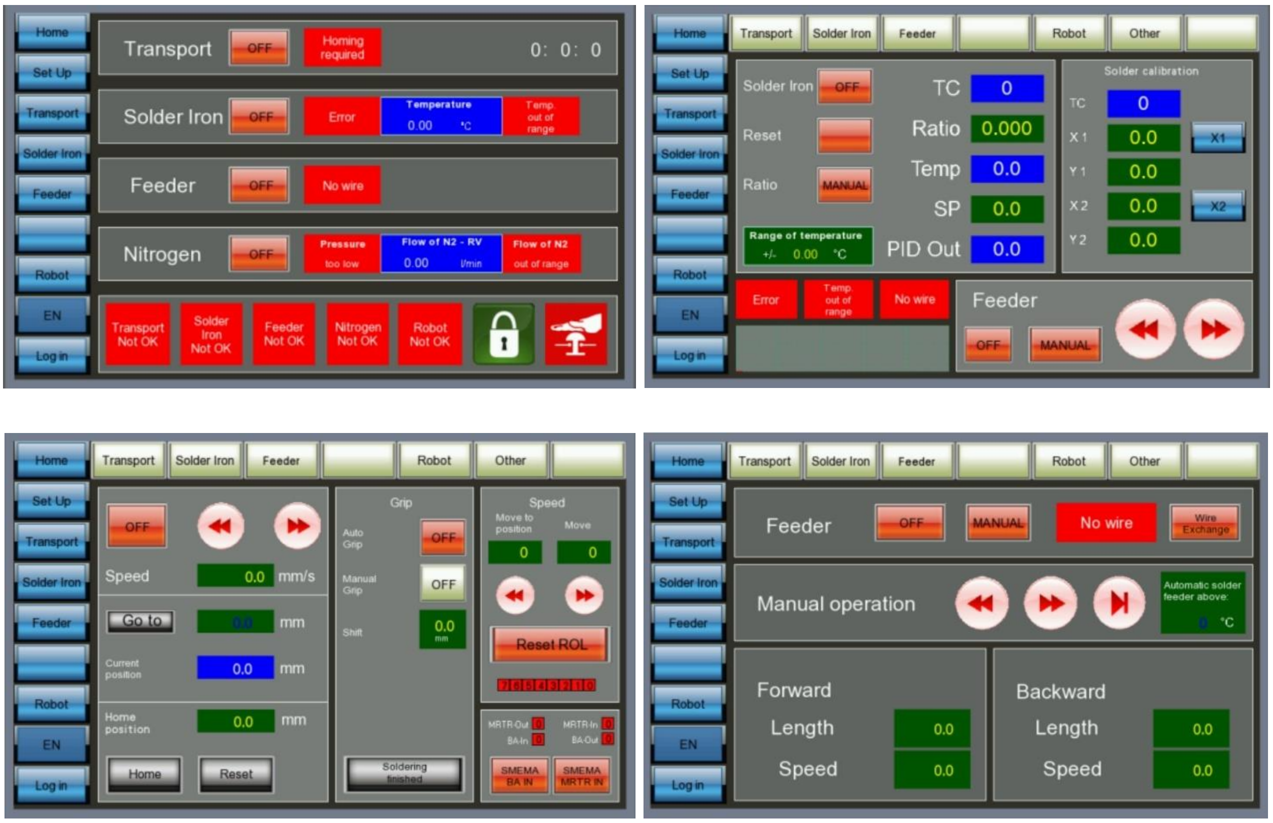


Images above are credited to a collaborative NCRD Poland project and paper by Andrzej Milecki. www.renexrobotics.pl/en/reeco-robots/

Obviously there are many other factors which will improve the performance during soldering but much of this will be specific to the board design and machine. A few examples should be discussed with your system and material suppliers and also featured in evaluation trials.

Access speed
Optimisation of acceleration and deceleration
Dwell times
Solder feed rate, amount and position on the joint or iron tip
Nitrogen and flow rate

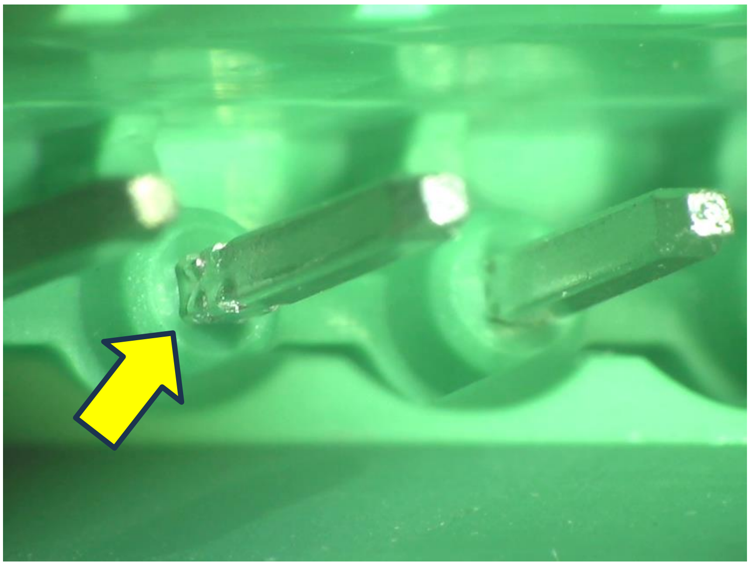
Machine software for controlling the process will be different for each supplier and it can take some time to become familiar with the function screens. We have included selected screens from one system for reference only, but this just illustrates the parameters and movement control available to optimise a process. Equipment supplier will provide detailed training on how to use and optimise the process for your design



The operating screen images and the PCB programming path illustrations are from a collaborative NCRD Poland project and paper by Andrzej Milecki. www.renexrobotics.pl/en/reeco-robots/

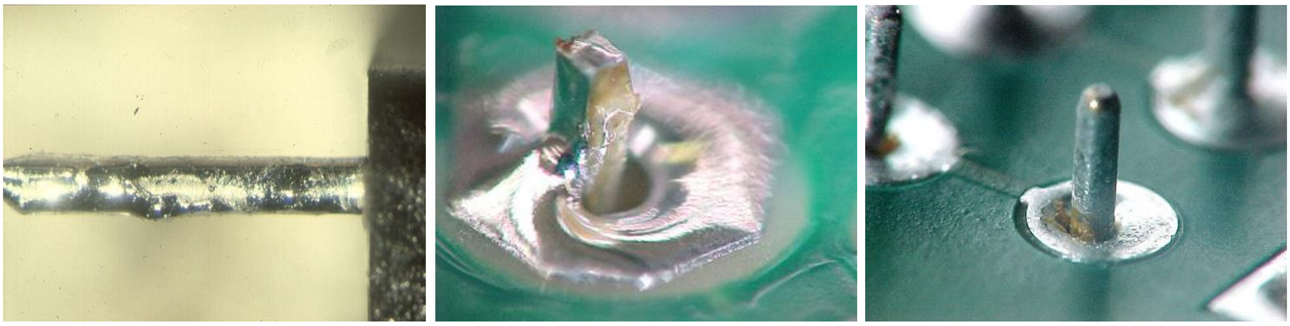
Component Compatibility

With any manufacturing process components must be compatible with the total assembly process being used or where crucial to the design; care must be taken to test compatibility before use. In simple terms the components must be solderable with the alloy & flux to be used. Any internal active parts must not be degraded by the soldering temperature used on the pins. In addition the body of the component must not be deformed or damaged by the heat being used. In the case of connectors the pins must not float or be displaced in the body of connectors as they will not mate correctly.



Connector pins above have moved in the plastic body during soldering. It is fair to say that the soldering temperature used may have exceeded the temperature of the plastic and the component suppliers' recommendations. Unfortunately many purchasing and design engineers do not check supplier details.

It is fairly uncommon to have issues with solderability on through hole parts these days. Typically component terminations are copper or steel, in the case of steel there is often a copper barrier layer then tin coating for good wetting. Other surface finish combinations can be nickel with a gold flash. Solderability testing of components and PCBs is well defined in IPC standards like IPC-J-STD-002E. Simple testing with a "Dip & Look" test is not uncommon and often quoted by component suppliers. This is not an ideal test as it does not pick up subtle wetting problems. What this means is a number of parts are taken, the leads first dipped in flux then dipped in a solder bath to look for complete wetting. All the details of the solder alloy, flux type, temperature and time would be defined.



Poor termination solderability during test and when subjected to a soldering process

A more sophisticated test for wetting is the Wetting Balance that conducts a similar test, but the actual measurement of the wetting time, speed and force are recorded. This allows for laboratory level analysis on the impact of different component termination plated finishes, storage and impact of flux and alloys. It is the only way of correctly measuring wetting and quantifying the results.

If the board assembly is to be subjected to other processes like cleaning or conformal coating these must not affect the mechanical or electrical functionality of the parts. Care should be taken when you are selecting parts for new products. The same level of care must be taken when defining second or third source supplier of parts. This point is often forgotten when trying to find alternatives suppliers or when trying to cost reduce the Build of Materials (BOM) of a product. It is not uncommon for a mistake to be made by subcontractors if customers do not define the requirements of their parts.

Component Assembly & Retention

A very important step in the use of robotic soldering is the need to position bare boards and components on an assembly with no movement. This part of the process must also be considered during the design stage like the process tooling and not left to shop floor engineers. All of this needs to be part of the Design for Manufacture Review (DFMR) process. This is often also forgotten by customers when visiting robotic soldering equipment suppliers for process trials. Many times sticky (Kapton) tape is used to hold parts in place or selected pins need to be tacked into the board before using the robot soldering system. Make sure you cover this point with your friendly supplier before arranging your trials!!

Preformed Components

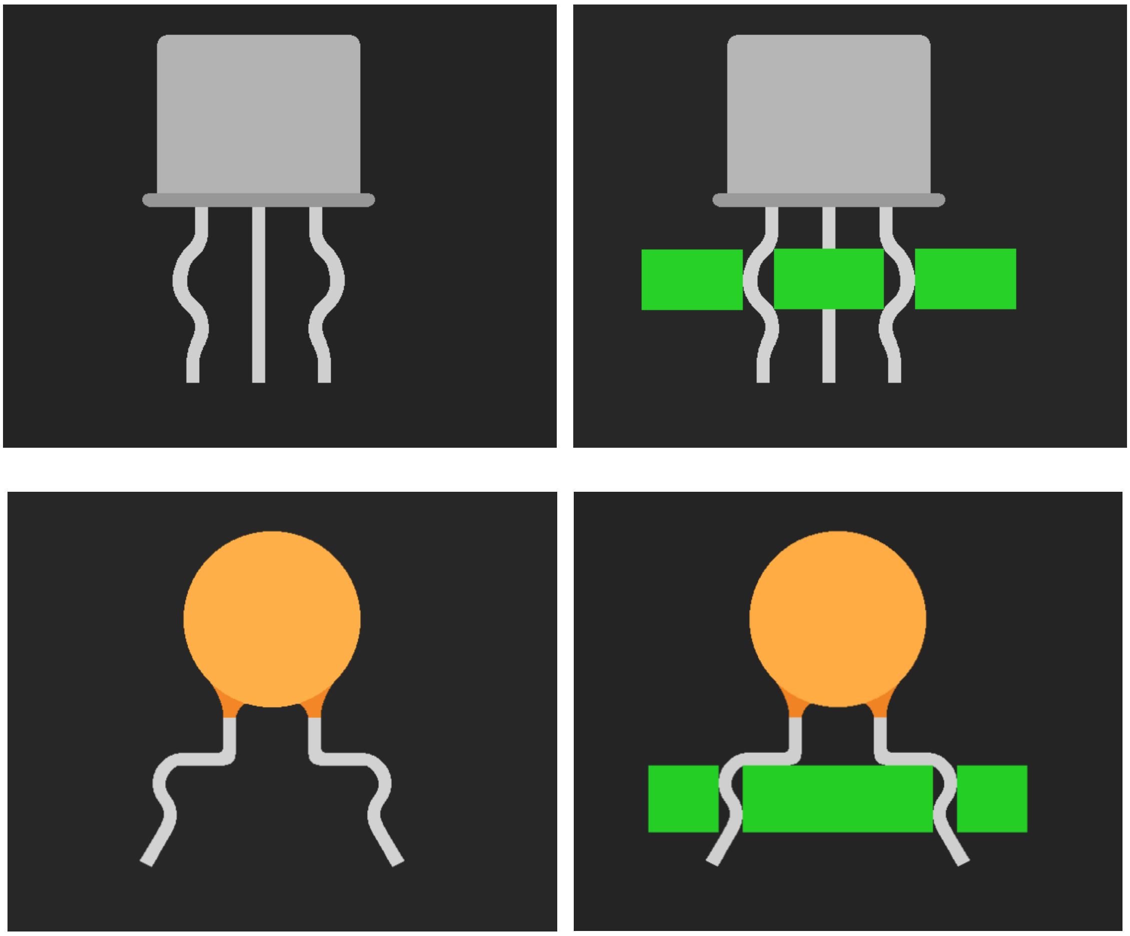
When we look at a printed circuit board BOM (Build of Materials) today they are mostly surface mount technology with some through hole connectors and selected electromechanical parts. This is one of the reasons why the use of robotic soldering can be financially beneficial, less through hole joints. Using a robot is determined mainly by the number of through hole joints to be formed and the positions on the board.

With connectors it is fairly easy for a designer, if correctly informed about the assembly restrictions, to select the correct parts. Connectors that can be manually or automatically inserted and locked in place with hold down features are ideal. Connectors are often available with different locking pins. These hold the connector in the board but make sure the connector pins will stay accurately in position in the through holes. It is possible that using hold down features can reduce the cost of shop floor tooling. It is not uncommon for SMT placement machines to be capable of inserting through hole connectors. However, the supplier recommended hole size for hold down features for manual or SMT placement vary considerably. Please check the parts and define the correct hole sizes.

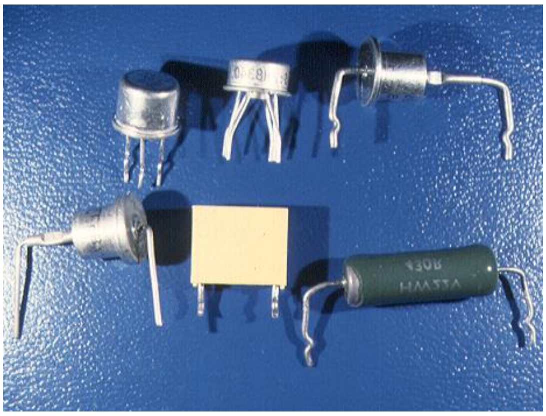
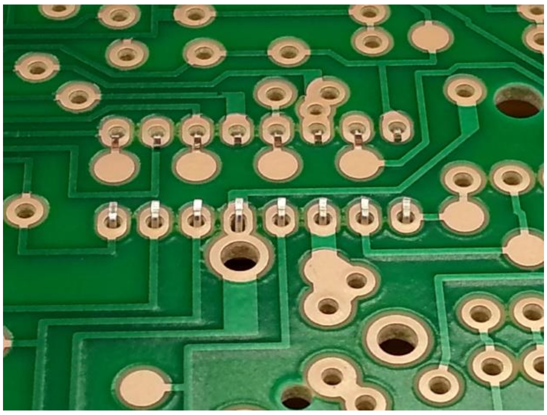
Consistent positioning of the pins to be soldered in the plated through hole is a must for automated soldering. With manual soldering operators can overcome different pin position, pin length and any lead clinch angles. If the pins move or are in a different location on each board this will lead to problems with your robots soldering iron tip placement. Your new best friend just wants to place the soldering iron tip in the same place time after time after time.

It is possible for electromechanical, large capacitors and other parts to have their leads preformed. This preforming of the leads allows the parts to be clipped into the board with limited movement.

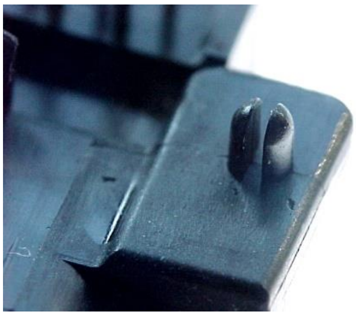
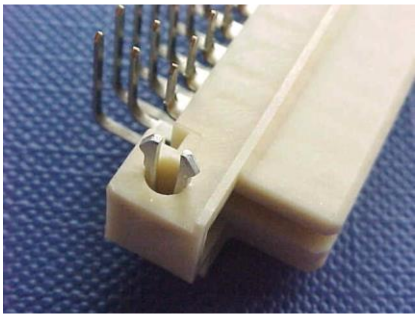
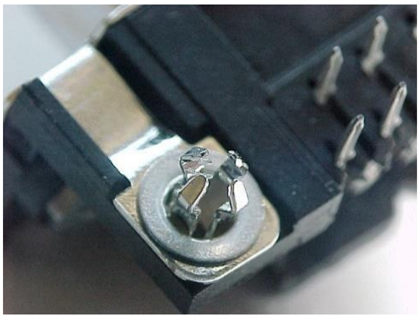
Preforming or joggling was a very popular process on through hole parts for high volume flow line assembly in the 80/90s. It also held components securely in place during handling of boards in loading racks and moving partly assembled board around a factory on bumpy floors. Preforming equipment was expensive but very popular for medium volume and military electronic production. Components can be located into the board very securely.



Illustrations above show preforming of two and three leaded components and how they are mounted into a printed board repeatedly

Examples of preformed leads before insertion & example of automatic inserted components with cut and clinch component leads

Examples above show some different forms of connector hold down/locking pins

Automated assembly of radial, axial and dual inline IC would still be possible as the leads would be accurately defined in terms of lead length and position relative to the through hole pad. However, the number of leaded parts used in designs has significantly reduced and hence the move away from wave soldering to selective soldering. Excessive numbers of through hole parts would not be suitable for robotic soldering based on the soldering time and cost.

I have only seen connectors inserted into a board directly with no fixing method once. In this case the connectors with retention pins were on long lead time and adhesive was used. Dots of adhesive were used, and UV cured to hold the parts in place for the board to be inverted for robotic soldering. Good engineers always find a way. Obviously, an assembly turnover pallet could have been used, but it still has to be designed and procured hence the economic case needs to be considered. Alternatively a 3D printer could have produced a support pallet overnight!


1 in 8 men will get prostate cancer
Check your risk of prostate cancer in 30 seconds.
prostatecanceruk.org/risk8
PROSTATE CANCER UK
Speak to our Specialist Nurses
0800 074 8383*
prostatecanceruk.org
Prostate Cancer UK is a registered charity in England and Wales (1005541) and in Scotland (SC039332).
Registered company number 02653887.
Statistics are for men in the UK
Call our Specialist Nurses from Mon to Fri 9am - 5pm, Wed 10am - 5pm
* Calls are recorded for training purposes only.
Confidentiality is maintained between callers and Prostate Cancer UK.
MEN4/FEB23
6937

PCB Assembly Tooling

One of the key points for successful robotic soldering is accuracy and repeatability of component pad/lead position. Without this the best robots in the world will be struggling, engineers and operators will be making adjustments 24/7. I will not be a great advert for your new equipment installation. Your production staff will rightly say ***"We can do a better job"*** just like the early days of surface mount component placement. A good group of operators with tweezers could beat a machine, sometimes more accurate too. ***Sorry I am showing my age now.***

There are many different ways of holding/positioning boards for robotic soldering which may include:

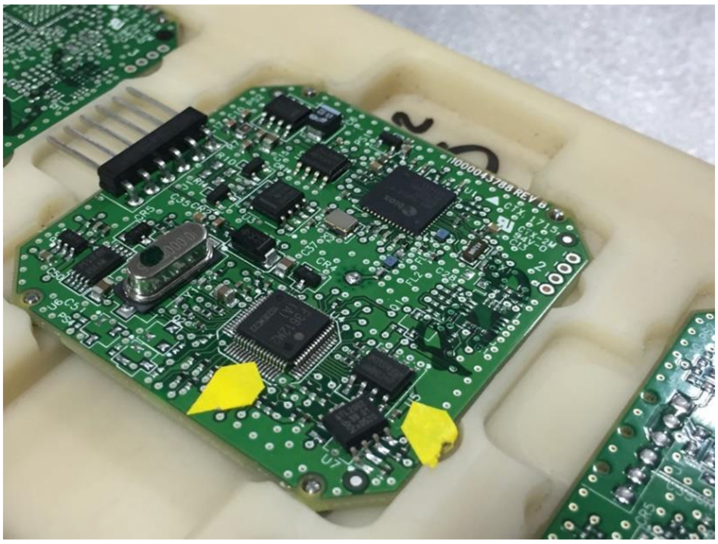
PCB edge alignment
Mechanical tooling pins
Optical alignment with fiducial marks

Each method can be used but will not give the same accuracy and repeatability based on the size of the board or when multi panels are used in production. Smaller panel size is best to improve the accuracy and repeatability, also true on flexibles circuits and some thin and specialist laminates. Another consideration is how are the components going to be retained in the board for soldering. With single or double sided surface mount any through hole parts will need to be inserted then held while the board is transported to the robot and inverted for soldering. There are possible exceptions, when the robotic iron or laser are mounted under the board on an inline system. However, in this case the correct method of component retention and hole size must still be considered.

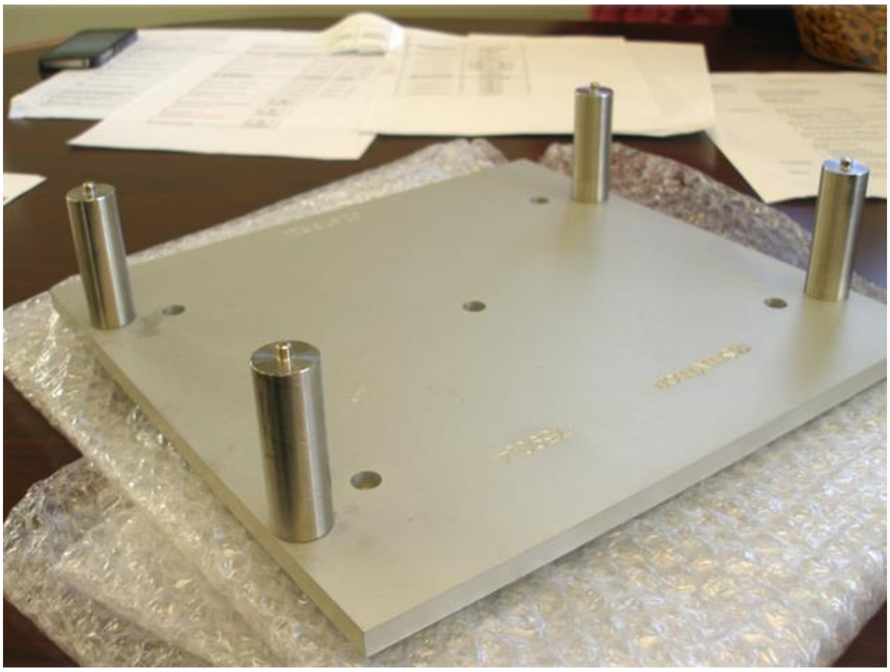
PCB edge alignment

Each assembled board for soldering is placed in a pallet just like a picture frame with the board held flat on all sides. The board is placed into the frame then removed after soldering. Clips are required to hold the board down flat to prevent vertical movement. All clips should be designed below the level of the fixture to avoid any possibility of contacting the robotic iron tip during movement. It is also good practice to have two or four finger slots/holes in the frame which allow easy insertion or removal of the board.

Normally its the thumb and forefingers used for placing and removing small boards. The fixturing technique is not the most accurate as its completely depends on the repeatability of the bare board routing process from the supplier



Customer example of tooling fixture holding the edge of the boards with recesses for easy handling/loading of boards Image kindly provided by Apollo Seiko



Example of customer tooling plate Image kindly provided by Apollo Seiko

Mechanical tooling holes

Boards can be placed on steel posts/tooling pins on the machine's base plate, then located in mechanically drilled tooling holes on the PCB. It is important to remember that these PCB tooling holes are not plated with copper. It is also important that the through hole components must already be retained in position with no movement.

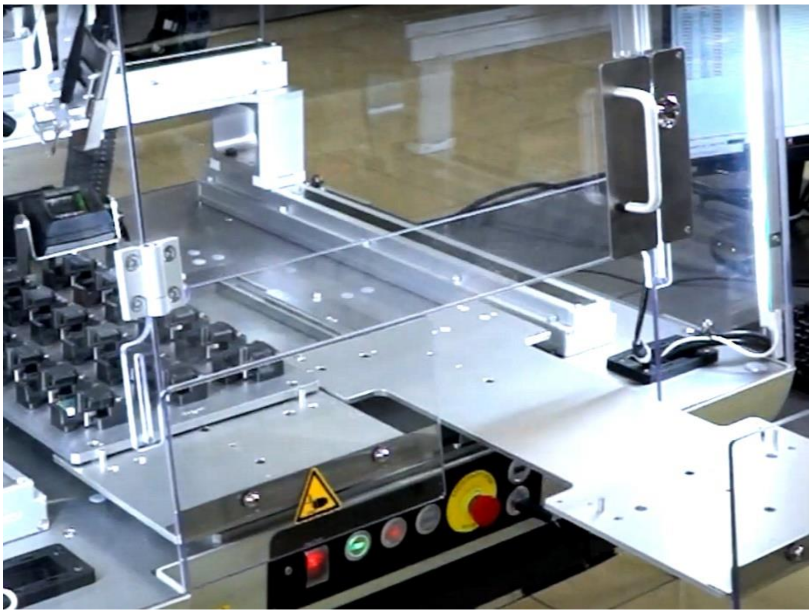
Alternatively the pallet can be part of the machine fixturing base plate and the board is loaded on to the pallet with the tooling pins in place. The connectors will have already been mounted in recessed openings on the pallet. This can allow for double sided surface mount designs where openings in the pallet allow parts to be located with clearance avoiding any contact with the fixture. This is the same procedure used for many years when wave soldering double sided surface mount products with through hole. The selective pallet masks the surface mount parts from the solder wave preventing damage to the parts or reflow of the existing joints, Sometimes!!!

Optical alignment with fiducials

Robotic soldering systems may have camera recognition to check the position of the board using etched copper marks on the bare board. It is exactly the same as used on surface mount designs for board alignment or localised marks for finer pitch part placement. Initial pin alignment may still be used but final correction is based on the fiducial marks. In the past we have also used plated through holes or their pads for optical alignment correction on larger connectors.

Most machines have the option of a matrix table base plate where tooling posts can be inserted on a grid and locked in different positions. It is important from the very start to define a datum height so that boards are always fixed at the same height along with any other machine feature like tip cleaners, tip height gauges, iron tip changeover stations, preheater plates or bottom extraction plates etc. This ensures that the free space for movement of the soldering iron head and tip are not compromised.

Pallets can be used for pre-mounting connectors then a cover or top hat fitted to the pallet to hold the parts in position flat to the board. The pallet may also need to be recessed for other components previously soldered in place prior to connector assembly on the topside of the board. The pallet is then turned over and positioned on the machine's working area.



Example of a double tray system for loading and unloading boards www.kurtzersa.com

Some machines allow two sliding mounting trays to be mounted in the working area at one time. This allows the machine to be soldering while a new assembly is loaded adjacent to the other but without any safety concerns. When the first board is complete it can be safely removed by the operator after the soldering head moves to the next board or the hold location.

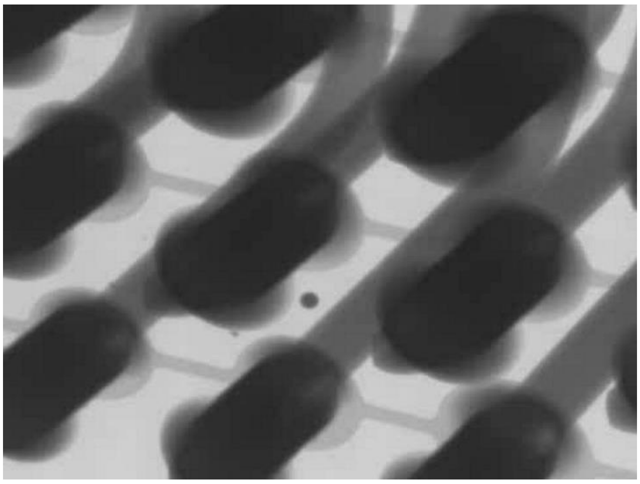
In this example connectors are pre-positioned into pockets in the pallet. The assembly is placed on to tooling pins then lowered on the pallet and locked in position. At this point the soldering process may proceed. However, if the system has optical correction, the system can confirm accurate alignment before starting the soldering cycle.

Depending on the board size and weight additional support may be necessary. Tooling pins may be required under the board to prevent the board bowing. If this occurs, then the soldering iron tip may only contact the pin not pin/pad resulting in poor heat transfer.

Assembly and soldering fixtures should hold and support the board with no movement or bowing. If or when contact is made by the soldering iron tip there should not be any board bounce. This is often seen in older or lower specification machines.

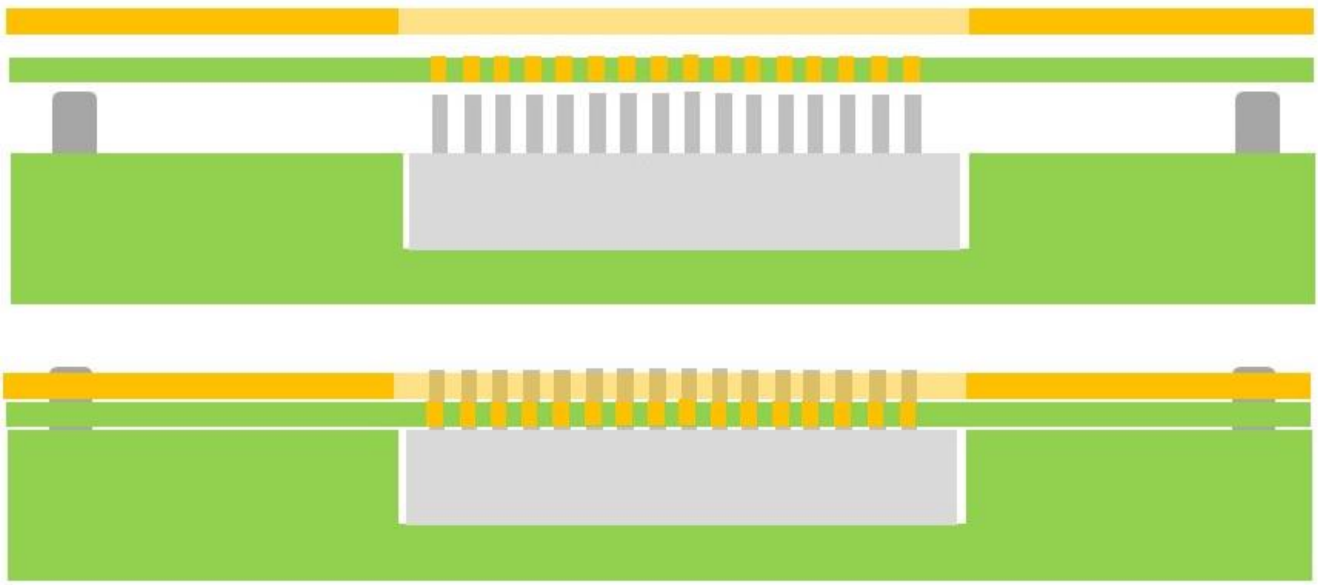
PCB Masks/Overlays

In some situations where a company are not able to prevent solder balling it may be necessary to have a protective mask/overlay on the surface of a board or pallet. We have experienced two cases where customers have insisted on this due to previous bad experiences with solder ball spitting. The automotive industry is particularly concerned about this issue and potential product liability and medical industries are in the same situation. On the medical side when a manufacturing process is defined for a product no changes are possible without a full process approval. This is often true even when everyone agrees the root cause of the problem can be easily fixed.

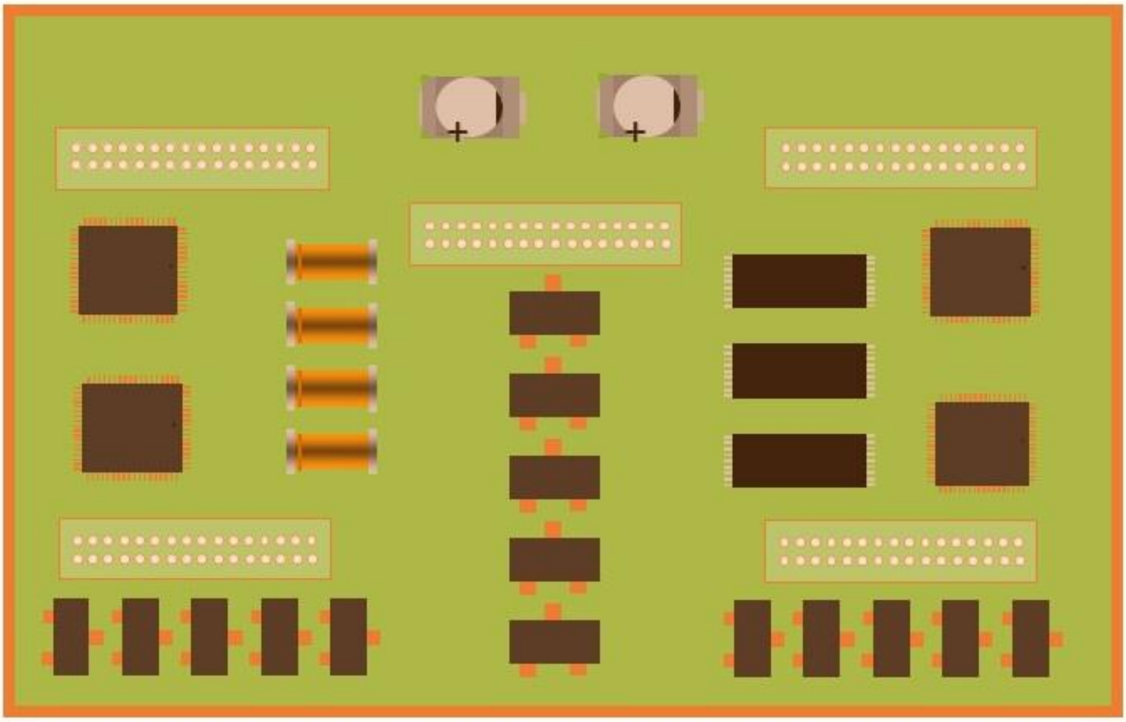
Examples of solder balls on the surface adjacent to the point of soldering. X-ray image shows a solder ball between termination on the surface of the board. Due to the size of the ball it is very unlikely to ever cause an issue.

Basically with the mask or overlay only the through hole joints are exposed for soldering and all other areas of the board are covered as illustrated in the diagrams. This could put more constraints on the machine programming and the cycle times. The mask material must be thin, light, ideally transparent, long lasting, static free and cheap. My alternative recommendation would be to lose a couple of seconds slowing the process and reducing the steps that increase the possibility of solder balls rather than the extra masking complication. We have discussed solder ball formation and ways of reducing balling in robotic soldering many times in our international workshops and online webinars. Always happy to pass on our experiences to engineers/customers to eliminate these process issues.



Masks or an Overlay can be positioned using the same tooling pins leaving only the areas to be soldered exposed. ***However, this does not prevent ball creation***

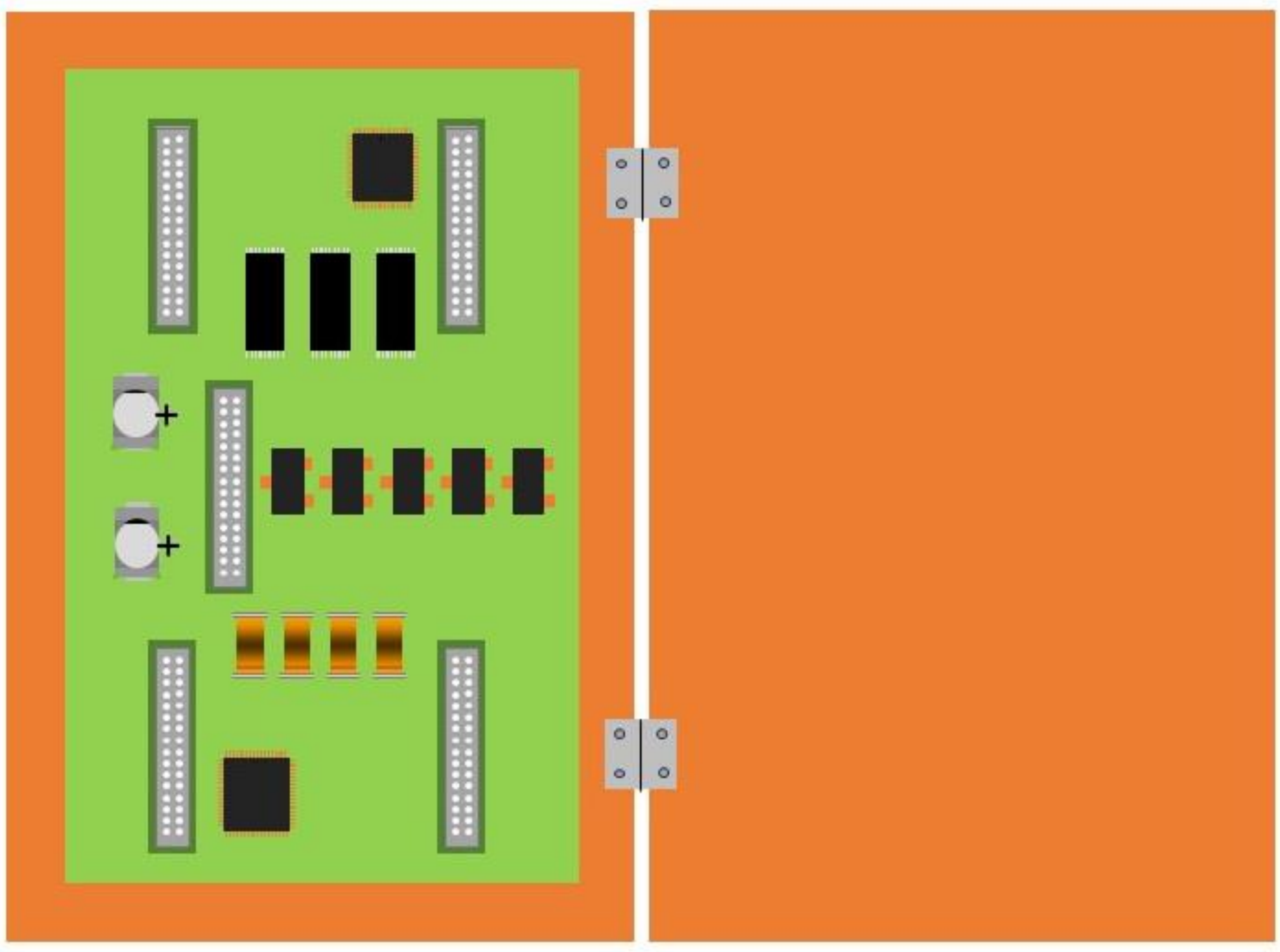
Ideally you should not need an overlay or top plate for soldering. It is much better to invest time reviewing the process with your suppliers. They spend hours and hours looking at robotic soldering processes and applications and will always offer advice on eliminating solder balling. Covers just get in the way, get dirty and not a great indicator for your process engineering.



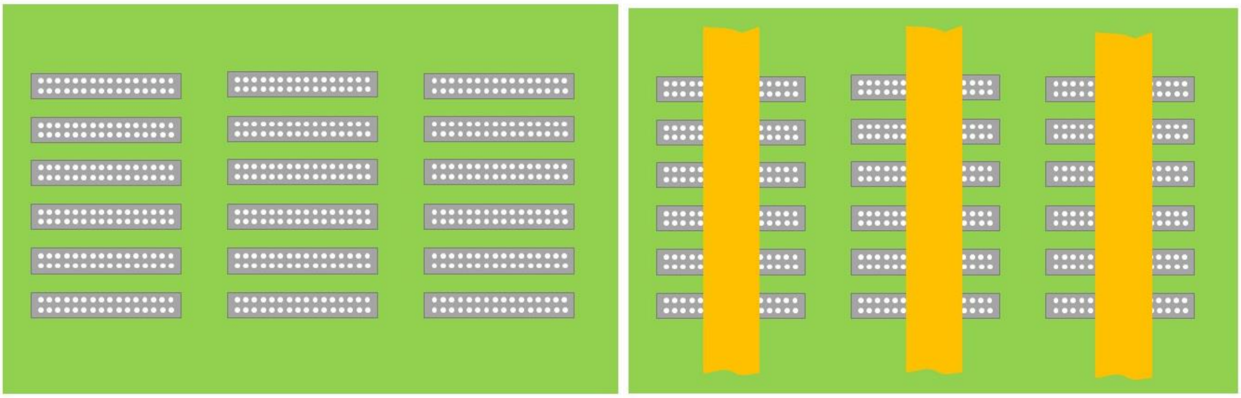
Example of overlay used to prevent solder balls and splashes on SMT components

Jigs and Pallets can be made from aluminium, high temperature fibre glass similar to wave solder pallets. Trade names for these types of products are Durostone or Ricocel. As the process temperature for these pallets will never be high, unless you are preheating the board, the focus must be on the accuracy and stability of the fixtures plus support for the board.

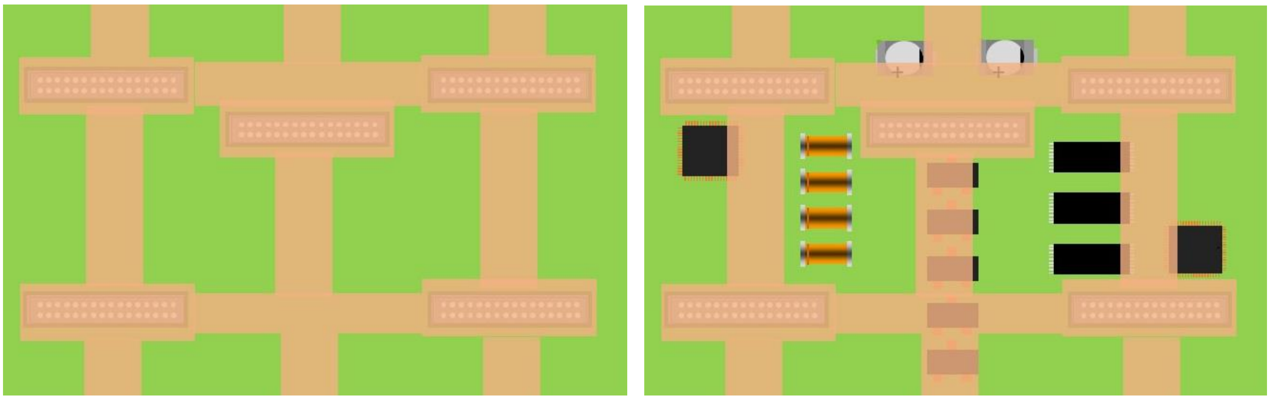
Assembly pallets can be used to locate connectors and other parts before locating and supporting the board. Care must be taken in pallet design and selection to avoid heat sinking of metallic parts. This could be a drain of heat away from the point of soldering. Remember that the location pin tips need to help guide the board before the connector pin shank comes into contact with the holes. If you do have problems with solder balls or your customer requires a top plate this can be added



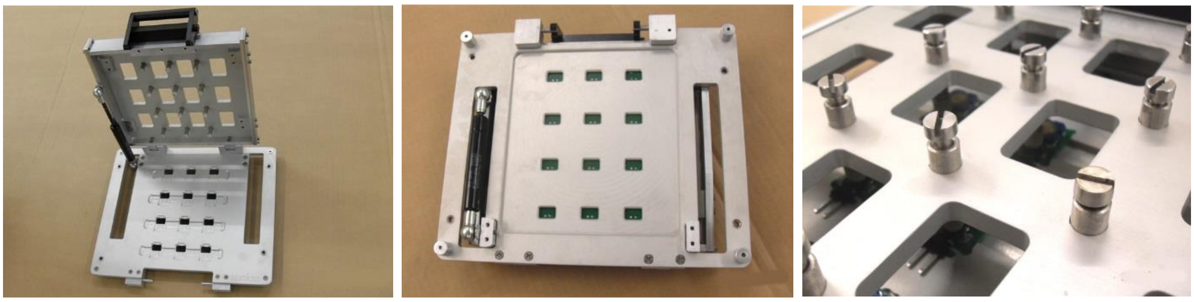
A simple hinged top plate can hold through hole connectors on single or double sided product while you flip the pallet over for soldering



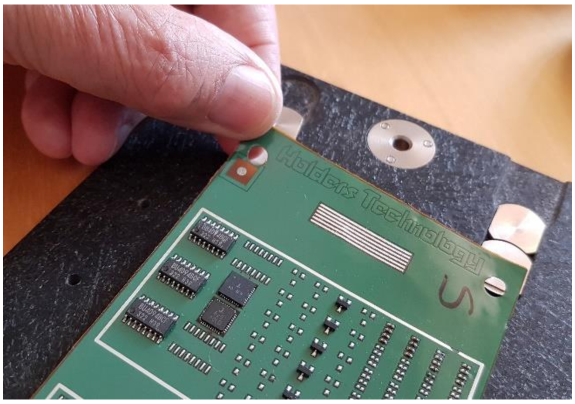
*Please try to avoid visiting your friendly Robotic Soldering Machine supplier with Kapton tape holding components on test boards. This will **NOT** give your equipment supplier the best chance of providing you great results*



An alternative to a solid top hat or cover plate is a spider. It is a simple and cost-effective way of supporting light weight connectors in your mixed technology board for through hole robotic soldering. It is the same concept used back in the day, or still used by some with wave soldering to hold components flush to the board.



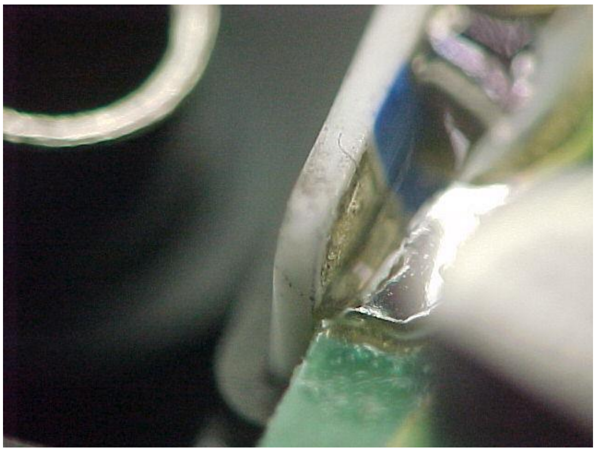
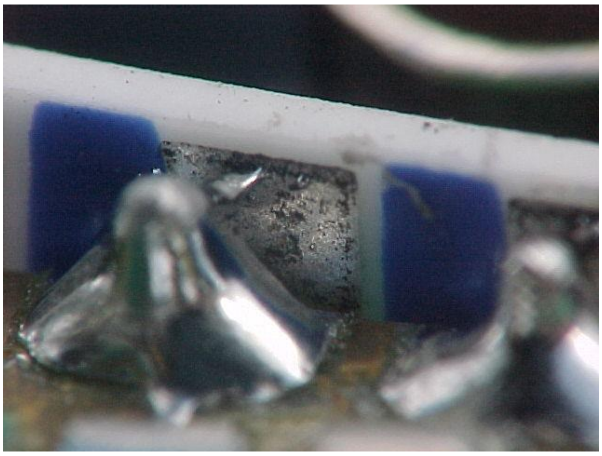
Metal fixture for holding components and boards for soldering. Images provided by Apollo Seiko



Example of soldering fixture uses for printing, placement and reflow of flexible circuits. The same flush pallet design could be used where robotic soldering was required for pre-mounted connectors

Solder Alloy

Our first experiences with a robotic soldering process used tin/lead cored wire and it was an investigation into soldering defects in Germany for a customer. A selection of joints were not correctly forming during soldering. However, the issue was a solderability problem and nothing to do with the automated soldering process. The surface contacts on the daughter board did not wet due to poor solderability of the tin coated pads. Either the solderability of the tin was poor to start with or previous reflow of the matting hybrid components or delay in second stage soldering had affected good wetting. If solder paste had been printed to the connecting pads when the board was produced the ideal surface for soldering, solder, would have been available.

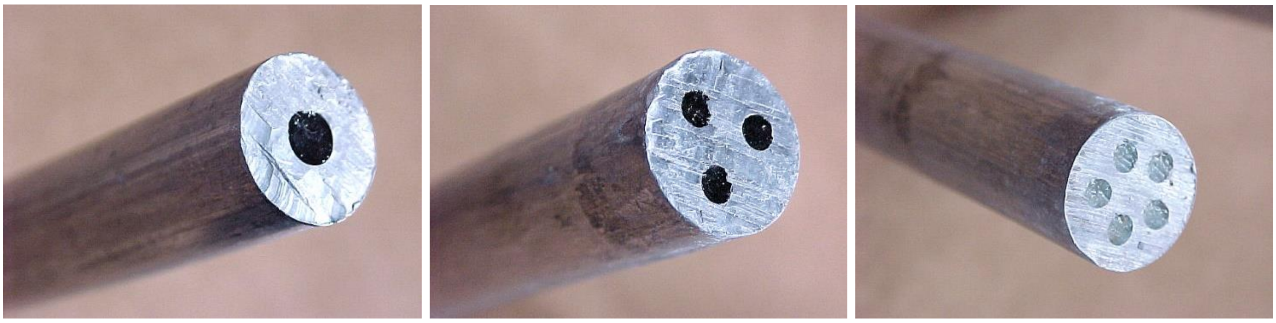
Example images above show open connections due to poor wetting on the pads of the ceramic hybrid daughter board

In the majority of cases all robotic soldering, either contact or laser is conducted with cored solder wire; however, it is possible to use solid wire if an alternative fluxing step can be added into the process. It is feasible for solid LTS be used for other applications with a little bit of engineering. This could make all soldering materials, paste and wire the same alloy but it is unlikely.

Cored Solder Wire

Cored solder wire is used for manual soldering, second stage assembly, rework and of course in automated robotic soldering. Core refers to the flux which is provided as one or multiple cores positioned inside the wire during manufacture. Solder alloy is produced as ingots which are extruded and then passed through a series of dies to reduce the wire diameter to the required size. The general size can range from 0.4 – 1mm. During the process the flux is introduced into the wire, the type of flux will depend on the solder alloy and the soldering application, the quantity will generally depend on the diameter of the wire and the temperatures used during soldering.

Three examples of lead-free cored wire are shown below, one with a single core and the other with multiple flux cores from a well know supplier. The five cores, or Famous Five, has been a trademark for one soldering brand since I first started engineering back in the day. I can also remember being shown the Turkish home of a proud local Multicore distributor, when I was running seminars in Turkey



Flux cored wire bars partly through their extrusion process

Cored solder wires are available in a number of solder alloys and the alloy can have an effect on the minimum wire diameter produced. Some alloys are more brittle and at some point cannot be reduced in size without breaking. Our experience suggested that some brittle wire can tend to age after manufacture and then break more easily during handling of the reals. We have also heard stories of low temperature bar for wave or selective soldering being dropped and breaking. However, this is not as much of a significant problem as wire breaking in manufacture or use as we can be talking about miles of wire.

Most of the well known lead-free solders, as well as those which have been especially developed fall into the mid-temperature range. Most of these compositions are combinations of the base metal tin with copper, silver, bismuth and antimony. Examples of the most popular of these alloys are shown below.

Examples of lead-free solder alloys melt between 200-230 degC

Generally there is a choice of alloy groups from which to select a lead-free material. As with tin-lead, each lead-free alloy has particular properties that may need to be examined for your application with robotic soldering.

Tin/Copper/Nickel Sn-0.7Cu 227^{o}C
Lead/Tin/Silver Pb93-Sn5-Ag2 296-300^{o}C
Tin/Silver Sn-3.5Ag 221^{o}C
Tin/Silver/Copper Sn-3.8Ag-0.7Cu 217-221^{o}C

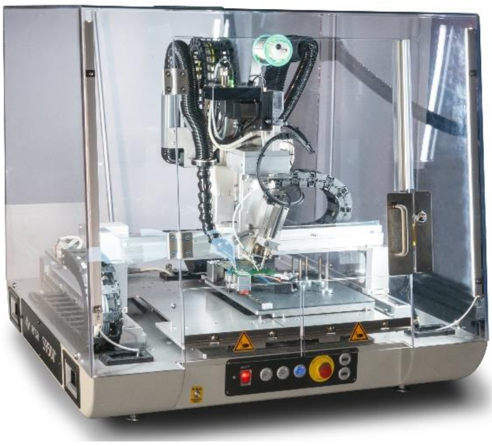
In addition there are still also tin/lead cored wires still used in selected industries and include:

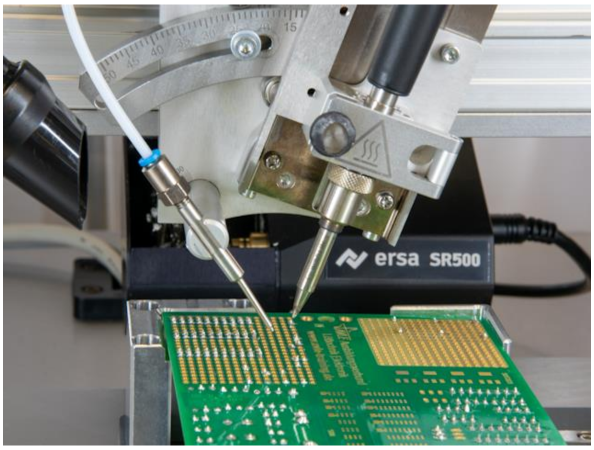
Tin/Lead 60/40 & 63/37 183^{o}C
Lead/Tin Pb980-10Sn 268-302^{o}C

Although melting temperatures are slightly higher than that of the tin/lead alloy they have been used as general purpose lead-free solder after the **Unnecessary European Ban** was introduces in 2006. At that time all soldering engineers needed to learn how to use different alloys in manufacture and the consequences. Regardless of the many disadvantages. Back in the day SMART Group in the UK was one of the first with SMTA in the US to embrace surface mount technology. During the years 2000 – 2006 before the ban, SMART Group members organised many conferences, workshops and exhibition features on how to handle and implement lead-free technology. Inevitably we did some more entertaining and silly stuff too

*Peter Swanson and author singing the Legendary Rock Anthem **"New Alloy"** in 2004.*
The recording is still available to download FREE ???

Finding the right Lead-Free alloy at NPL launch of the **DTI Lead Free Cookbook** interactive training CD ROM helping the understanding all the practical issues in the real world 2004





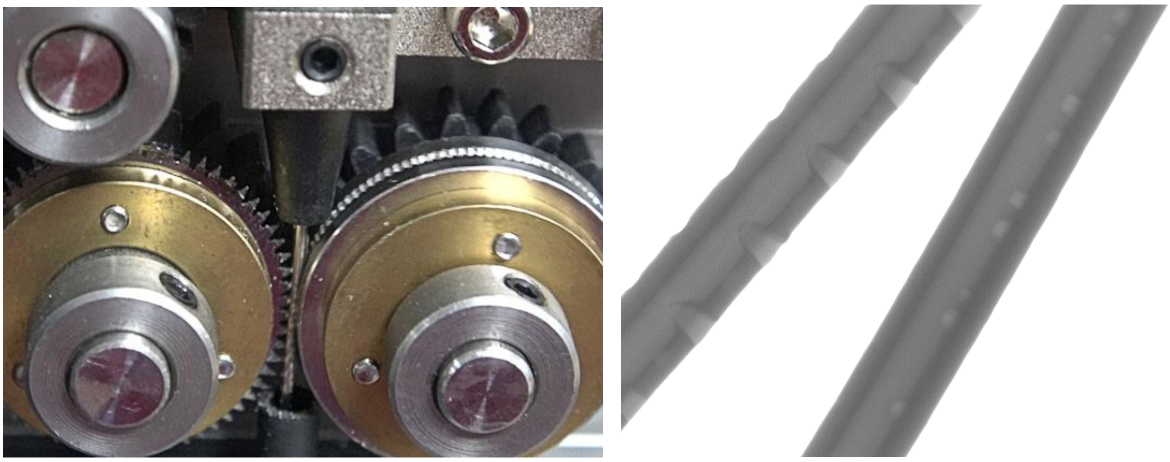
First example of a tabletop robotic soldering system with full enclosure. Close up of the system showing soldering iron, solder wire feed and local fume extraction www.kurtzersa.com

It is worth mentioning that, depending on which industry you work in, the meaning of High Melting Point (HMP) solder will vary. It is always important to check the alloy percentage in question rather than just assume that HMP means lead/tin.

During use in robotic soldering it is beneficial for the flux cored solder wire to be correctly punctured while it is being fed to the soldering iron tip. The indexing wheel should create holes/slots in the wire centrally and reach down to the core of the wire to be effective in reducing spitting. There are a couple of ways to see the depth of the puncture. A piece of wire can be taken and placed in an X-ray machine if available on site.

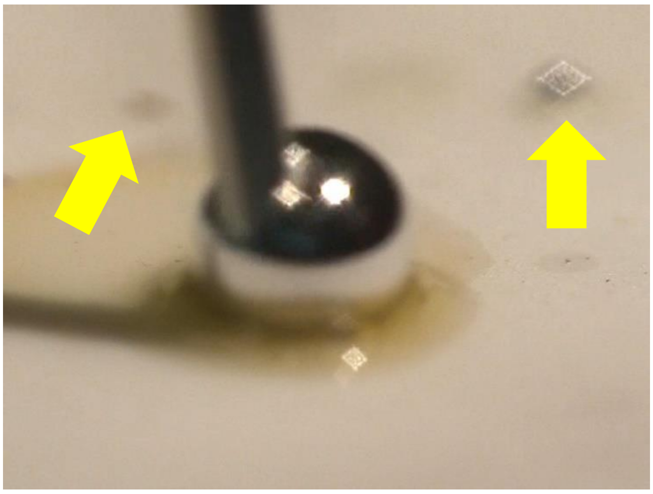
Alternatively, the end of a piece of cored wire after feeding can be placed on a soldering iron tip or a hot plate. For test the soldering temperature must be set just below the reflow temperature of the cored wire; this will allow visible evidence of the flux escaping from the indents to be easily seen. The goal is to eliminate high-pressure escape of volition flux and cause solder balls. Solder wire suppliers have also been working to improve the performance of spitting by re-examining their flux chemistry.

.



Images show solder wire feed mechanism and the indent blades. The X-ray image shows the wire after feeding and the indents in the flux core to reduce spitting

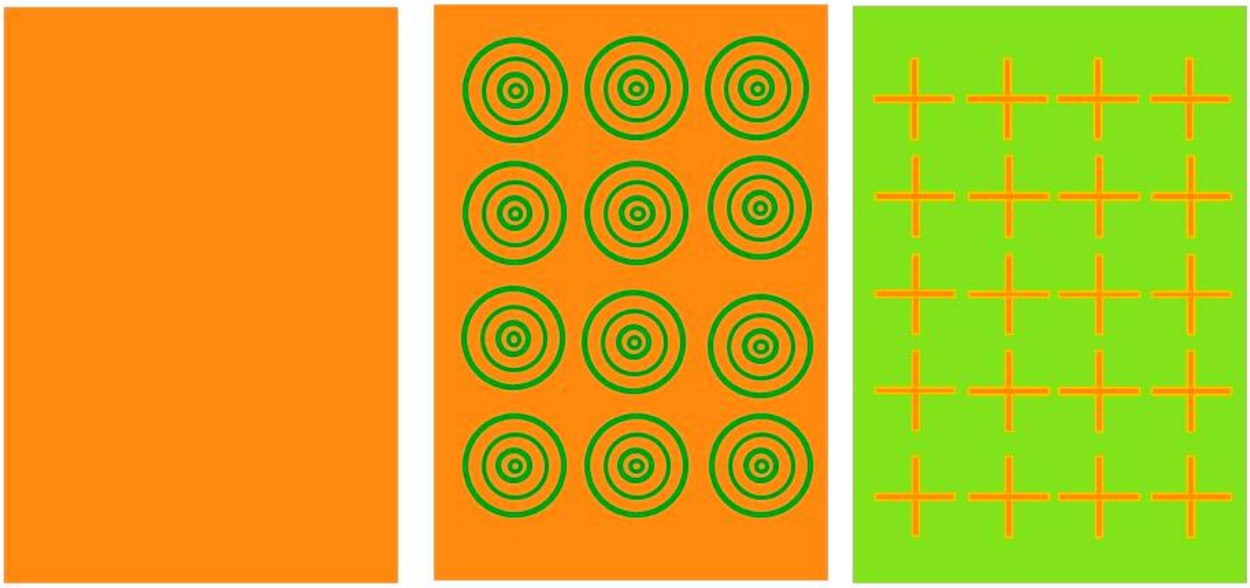
Three lengths of cored solder wire are shown, after placing on a hot plate set below the solder alloys reflow temperature. The three images show the flux rising in the indents created during wire feed and then flowing around the wire.



Simple demonstration of cored wire reflowed can be performed on ceramic plates with indented and non-indented wire showing impact on solder balling. Ideally a high-speed camera would be used to show solder balling, but the author did not have any budget!! The first image shows solder balling as indicated by the arrows. There is no balling in the second image and you can make out the flux escape and indent marks on the wire. We noted this solder balling issue on our second robotic soldering project in 2018. Some of the problems were due to the speed of soldering and changing the technique away from feeding to the heated joint. The preference due to throughput speed was to introduce the solder to the tip surface first.

Well before the robotic soldering process benefits of wire puncturing were noted, other engineers had developed wire scoring to aid faster hand soldering without balling. The wire used was scored with a “V” slot to again reduce the sudden pressure release during soldering. We could debate improvements in the flux chemistry at the time could have helped. However, it is great to see simple engineering solutions to problems. Wire puncturing is best for robotic soldering as it provides process benefits and the simple way of accurately indexing the cored wire to form the joints. Modern wire feeding heads strive to have the indexing points as close to the point of soldering to better control the feed length of the wire.

Above is the “V” score wire section, the process option offered to industry some years back and still commercially available with several upgrades in design see www.en.bonkote.co.jp/

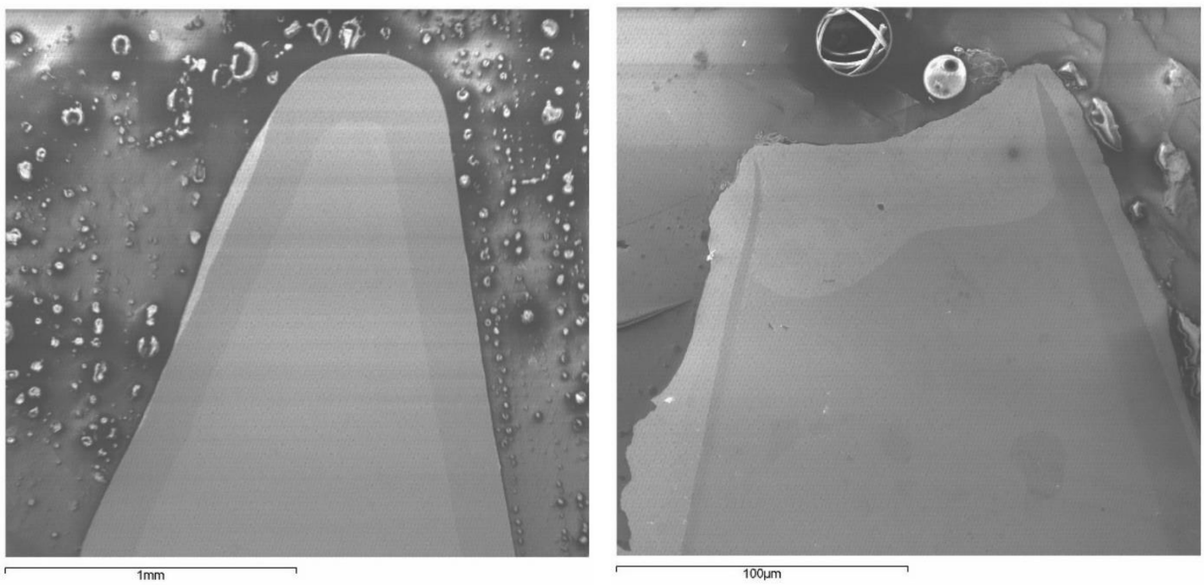


Solder spatter or wetting test boards could be designed as above with a single sided boards with copper OSP or nickel and gold flash surface finish. One is just a sheet of laminate, the second has a bomb site pattern. The soldering iron tip is located a fixed distance from the surface of the board before feeding a known length of wire to the solder tip. Other engineers have used small copper tiles and contact the surface with fixed soldering times. Any spitting is seen on the surface of the copper or gold.

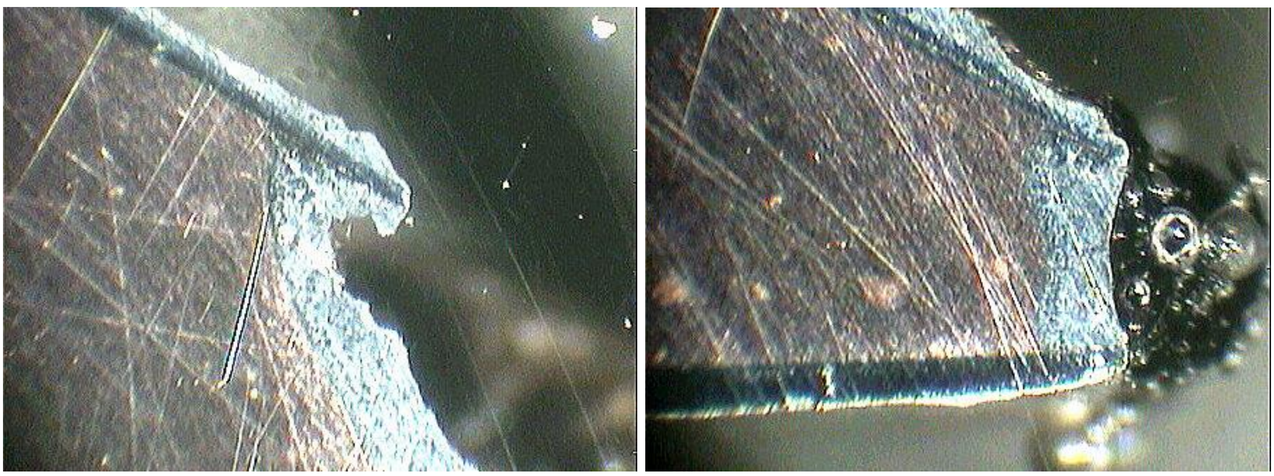
Although cheaper to use a sheet of copper laminate without etching the pattern having a target gives a reference for counting solder balls and splatter the cross pattern could be used for wetting tests and also to show the impact of nitrogen and temperature on the wetting performance. The solder/tip is added to the centre of the cross, the disadvantage of this is the impact of the test board on temperature. Unlike a reflow pattern the temperature drop during soldering would directly affect the wetting distance. However, if the actual production assembly required preheat as part of the process this may be a useful test technique.

Soldering Iron Tips

There are many different types of soldering iron tips, each designed to provide the best possible solution for different soldering tasks. Some tips can be obtained from different sources so care needs to be taken in understanding consistent manufacturing quality. Some supplier brands suffered from poor and inconsistent plating which was highlighted during the original transition from tin/lead to lead-free. Often the operator was blamed but the root cause was plating quality. In the case of automated soldering the temperature, contact pressure and the number of soldering operations per tip are recorded. This allows better comparison if alternative suppliers become available or are considered. There is less choice today in tip suppliers as these items have more custom features required for automated soldering.



SEM Images above clearly show satisfactory plating and a failure point on one tip. The barrier layer on the tip is missing and the copper core has been slowly dissolved



Hand soldering tip microsections after plating failure during introduction with lead-free soldering

After the industry's introduction of lead-free and the issues of tip life solder wire producers worked with iron manufactures to help develop wire combination to improve the life of the tips and reduce the degree of oxide formation. Soldering iron tips are the doing part of the process just like in hand soldering and they will have a limited life. Through experience production staff running an automated process should be able to define when to change a soldering tip before it starts causing process defects. Planned changes are much more efficient than defects causing unplanned maintenance.

To deal with high thermal demand solder joints in multiple position on boards selection of a high wattage soldering iron may be necessary. Where a company regularly solders high mass boards for the automotive and high voltage applications moving from a 120 – 200 watt may be beneficial

Nitrogen Soldering

The use of nitrogen in manual soldering has been available for many years but when first introduced the cost and benefit were debatable. Our forced introduction of lead-free soldering at elevated temperatures did increase interest and possible benefits for using nitrogen. In the case of manual soldering the tooling supplier does need to adapt his soldering tools to accommodate a nitrogen flow directly to the point of soldering to ensure a reasonably inert atmosphere is created around the tip and joint area. Most suppliers of soldering irons and robotic machines offer a system option, or a separate unit can be integrated to a workstation.

In our experience it is not necessarily the obvious reasons for using nitrogen like better wetting, less flux decomposition and reduced oxide formation on the tip. These are all practical benefits in soldering but more so when you automate a process and want the minimal operator process intervention for cleaning. A build up of oxide and flux on a tip can lead to contamination on the joint surface or poorly formed joints. It is also possible to see residues fall on to the surface of an assembly during head movement to the next connection point forcing early cleaning cycles.

We have experienced benefits of using a localised nitrogen generator on soldering consistency. One of which related to the preheating benefits to the speed of soldering. By preheating the joint area and the wire as it is fed to the joint interfaces the soldering operation will benefit the total process cycle. It is very useful to use a thermal imaging camera to track the changes in temperature around the joint area. This can be done while adjusting the temperature and the flow rate of the gas. This is best achieved with nitrogen being fed around the iron tip with a specifically design tool.

The thermal IR images on page 17 were taken during a customer project to improve the soldering process cycle and to try and reduce defects in manufacture. Maintenance departments on site often have these useful tools that process engineers may also benefit from. They are normally used for tracking hot spots in buildings, over heating on wiring, thermal heat loss on building etc.

Close up around the soldering iron tip where nitrogen flows towards the soldering area during initial heating and soldering operation

Close up view above of the nitrogen feed through the soldering iron collar to the tip of the iron on an alternative suppliers product

Flux Application

We have not specifically conducted any robotic soldering trials with solid wire without flux. However, we have created joints and videoed these to demonstrate successful soldering and rework of through hole joints with LTS Tin/Bismuth and Tin/Bismuth/Silver wire. One of the issues is the brittleness of the alloy and the difficulty in forming cored wire at the sizes normally used in manual soldering or rework. Suppliers do not want to make these wires unless there is a real market for them. Typically 0.6 - 1mm cored wire is used for rework of joints in production. Tin/copper cored wire is often used for rework and touch up even with low temperature alloy joints from reflow, selective or wave soldering.

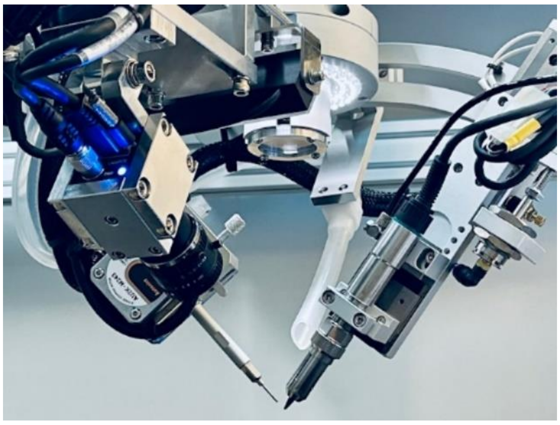
Solder Paste Application

It is also possible to use solder paste jetting to a pre-assembled board for possible laser soldering. This option is available from a couple of suppliers, but the solder paste must be selected for this specific application. Paste suppliers are well aware of the needs for special paste for high-speed jetting. However, the combination of this type of paste and the faster heating and reflow speeds with laser need to be carefully evaluated to prevent paste slump, displacement and solder balling. Laser soldering has been successfully used on pins for intrusive reflow but in this case why use robotic soldering.

Solder paste can be dispensed around or over pins as an alternative process when connectors have been preassembled. This was first demonstrated by Camelot at one of the authors **Pin In Hole Intrusive Reflow Design** workshops with nitrogen or vapour phase reflow not laser

Robotic laser soldering has improved significantly from the first of the author’s projects with cored wire. The solder joint formation and optimisation of the joints was very repeatable but the actual heating repeatability of areas on the substrates was not on some equipment

Example of a robotic soldering system with the control software visible along with a close up of the soldering operation. A system like this would feature all safety curtains, local extraction and may include pre heat for high mass boards www.promationusa.com



Robotic soldering station head assembly. Close up view shows the iron, cored solder wire feed, local fume extraction, camera and lighting www.promationusa.com

Dual soldering heads are another potential option for two work areas. One board can be loaded while a second is being soldered. There are different options for safety interlocks on machines www.promationusa.com

Soldering Tip Cleaners

There are three different cleaning stations available from robotic system suppliers. Just like manual soldering it is necessary to clean tips after a soldering operation. The frequency of cleaning is dependent on a number of factors, temperature, flux type, solder alloy, also the shape of the tip is a factor in your choice. It became common for drag soldering to be used to speed up soldering on connector locations and due to tip shape the cleaning steps may be more of a challenge. In addition, users have found that a static cleaning point followed by a rotary cleaning brush may be more effective on split tips

Experience shows that just lowering the tip into the cleaning media is not ideal. If you have a lot of hand soldering experience, do you just push the tip into a sponge or metal brush? Based on the soldering experience with your robot and observation after soldering groups of termination will allow the best tip cleaning positioning and movement. Often the tip may be pushed into the rotating the brushes then moved to one side before withdrawing the tip

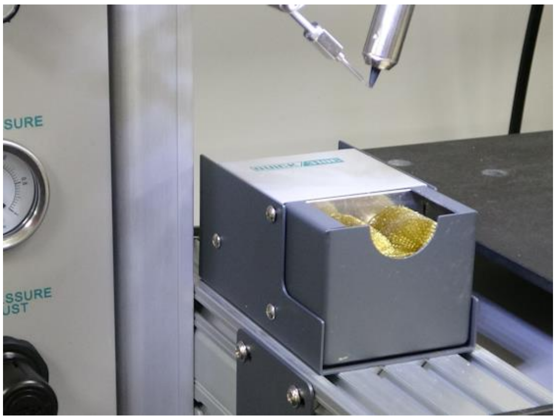
Whichever system is used it must clean the tip with no damage to the plating or cause a change in tip temperature. Some suppliers have used colour in either fixturing/brushes to avoid brushes being used on tin/lead and lead-free solder to avoid any cross contamination.

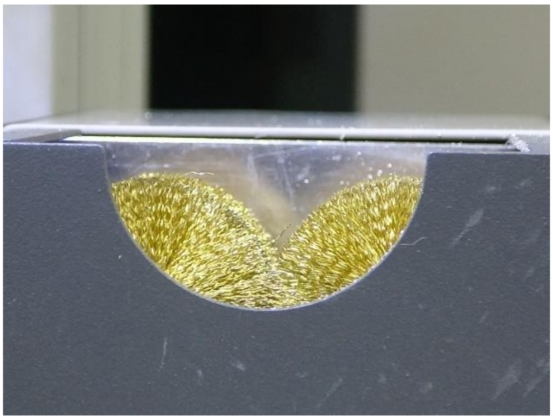
Sponges

Moistened sponges are the traditional solution for manual soldering and still work well. This is provided they are not overly wet. As a trainer I often see excess water added to sponges in production. The concern with a simple sponge in an automated work cell is forgetting to add water periodically, out of sight out of mind. In addition robotic systems are normally confined in a perspex cabinet where the temperature can rise leading to possible quicker drying out of sponges

Metal Brush

Fine metal brushes work well provided they are also maintained. Often these get clogged with flux residues, but this is dependent on the wire used and infrequent maintenance. Metal brush systems come in two types either static, which best resemble metal swarf formed when machining metal in a tray, or a rotary brush





Two views of the fine metal brushes used for cleaning tips with rotary movement of the brushes and a combination of tip movement to achieve satisfactory results

Air Blow Off

More recently high-pressure air blast has been used to remove residues on the soldering iron tip. Units capable of blowing in one, two or with a circular blast around the tip to remove residue solder, flux or other contamination. In the past a combination of rotary wire brushes has been available for initial cleaning then as the iron tip is removed an air jet is used for a final cleaning operation. There are different options and engineers need to look at the best option for the process, product and material being used.

Tip Changeover Stations

It is a benefit to some customers to be able to use more than one soldering iron tip size or shape. This is particularly true if the assembly to be soldered has a requirement for both individual points to solder and a need for drag soldering on multiple terminals.

Having the ability to change tips as part of a process sequence may be beneficial but it will have a small impact on cycle time. Automated changing of tips is possible with some suppliers; however, the changeover station must provide the opportunity to check the actual tip length after change. Alternatively, a separate sensor needs to be available to make sure of the accurate positioning of the tip z height before contact with solder joint interfaces.

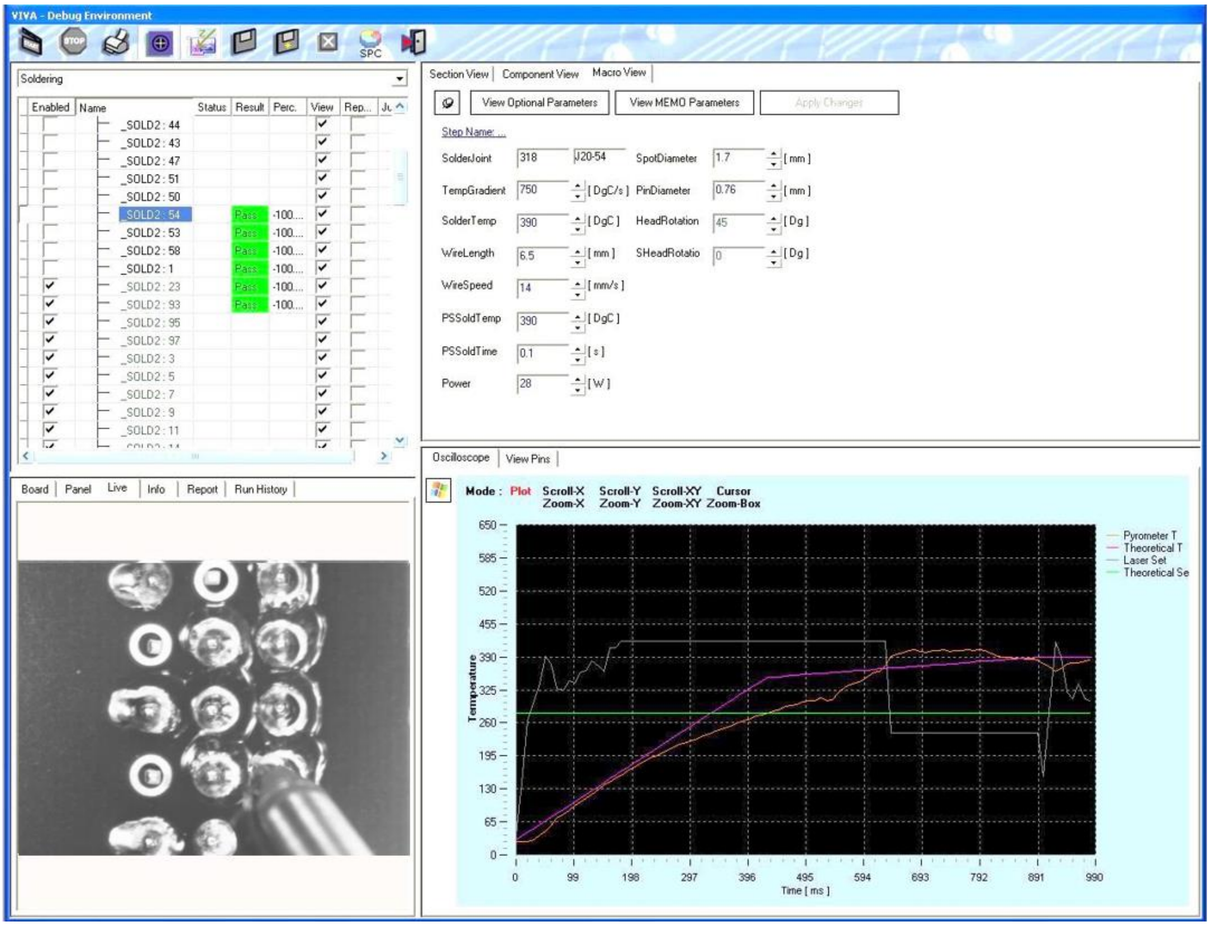
PCB Preheating

Preheating a board assembly may be necessary to overcome thermal issues caused by the design or where the components are thermally demanding. Either of these issues would make it difficult to obtain satisfactory hole fill and slow the process. Moving from 1-4mm thick board would also make soldering difficult due to the mass. Any of these issues have often required board pre-heating for manual soldering particularly with the move to lead-free. Products that require rework and desoldering are often preheated.

We have already highlighted that some preheating may come from the nitrogen if used around the tip of the iron. There is of course some heating of the cored wire before contacting the iron tip. Ideally, we want to avoid higher setpoint temperatures on the iron or longer contact times between the soldering tin and component termination and pad.

Preheating boards is not that common on robotic systems but inevitably it may become a more common option in the future. There are two different preheating systems that may become common in the future, IR and panel heaters. IR will probably be more popular as they are faster to respond and turn off. A solid panel heater may be less expensive, but it will contribute to constant heat input around the working area of the equipment. This in turn may also dry out cleaning sponges more quickly.

It will be important to conduct some form of temperature measurement on each high mass board design during your Design for Manufacture Review. This could be conducted with a thermal imaging camera or thermocouples and a datalogger normally used with reflow soldering. Thermocouples may be more effective when specifically looking at the impact of high mass or temperature sensitive components. In these cases the thermocouples may be mounted in plated through holes or under the body of parts.



Above an example of SEICA software traceability per joint which is ideal for engineers to have data for process audits and for customers that want full traceability during laser soldering. During our NPL project we generated 100’s of video files with profiling data with high temperature solder www.seica.com

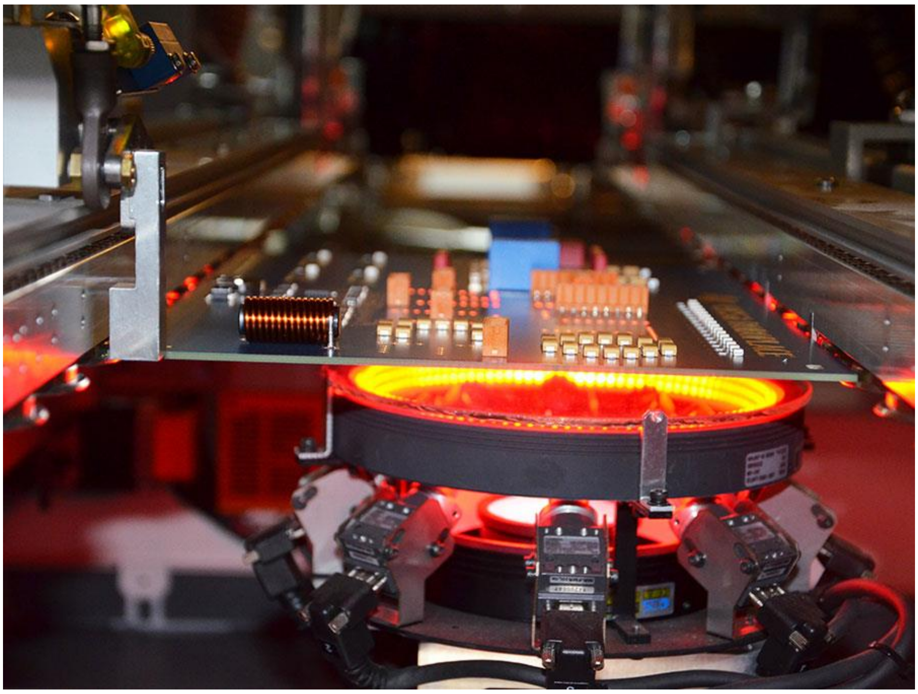
Japan Unix have management software for monitoring laser soldering as shown above. In addition they have the capability to monitor key data on contact soldering. A key point for traceability on both automotive and medical applications www.japanunix.com

Solder Joint Inspection

The majority of through hole solder joints are inspected manually with the same industry criteria used for inspection of wave, hand or selective joints. Included in this book are many examples of robotically produced solder joints with different alloys. In all cases it is possible to inspect solder joints on the opposite side to the component and connector. There are many situations where joints are not visible due to the component body. With a well-controlled, defined and optimised process you should easily exceed the minimum requirements of IPC 610 standards.

Automatic Optical Inspection

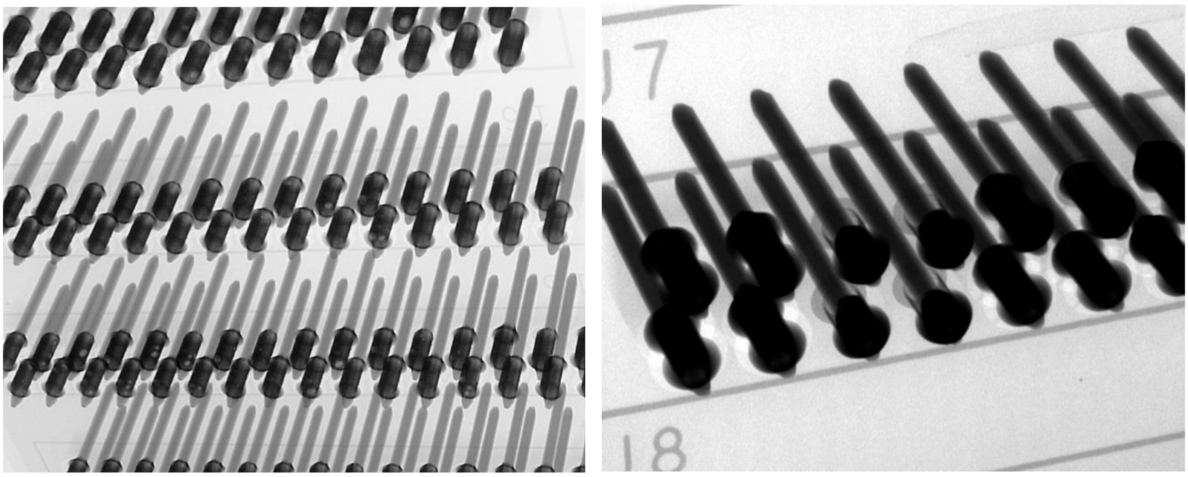
In recent years there has been an increase in the use of Automatic Optical Inspection (AOI) mainly for surface mount terminations. Suppliers have adapted their vision technology/lighting and added algorithms to allow assessment of through hole joints. This speeds up inspection; however, it does not allow assessment of through hole fill and topside fillet inspection.



Example of AOI being used for through hole solder joints www.kurtzersa.com

X-Ray Inspection

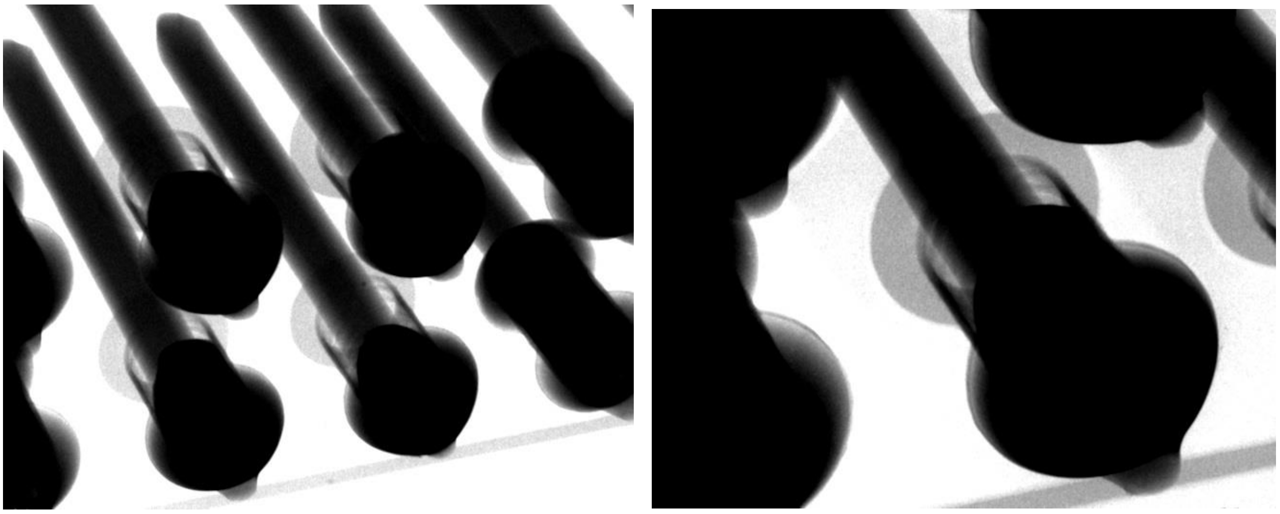
Since the introduction of area array packages like Ball Grid Array (BGA), Quad Flat Pack (QFP) and flip chip, X-ray inspection has been more readily available. Many X-ray systems today allow for manual and semi-automatic inspection of hidden joints which allow for complete through hole joints to be inspected as illustrated below in this low magnification view. However, many customers, particularly those in automotive, will want a better view of the topside fillet wetting



Multiple satisfactory solder joints produced with LTS on a test board. All of the joints show joints in excess of 75% fill of the holes and most with topside fillets

Regardless of the specifications most people do want to see topside fillets and 100% fill. This is fine as your goal in manufacture but not with repeated rework. If you meet the required standard for the product do not just accept less, work hard to improve process quality as an ongoing project, don't remove unnecessarily

As this author often said in his workshops "Always aim to continually improve your process to prevent unnecessary rework a process does not get better by itself"



X-ray images above show a range of solder fill between 50-100%. Selected joints have topside fillets which would not be visible with the connector body over joints

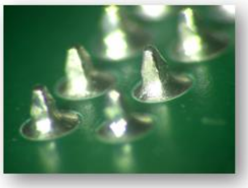
Depending on the X-ray machine's capability it is possible to highlight process or environmental testing defects with clarity. X-ray systems improve all of the time and if you can't afford but need high resolution capability consider contracting out inspection services when required. There are many occasions the author has seen less capable X-ray systems purchased simply to tick an audit requirement which is very poor practice. If it can't show you what you want and need to see it's just not much use!!

Bobwillis.co.uk

Low Temperature Solder Joint Inspection Criteria

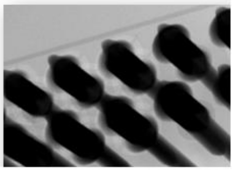
Satisfactory

Solder joints produced with Tin/Bismuth/Silver (SnBiAg) with a robotic soldering process at 240°C. Although the solder does not cover all of the pad surface, this is perfectly acceptable solder joints



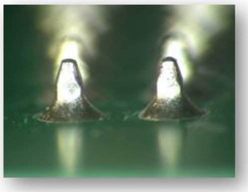
Satisfactory

Satisfactory joints formed with SnBiAg solder alloy during robotic soldering. X-ray inspection was used to show the degree of the through hole fill on a copper OSP finish printed board assembly



Satisfactory

SnBiAg solder joint produced with a robotic iron soldering process at 240°C. The solder has formed a perfect solder joint with a concave fillet formation



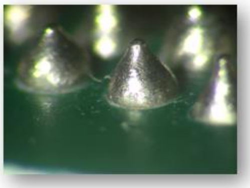
Produced by: Bob Willis bob@bobwillis.co.uk www.bobwillis.co.uk

Bobwillis.co.uk

Low Temperature Solder Joint Inspection Criteria

Satisfactory

Solder joints produced with Tin/Bismuth (SnBi) on a robotic soldering process at 240°C



Satisfactory

Satisfactory joints formed with SnBi solder alloy during robotic soldering. X-ray inspection was used to show the degree of the through hole fill on a copper OSP finish printed board assembly

Satisfactory

Solder joint produced with SnBi alloy and a robotic iron soldering process at 240°C. The solder has formed satisfactory solder joints on the top side of the board. The solder has not fully wetted the nickel/gold pad surface

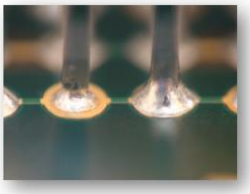
Produced by: Bob Willis bob@bobwillis.co.uk www.bobwillis.co.uk

Bobwillis.co.uk

High Temperature Solder Joint Inspection Criteria

Satisfactory

Solder joints produced with tin/copper on a laser soldering process. Although the solder does not cover all of the pad, this is perfectly acceptable. There is variation in the solder volume per joint which is not uncommon with automated wire feed process

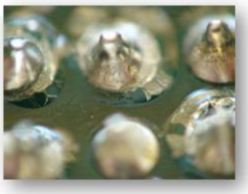


Satisfactory

Satisfactory joints formed with tin/copper with laser soldering. X-ray inspection was used to show the degree of the through hole fill on the printed board assembly

Satisfactory

Solder joints produced with tin/copper with laser soldering process. The solder has formed a perfect solder joint with a concave fillet formation. There is excess flux present on this example but if cleaning is used this would not be a problem



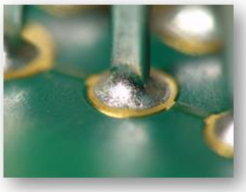
Produced by: Bob Willis bob@bobwillis.co.uk www.bobwillis.co.uk

Bobwillis.co.uk

High Temperature Solder Joint Inspection Criteria

Satisfactory

Solder joints produced with SnSb on a selective soldering process at 340°C. Although the solder does not cover all of the gold pad, this is perfectly acceptable

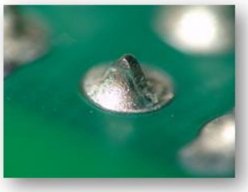


Satisfactory

Satisfactory joints formed with SnSb solder alloy during selective soldering. X-ray inspection was used to show the degree of the through hole fill on a gold finish printed board assembly

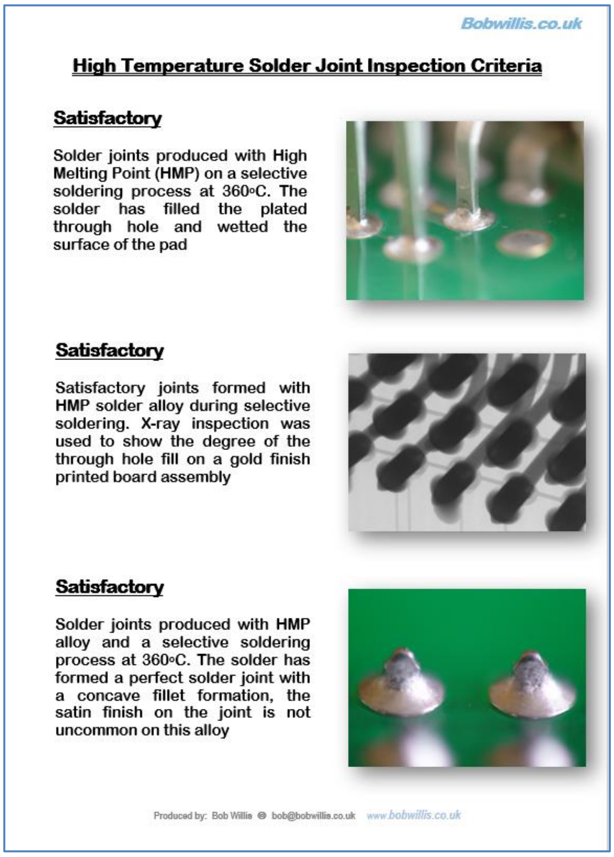
Satisfactory

Solder joints produced with SnSb alloy and a selective soldering process at 340°C. The solder has formed a perfect solder joint with a concave fillet formation



Produced by: Bob Willis bob@bobwillis.co.uk www.bobwillis.co.uk

Our inspection wall chart set are all available FREE to download via the smta.org website as part of training media package sponsorship



Bobwillis.co.uk

High Temperature Solder Joint Inspection Criteria

Satisfactory

Solder joints produced with High Melting Point (HMP) on a selective soldering process at 360°C. The solder has filled the plated through hole and wetted the surface of the pad

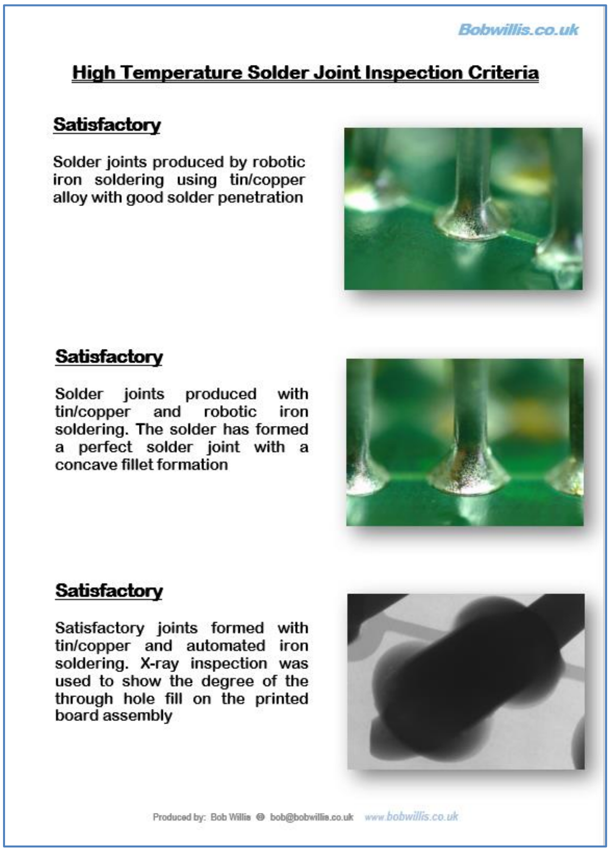
Satisfactory

Satisfactory joints formed with HMP solder alloy during selective soldering. X-ray inspection was used to show the degree of the through hole fill on a gold finish printed board assembly

Satisfactory

Solder joints produced with HMP alloy and a selective soldering process at 360°C. The solder has formed a perfect solder joint with a concave fillet formation, the satin finish on the joint is not uncommon on this alloy

Produced by: Bob Willis ⊕ bob@bobwillis.co.uk www.bobwillis.co.uk



Bobwillis.co.uk

High Temperature Solder Joint Inspection Criteria

Satisfactory

Solder joints produced by robotic iron soldering using tin/copper alloy with good solder penetration

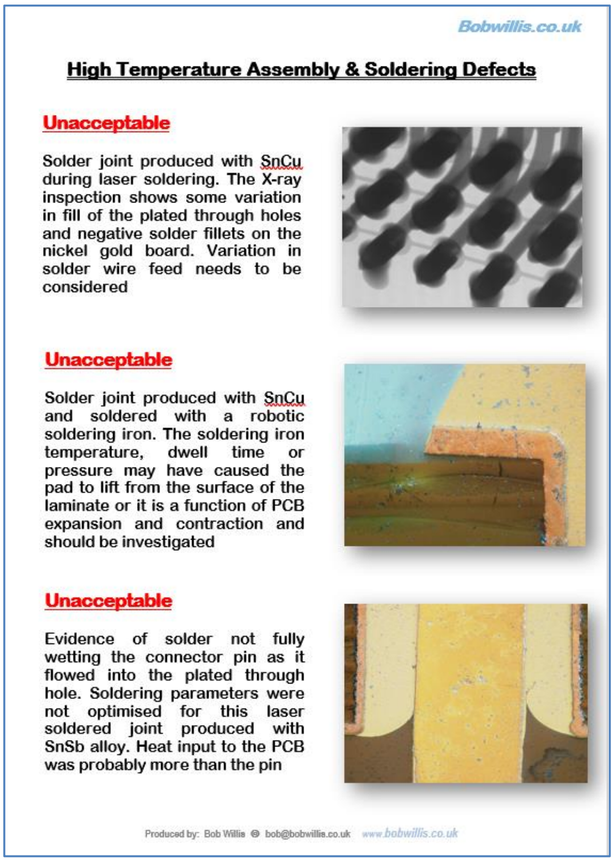
Satisfactory

Solder joints produced with tin/copper and robotic iron soldering. The solder has formed a perfect solder joint with a concave fillet formation

Satisfactory

Satisfactory joints formed with tin/copper and automated iron soldering. X-ray inspection was used to show the degree of the through hole fill on the printed board assembly

Produced by: Bob Willis ⊕ bob@bobwillis.co.uk www.bobwillis.co.uk



Bobwillis.co.uk

High Temperature Assembly & Soldering Defects

Unacceptable

Solder joint produced with SnCu during laser soldering. The X-ray inspection shows some variation in fill of the plated through holes and negative solder fillets on the nickel gold board. Variation in solder wire feed needs to be considered

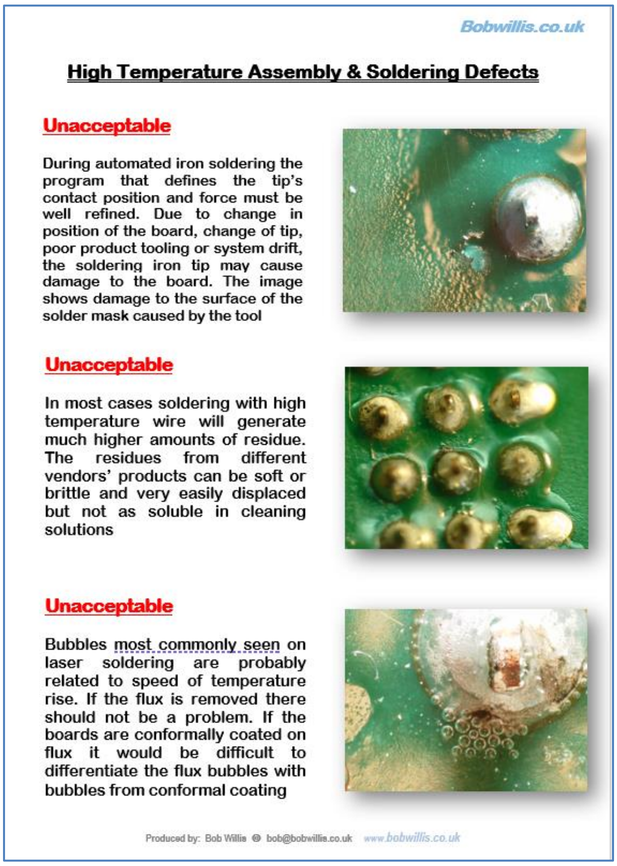
Unacceptable

Solder joint produced with SnCu and soldered with a robotic soldering iron. The soldering iron temperature, dwell time or pressure may have caused the pad to lift from the surface of the laminate or it is a function of PCB expansion and contraction and should be investigated

Unacceptable

Evidence of solder not fully wetting the connector pin as it flowed into the plated through hole. Soldering parameters were not optimised for this laser soldered joint produced with SnSb alloy. Heat input to the PCB was probably more than the pin

Produced by: Bob Willis ⊕ bob@bobwillis.co.uk www.bobwillis.co.uk



Bobwillis.co.uk

High Temperature Assembly & Soldering Defects

Unacceptable

During automated iron soldering the program that defines the tip's contact position and force must be well refined. Due to change in position of the board, change of tip, poor product tooling or system drift, the soldering iron tip may cause damage to the board. The image shows damage to the surface of the solder mask caused by the tool

Unacceptable

In most cases soldering with high temperature wire will generate much higher amounts of residue. The residues from different vendors' products can be soft or brittle and very easily displaced but not as soluble in cleaning solutions

Unacceptable

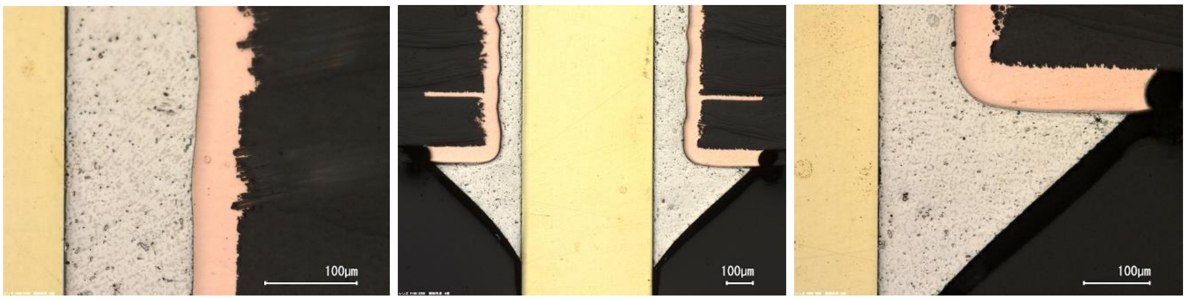
Bubbles most commonly seen on laser soldering are probably related to speed of temperature rise. If the flux is removed there should not be a problem. If the boards are conformally coated on flux it would be difficult to differentiate the flux bubbles with bubbles from conformal coating

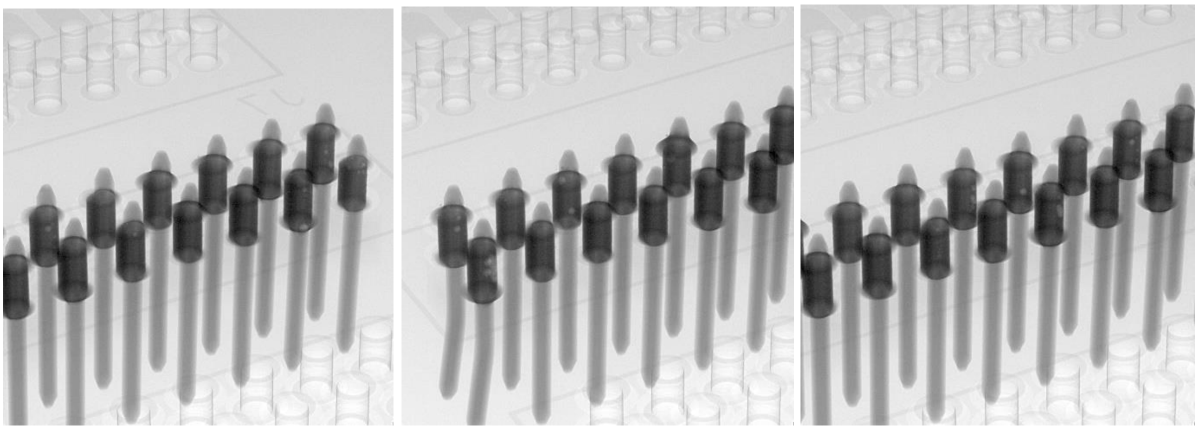
Produced by: Bob Willis ⊕ bob@bobwillis.co.uk www.bobwillis.co.uk

Our inspection wall chart set are all available FREE to download via the smta.org website as part of a training media package sponsorship

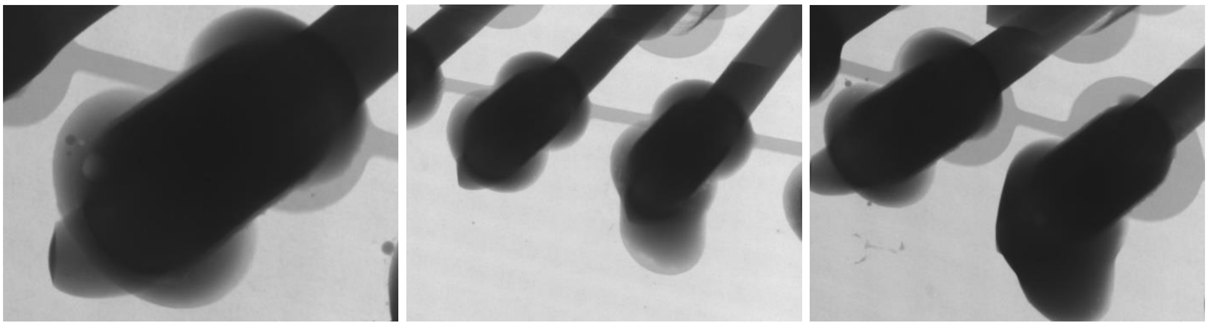
Selected Process Trials Results

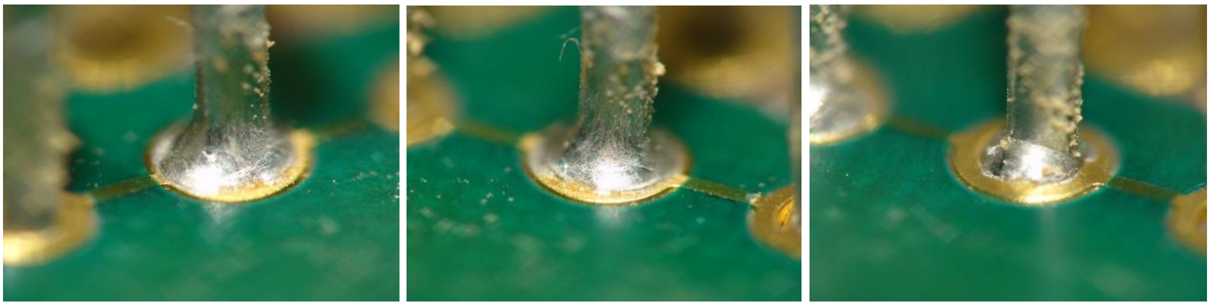
For reference we have included results taken from our robotic soldering trials over the last couple of years. They include optical and microsection views of different solder alloys and complement the author's Robotic Soldering Inspection Wall Charts that can be downloaded FREE from the SMTA website by searching in their Knowledge Base www.smta.org



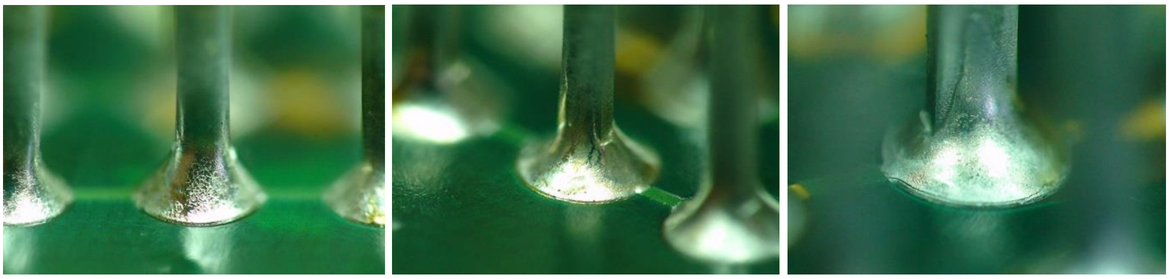


Three microsections shown above taken after successfully soldering with LTS Tempsave B37 with a soldering system from ***Promation USA****. A set point temperature of 265°C was used. Sample boards featured both gold over nickel and copper OSP. The X-ray images were taken from the same project*

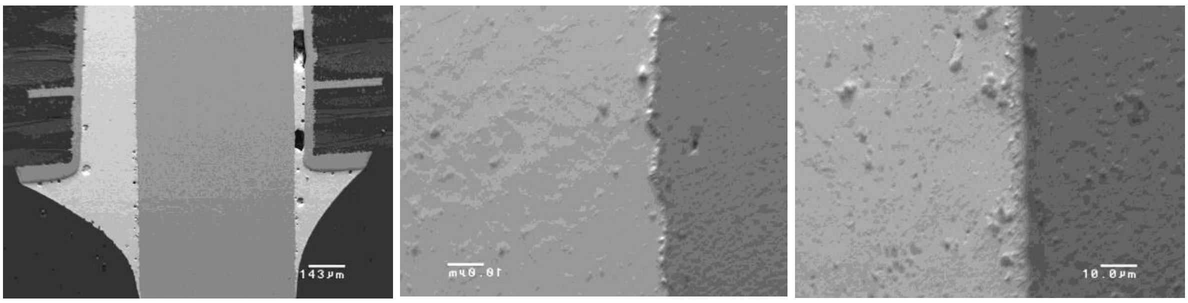




Three X-ray and optical images above taken on gold over nickel boards soldered with HMP solder using a laser system from ***Wolf Produktionssysteme GmbH.*** *Samples on this project were produced successfully with contact and laser soldering*



Three optical images above taken from gold over nickel boards soldered with SN96 solder wire using a system from ***mta Automation SA***

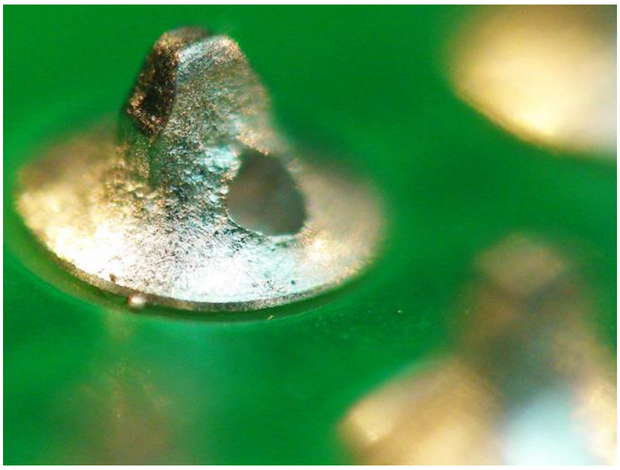


Three SEM images above taken from gold over nickel boards micro sectioned after soldering with SAC solder wire using the ***"Firefly"*** *laser soldering system from* ***Seica****. The voids present are a function of the PCB outgassing not the soldering process or wire*

Process Defects – Causes & Cures

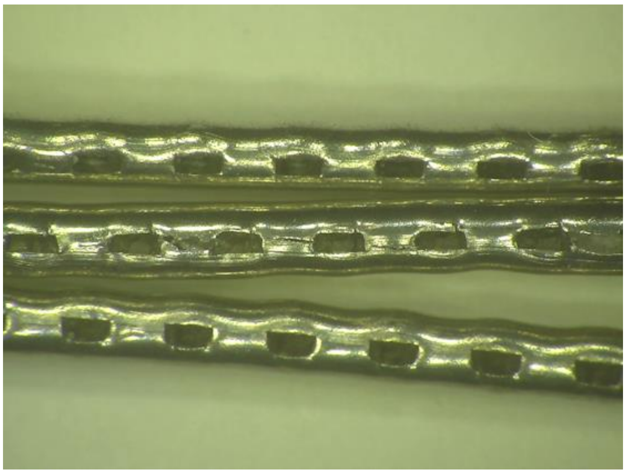
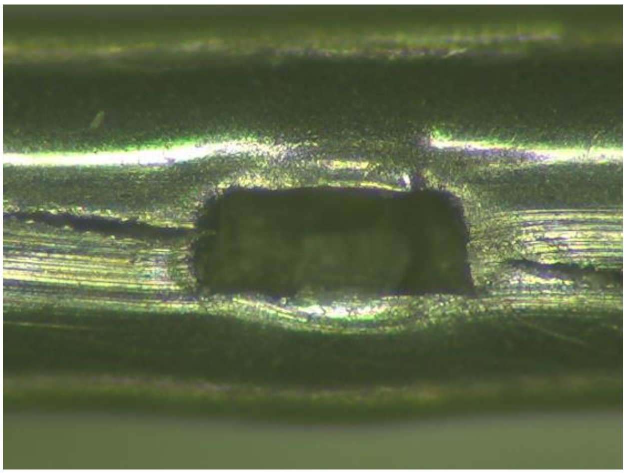
The following defect examples have been experienced over the last few years. During process investigations and use of robotic soldering in factories and technical institutes has allowed us to look more deeply into the process. Some of the examples were also part of our NPL project on high temperature soldering with Dr Chris Hunt. A copy of the **NPL Good Practice Guide on High Temperature Soldering** can be downloaded from the NPL website. Also defects and videos seen during the launch of our proposed exhibition feature **"The Robotic Soldering Experience"** are available to assist engineers understand the process and viewed online see page ?

PCB Outgassing Voids

During soldering of plated through holes outgassing from the hole can always occur when water vapour escapes from the board through thin copper or breaks in the copper plating creating voids in the solder. As the solder solidifies either a small or large void can be visible as shown. Voids are very unlikely to cause any issues and there are simple ways of testing boards for outgassing. We have investigated outgassing many times over the years with a simple oil test to show what happens during heating, it is also featured on our ***YouTube*** videos. It is always best to fix the real reason for the problem.

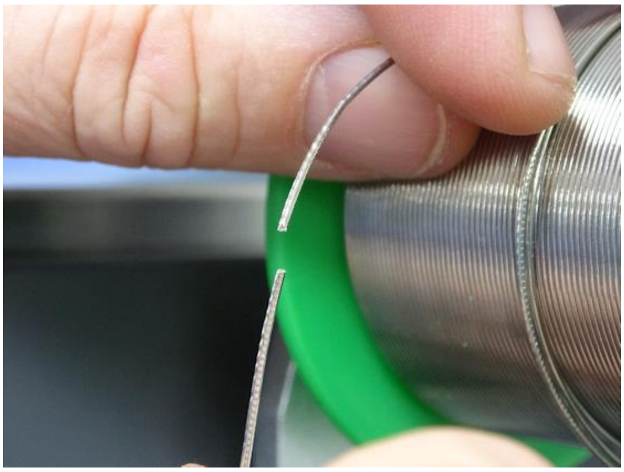
Cracking of Low Temperature Cored Wire

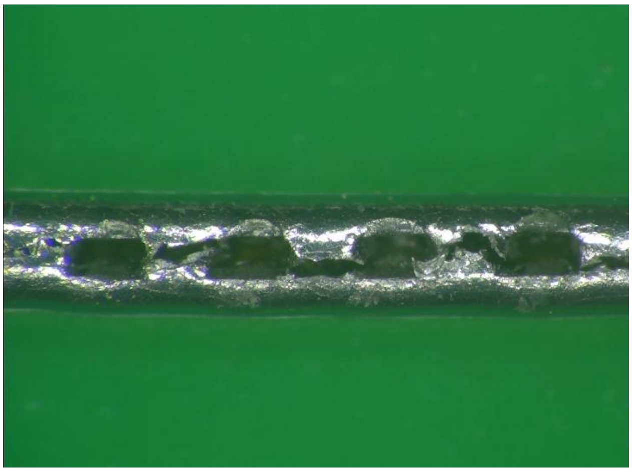



When cored solder wire is fed/indexed through to the soldering iron tip or laser beam the drive wheels indent the wire. This is a good thing that shows positive feeding but also one of the ways of eliminating spitting and solder balls forming. In this case the cracking of the wire is due to the hardness of the solder wire. SnBi & SnBiAg low temperature solder is not very ductile hence cracking can be visible. Although this does occur on the machine, we have not experienced it causing breakages in our process trials or process reports in production.

We have experienced wire breaking during handling and when a reel was dropped before positioning on the machine. We also believe that the wire can age during storage. We have no specific data but have experienced that over a six/nine month period there was a change in the apparent brightness and ductility of a batch of cored wire we were using for testing in the US and UK. Very good friends in Japan from Nihon Superior, Tetsuro Nishimura & Keith Sweatman have investigated this phenomena with the Queensland University, Australia.

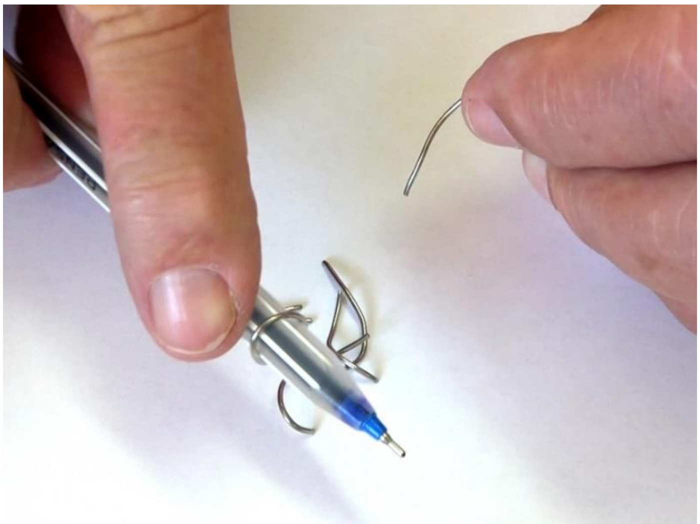
LTS Cored Wire Breaking





We have had experience with LTS Low Temperature Solder wire breaking with poor handling, but never on a robotic soldering machine in normal operation. Loading the wire to the machine must be done carefully. We have only seen breakage when the reel of wire was previously dropped. We have seen wire lengths that have already been pierced in the feeding head then break when re-fed. This is probably due to cracking of the wire when indented to the flux core which is illustrated in the second image; this would not be normal practice.

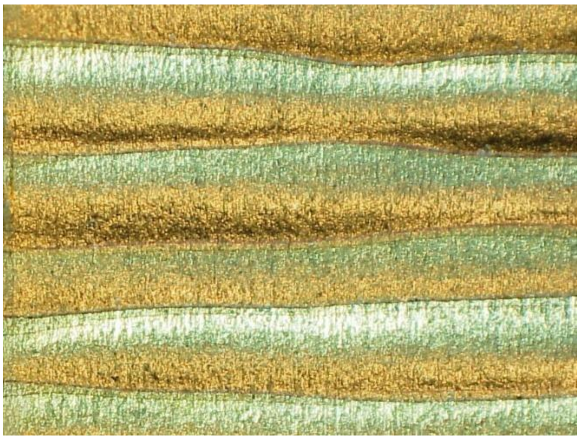
Testing LTS Cored Wire

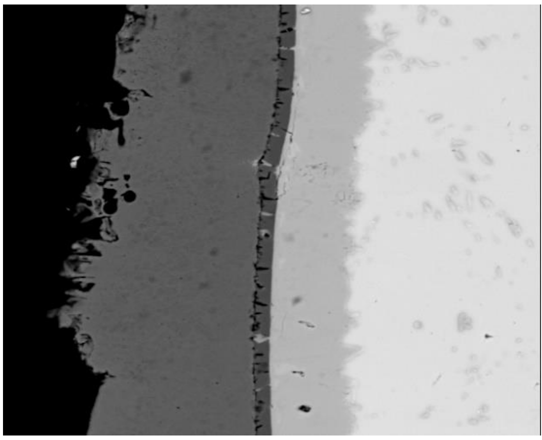


Very simple test for cored wire to illustrate brittleness. The author was winding a sample of wire around a pen with little success. This sample was as reeled although nine months old from the date of manufacture. This is very easy to perform successfully with tin/copper, tin/copper/silver, tin/lead. We have often used this method to make simple solder preforms for reflow soldering. However, with brittle wire it is very difficult with tin/bismuth and tin/bismuth/silver. During our low temperature trials we noticed a change in the ability to handle the wire over time. A new sample performed much better than samples used after 9-12months.

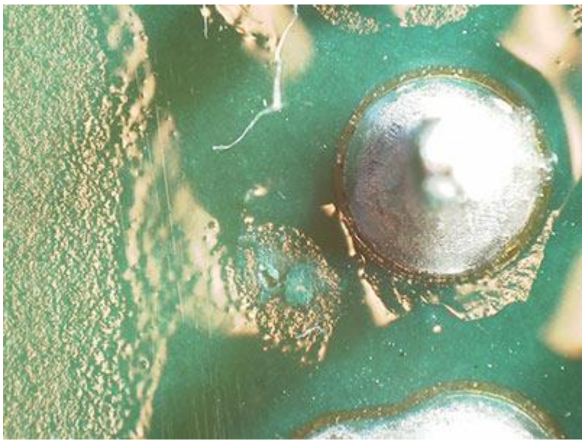
Nickel Layer Cracking

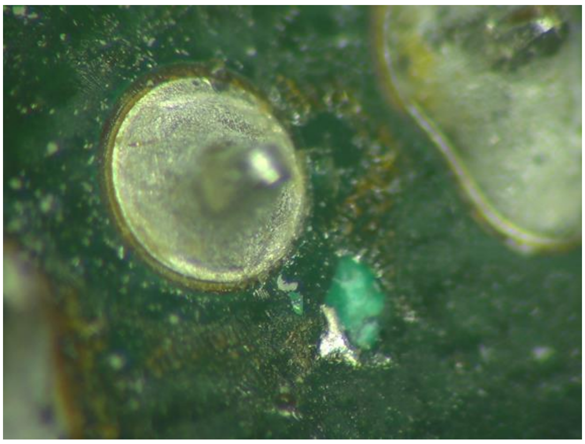
As part of the NPL project we conducted long term storage 1000hrs at elevated temperature 200ºC on different alloys which had been soldered by laser and contact robotic soldering. The real reason for this was just to gain an understanding of the impact of the time/temperature/alloy. The interesting chance was the PCB nickel layer becoming more brittle. During peel testing of copper/nickel/gold tracks you could see and hear evidence of cracking as the samples were peeled from the substrate. The images below show the cracking visible on the surface of the test foils. SEM view of the microsection show cracks visible in the nickel which was standard surface finish on a board.





PCB Substrate Damage



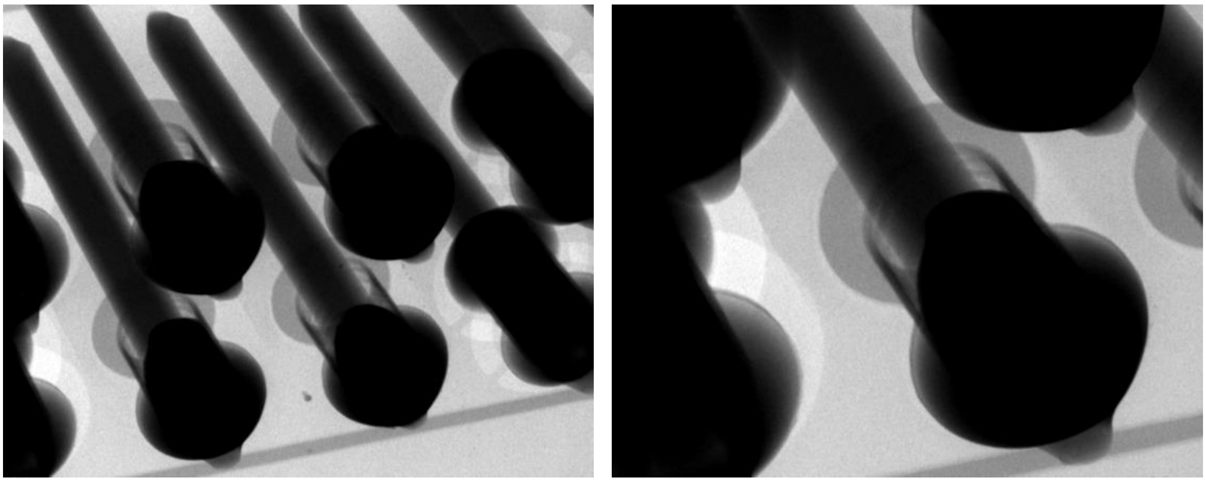


Although products designed for high temperature applications with organic substrates can stand up to high soldering temperatures, they can still be damaged. This can occur if the soldering times or temperatures are too high or, in the case of soldering irons, the tip comes into contact with the surface of the board or pad for too long a period of time.

During automated soldering the program that defines the tip's contact position and force must be well refined. If, due to change in position of the board, change of tip, poor product tooling or system drift, the soldering iron tip will cause damage to the board. It is also possible for damage if the z height position of the tool tip is poorly controlled with no pressure sensing. The image above shows damage to the surface of the solder mask caused by the tip. It is, however, possible to change the shape of the pad so it has a teardrop shape or a tail opening in the solder mask where the tip can land and make contact. This makes better thermal contact between the solder on the tip and the pad increasing the speed of soldering without damage to the board surface.

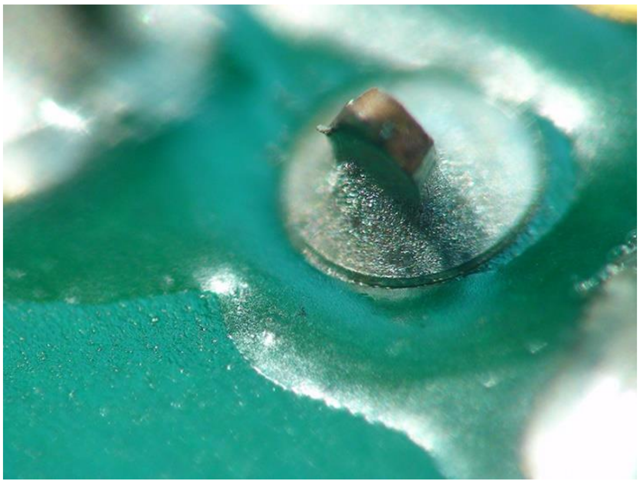
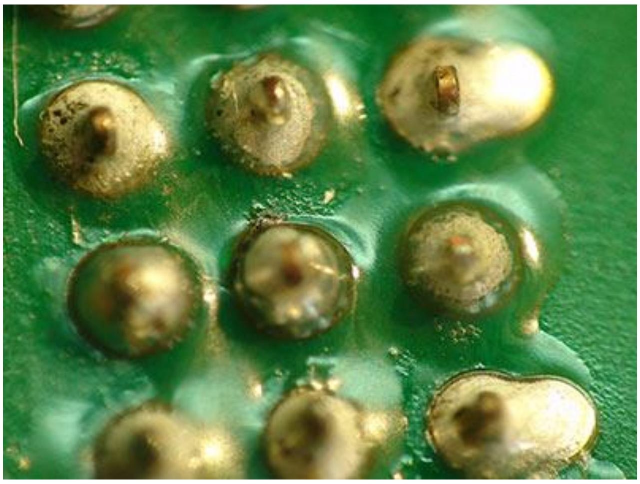
Although not illustrated in the examples damage can occur on the pad and solder mask with special forked soldering iron tips when drag soldering. Drag soldering does have the advantage of more speed.

Variation in Solder Fill



The solder volume varies on these joints when inspected with X-ray. However, it is often the case that the topside of IDC connectors is not visible optically, but this should be considered during process set-up. It is not standard practice for topside joints to be checked as they are not visible and although AOI can be used for bottom side joints it is not possible to use this on topside. We do find some automotive and military producers do run batch samples though X-ray as a process control measure, but good NPI procedures should pick up solder volume variation.

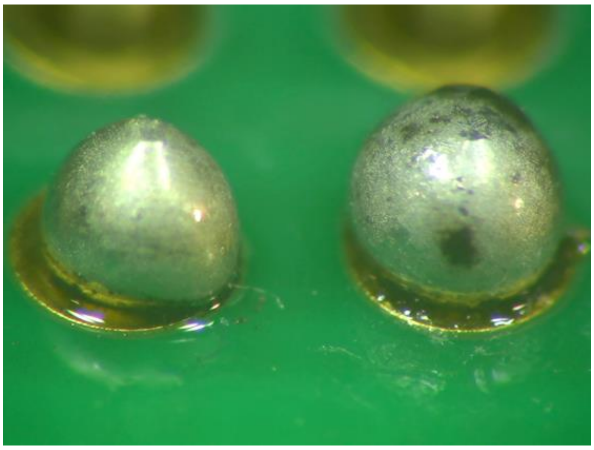
Flux Residues

In most cases soldering with high temperature wire will generate much higher amounts of flux residues on the surface of the board. This is certainly the case on all of the materials used and from practical experience in the field. The cored wire used for this application could be re-developed to reduce the residues for a no clean process. However, the residues and the flux balls that are often seen to spit during the soldering operation may not be considered an issue in some industries.

The residues from different vendors' products were either soft or hard and brittle and very easily displaced but not as soluble in cleaning solutions. The key thing is to select the correct combination of flux and cleaner; cleaning material suppliers have normally conducted many trials on different materials and have great databases of results against time and fluid concentrations.

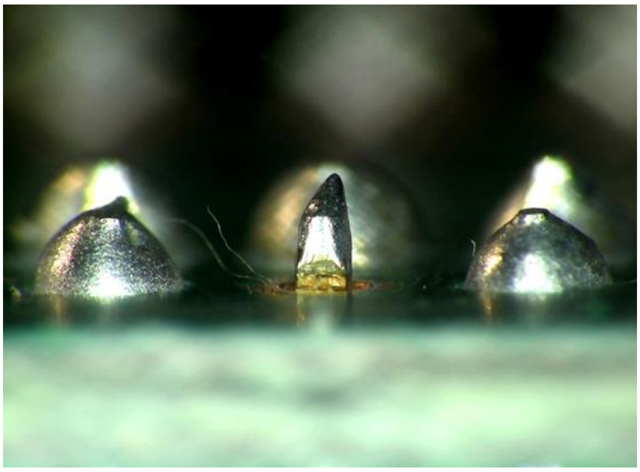
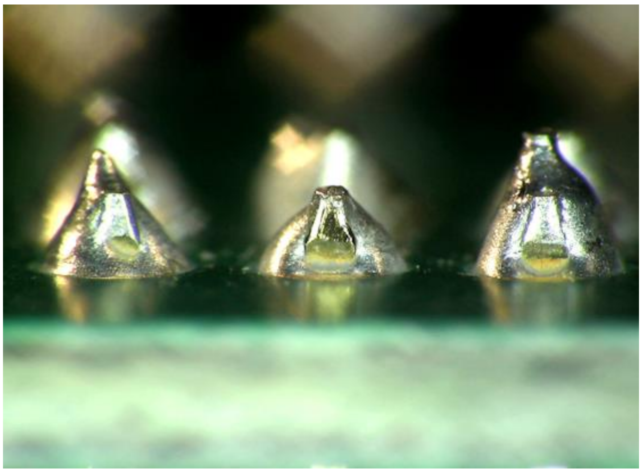
Incomplete Solder Joints




Here we see solder wetting connector pins but not the pads on a PCB. The first assumption is the solderability of the board is poor which could be the case after previously passing through two reflow soldering steps before through hole insertion and robotic soldering. However, a combination of slow wetting of the pads and insufficient time for the heat to penetrate the bulk of the board could result in the same image. The solder was in a liquid state, but it only formed a ball on the pins before solidifying.

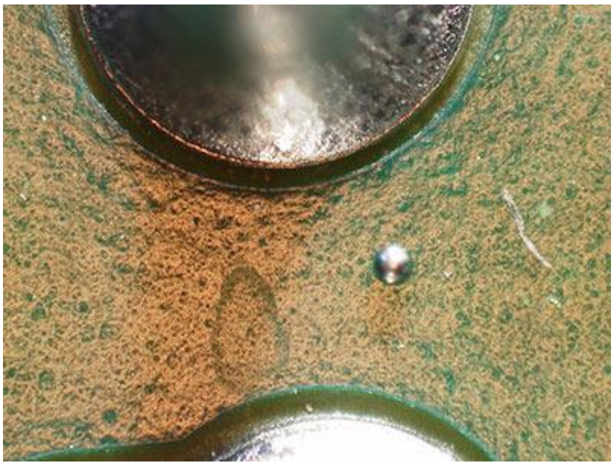
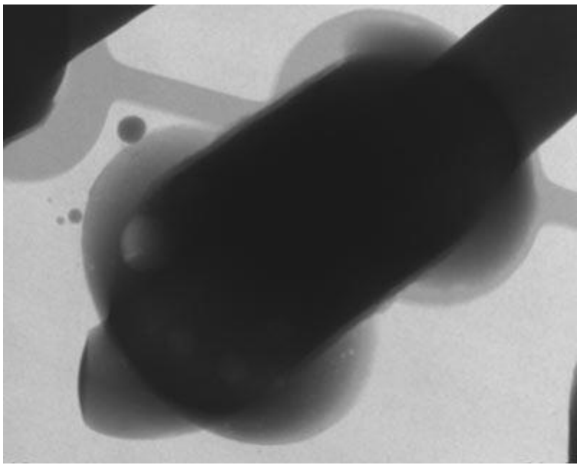
Further examination of the process with video capture should easily establish the cause. Please remember the solderability and changes in surface finish wetting will occur after the first and second reflow operation. This is again impacted by the peak temperature, cool down of each reflow step and total hold times between each process step. All of this is before we start talking about possible board paste washoff and nitrogen levels. It is not always the PCB supplier's fault is it?

Incomplete solder Joints

Example of inconsistent and incomplete joints; in this case the problem was related to the correct hold time and soldering iron temperature setting. This did not allow the correct time for heat to penetrate the board and the solder to capillary through the hole.

Solder Balls

Solder balls, due to spitting, is a reality in both laser and robotic iron soldering based on the pressure placed on engineers to speed up the process. X-ray image shown above illustrates solder balls. In the first case a side by side evaluation needs to be conducted to compare the materials, making sure the wire size and flux core are of the same volume. By testing different wires with the same feed rate and temperatures, wetting performance and degree of spitting can be compared on specifically designed test boards. After initial testing the best wires can be tested with wire scoring or indentation to increase the speed of operation without solder balling.

Reducing spitting by creating exhaust paths for the volatile material in the cored wire as the solder moves from a solid to a liquid can be very beneficial and demonstrated many times.

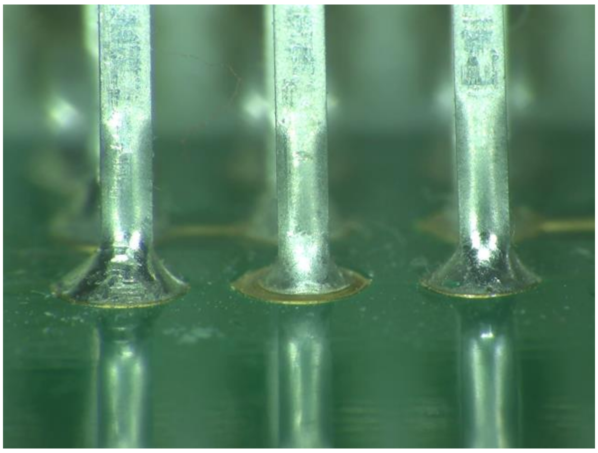
Solder balls can also be seen on selective soldering process. This may be more apparent than with conventional lead-free soldering as the temperatures used are higher. Care needs to be taken with the product design, solder mask selection/specification and the flux material being used to reduce balling even if it is acceptable in some standards. Don't just accept balls because standards allow.

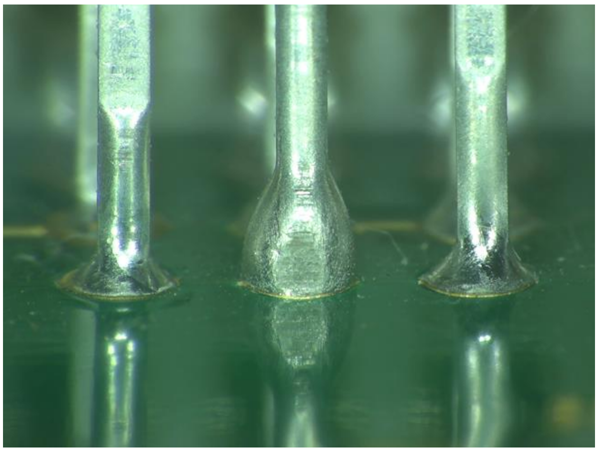
Bubbles in Flux Residues

These bubbles were most commonly seen on laser soldering and probably related to speed of temperature rise and cooling during the soldering operation. If the flux is to be removed by cleaning and, provided the material is still soluble in the chosen cleaning system, there should not be a problem. If the boards were to be conformally coated as part of a no clean process it may be difficult to differentiate flux bubbles with voids or bubbles from the conformal coating process.

The bubbles are an indication of the process parameters used and the flux vehicle which could be developed further for this type of automated process. Inevitably where there is an increasing market for wire for this type of operation there may be more innovation. As the bubbles are in the flux residues it is probably not an issue; however, it is a task for engineering to develop the process with material suppliers to optimise the process.

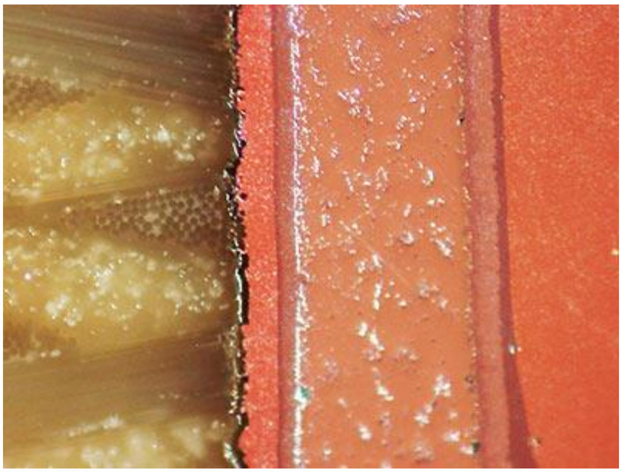
Solder Volume Differences

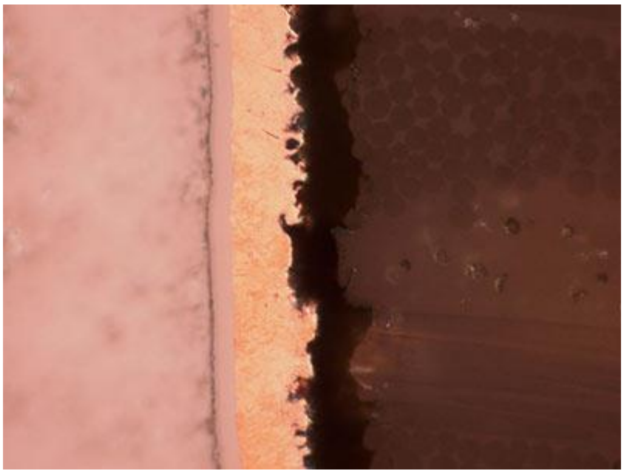




During the assembly trials there were examples of solder volume variation from pin to pin which should be examined further. As the plated through hole size and thermal demand on the test board were fairly consistent this was probably related to solder feed variation on one system or its initial set-up. It is also worth remembering that where solderability is very high connector pins can suck solder away from the main joint suggesting the solder volume available was variable.

Copper Plating Separation

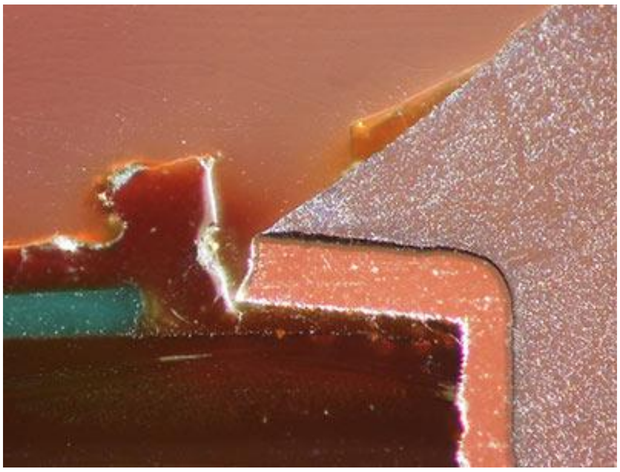


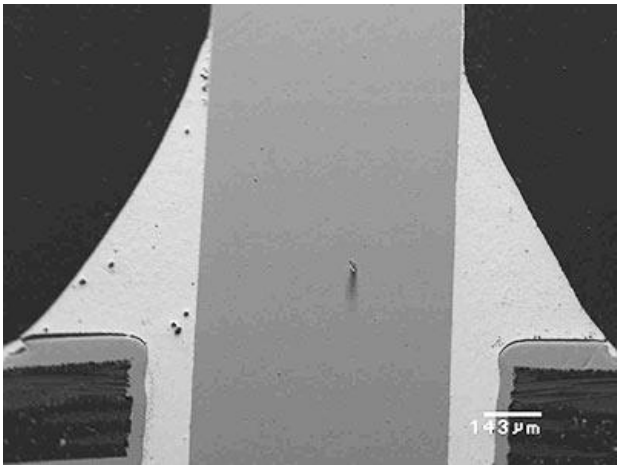


During soldering if excessive heat or duration at elevated temperature is experienced the copper barrel can pull away from the surface of the drilled fiberglass in a plated through hole. This can occur where the adhesion of the copper to the drilled hole is poor during manufacture, but in the cases shown above it was related to the soldering parameters used.

Some separation of the copper barrel can occur during temperature cycling of through hole joints, but it is uncommon to see this cause an electrical failure. Selected areas of copper separation in the barrel can also lead to a reduction in through hole pull-out strength, but again not necessarily lead to failures. In the case of the two samples above further optimisation of the soldering parameters should eliminate the copper separation.

Solder Pad Lifting





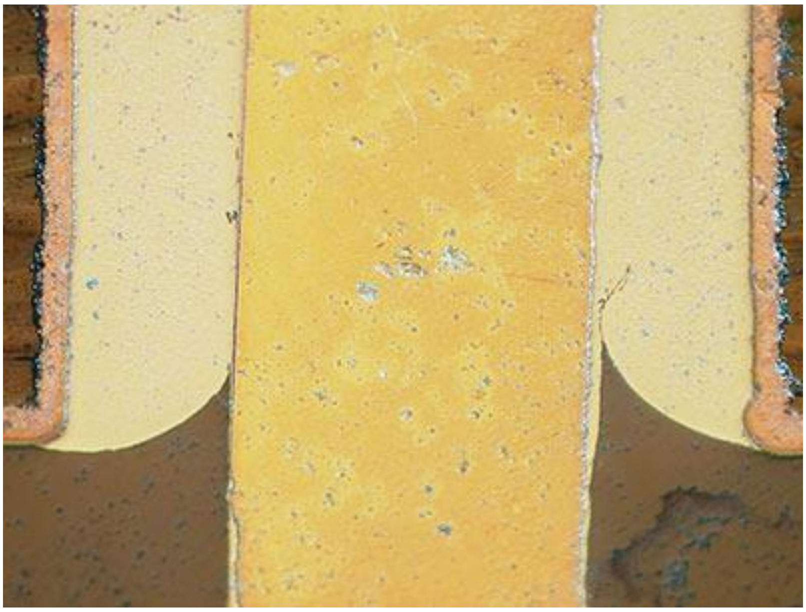
Solder fillet/pad lifting was first experienced in lead-free soldering and has been found in selective, wave and intrusive reflow. It has not, to date, been seen to cause a reliability problem and is not always associated with all through hole joints on a single board.

A solder joint can be perfectly formed but either the edge of the pad is lifted from the surface of the laminate or the solder fillet tip lifts from the edge of the soldered pad. Both points are highlighted in IPC 610F "Acceptability of Electronic Assemblies" for lead-free materials only. It is considered acceptable for class 1, 2 and 3 products. The microsection images above show pad lifting from the surface of the laminate on the left, this type of issue was experienced in the past with tin/lead when soldering thick multilayer boards where the expansion and contraction of the board after soldering left the copper pad lifted from the substrate

The same phenomena have been seen on some samples from our projects. The image on the right shows the solder fillet lifting from the surface of the copper pad. This can be seen on tin/copper and SAC alloys which would be acceptable to IPC. However, where it happens on HMP solder which is lead based, it would not necessarily be acceptable, suggesting the need to change the statements in the IPC 610 which just relate to lead-free exceptions.

One company suggested using solder mask defined pads to reduce pad lifting. It can prevent you seeing the pad lift!!

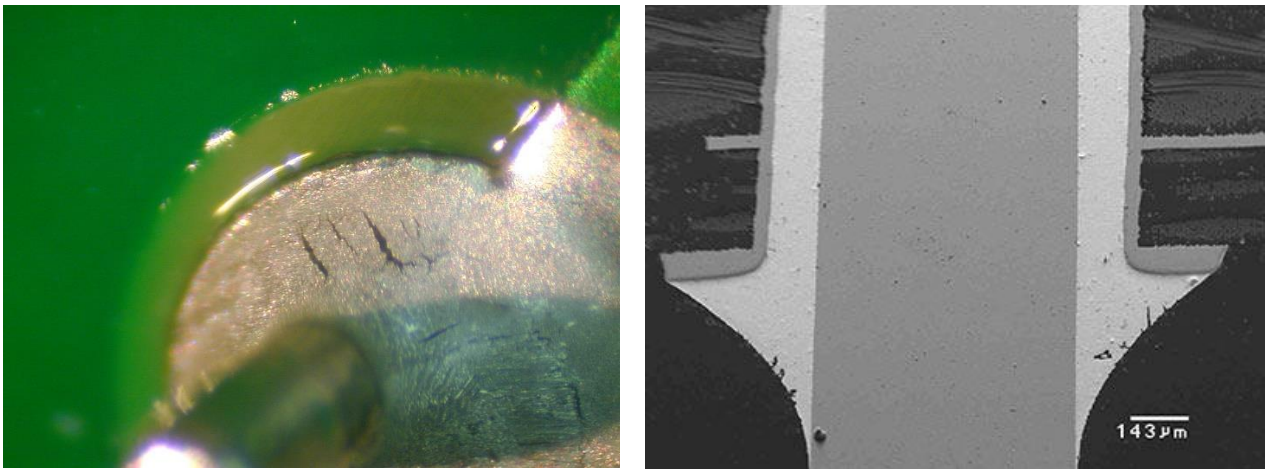
Poor Thermal Balance



During soldering it is important to get the right heat input to an assembly, particularly with laser soldering. Balancing the heat to the board, pin and solder wire is necessary to achieve successful hole fill. Robotic soldering with laser or solder tip needs to be programmed into a computer whereas an operator watches and responds to the soldering conditions and the solder pull through. Some systems have IR sensors and do learn and adjust the soldering parameters based on performance; however, most are dumb. In the case above satisfactory soldering has been achieved to the plated through hole but the pin was not raised to correct temperature prior to feeding the solder into the joint.

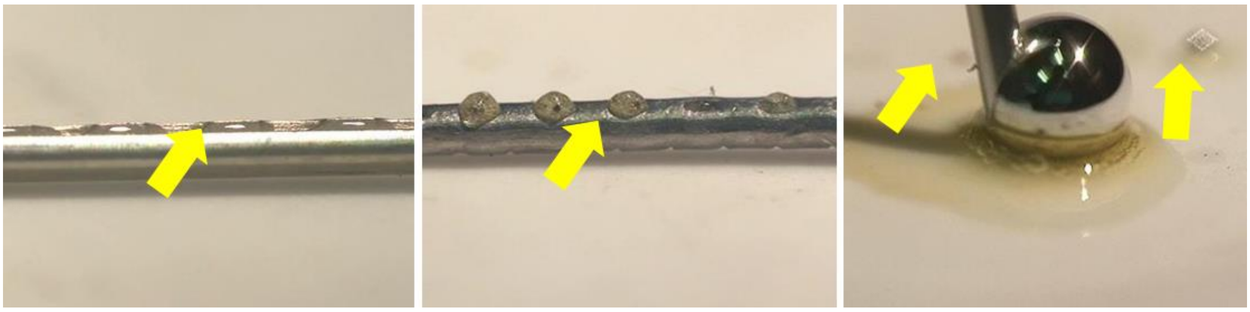
The mass of the pin and possibly the body of the component has caused slow heat transfer and then stopped the solder wetting to the pin. The impact of temperature drop in the solder has also resulted in the solder not flowing on to the opposite side of the board.

Solder Fillet Tearing



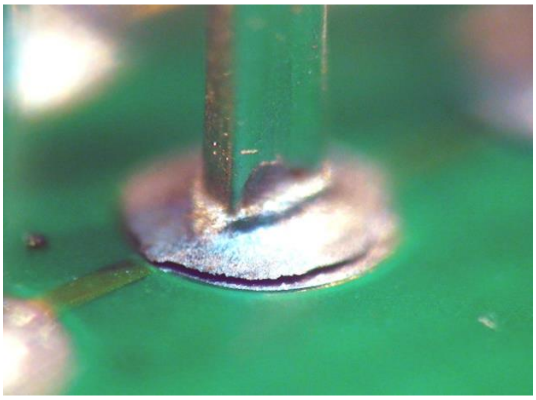
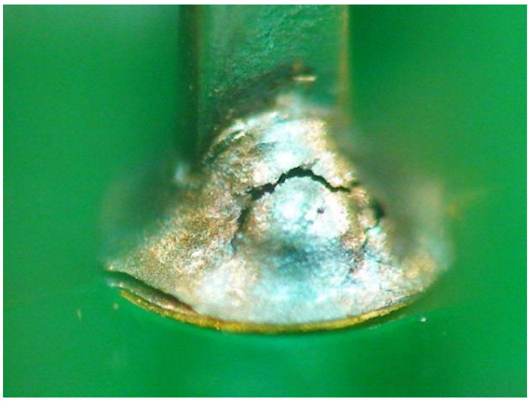
Again this is a process phenomenon we have seen with Tin/Silver/Copper SAC alloys during the introduction of lead-free. It is not normally seen on tin/copper, but more commonly seen on higher silver SAC. During these trials the tearing was seen on the surface of robotic laser and iron soldered joints but did not appear to increase in size after static ageing or during temperature cycling. Tearing is highlighted in IPC 610F "Acceptability of Electronic Assemblies" for lead-free materials only. It is considered acceptable for class 1, 2 and 3. The two microsection images above show the tearing of the solder between the grain boundaries on the surface of the joints. The optical image on the right shows the surface of one joint with the tearing visible in the solder surface.

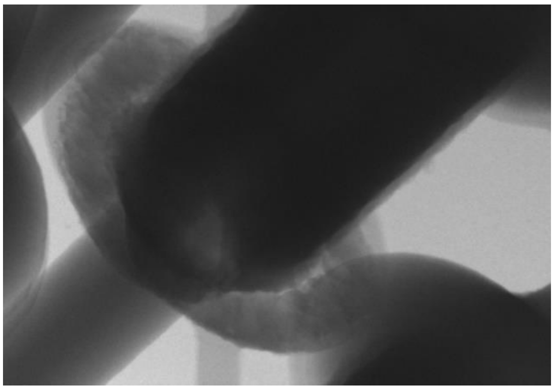
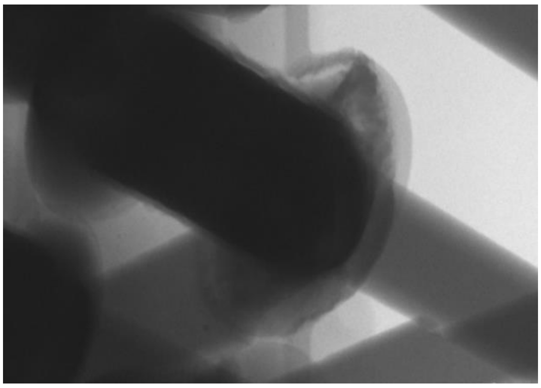
Flux Spitting/Condensation



Flux spitting is really just the same problem as solder balling. Trying to solder at a speed that is too fast for the material or process settings being used. If a company was to develop newer cored wire with less flux volatility engineers/managers would still try to push the process window, hence the best way is to look at the materials and the process. Testing and observing the soldering characteristics of the wire during heating and reflow is very useful to compare products. JapanUNIX has used high speed video to show the difference in wires and the degree of solder balling. Video on their website show the benefits of wire preparation prior to reflow. During our projects we have also been able to control the flux volatile escape from the wire and, with it, spitting of solder balls using the same indent techniques. We have used different video methods to show flux escaping and balling due to the lack of a high-speed camera or a budget to hire one

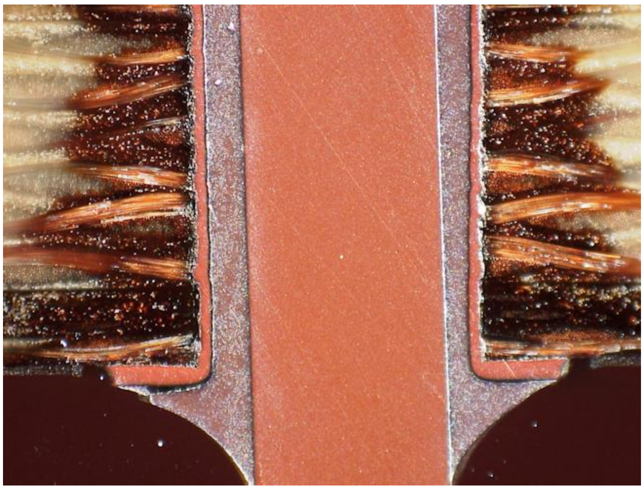
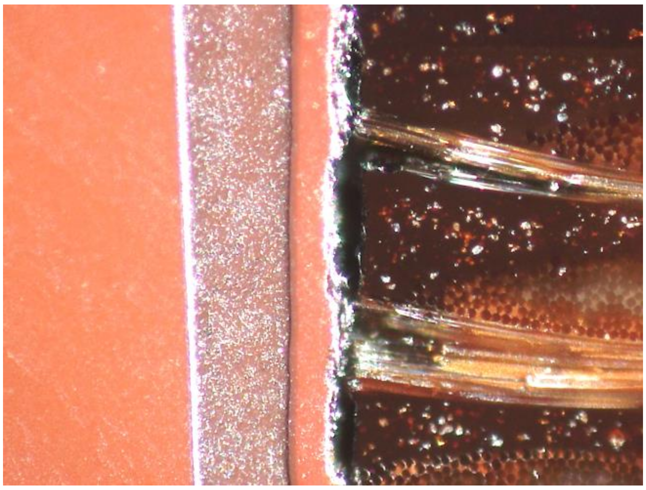
Solder Joint Damage

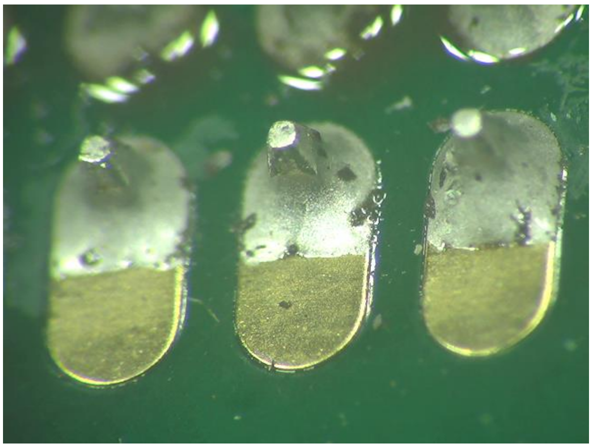
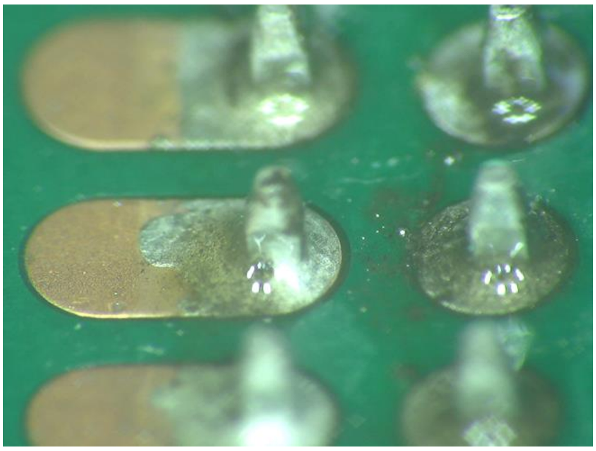
The solder joint damage shown in these examples was apparent after temperature cycling of high temperature solder joints. This type of HMP alloy is distorted during repeated thermal cycling and it was the author’s first experience in testing this type of alloy. The deformation of the joints under test was also noted when examined under X-ray inspection and shown above.

Excess Temperature

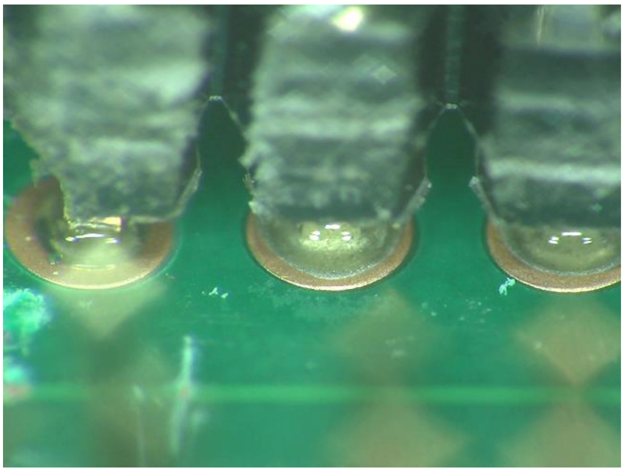
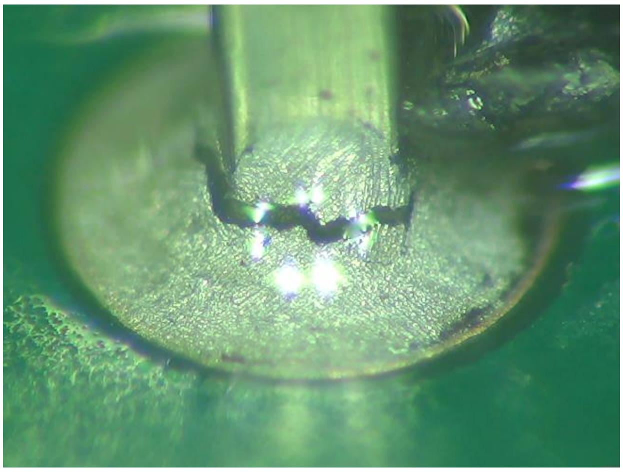
There is a lot of heat damage visible in these microsection views after soldering. The discolouration is plainly visible in the heat affected areas and the laminate. There is evidence of separation of the glass and epoxy. Either the temperature or the dwell time was incorrectly set, the solder joints were satisfactory. There was evidence of heat damage to the laminate around the plated through hole.

Incomplete Pad Coverage

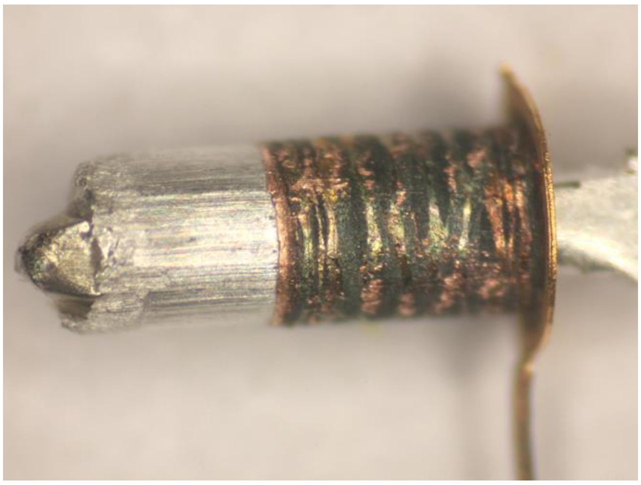
Examples of satisfactory solder joints on extended joints. There is gold exposed on the first image and copper on the OSP board. There should be no requirement to fill all of the pad with solder when satisfactory joints have been produced.

Solder Joint Damage

As has already been covered low temperature joints can be brittle. Care must be taken even on plated through hole joints when mechanically inspecting connections. In this case the connector body was being manually lifted to expose topside solder joints. As shown on the second image one of the joints was cracked by the author. Obviously using X-ray is a perfect way of looking at through hole fill in production. We often don't have all of the equipment available during soldering trials. We can all make simple mistakes.

Solder Joint Pull Strength




Here are two through hole solder joints which have been subjected to pull force measurement after robotic soldering and thermal cycling. In both cases the copper barrel of the copper plating has been pulled out of the printed board. Typically there is a reduction in pull force measurement after thermal cycling; however, there is not specific standard or criteria for this type of test

Incomplete Solder Fill – Pull Back

Solder wire pull back can result in incomplete solder fill of plated through hole joints. I feel that many users may have experienced this in the past but not quite sure of the reason. The three images above were taken from one of our process videos and shows what has happened. The first image shows the tip in contact with the pin and pad and the solder wire being fed to the joint. In this case we were processing a low temperature solder wire. The wire temperature has increased away from the point of contact and then due to surface tension pulled away and separated from the joint area. In the second and third image the tip remains in contact with the pin and pad but there is no solder wire contact to allow through hole fill

Examination of the wire feeding speed, length plus tip temperature specifically needs more consideration for the tin/bismuth solder wire which has a reflow temperature of 138°C

Flux Splatter

Flux spitting from wire and solder paste is not uncommon and normally is related to the material being used. However, the amount of flux spitting can be improved through better optimisation of the process parameters. In many applications flux resides on the surface of the board are not an issue if the process materials have been confirmed as a suitable no clean product with laboratory testing.

However, it is an indication that improvements can be made before volatile spitting becomes random solder balls on the surface of the board become an issue. Many customers running no clean will not accept random solder balls on their product.

Secondary Reflow of Joints

In this example a solder joint is being formed but, due to the close proximity of a join in the back row, the solder fillet is reflowing again which is undesirable. Part of the reason is that the solder joints on the back row are excessive in size, more like bulbus joints. There are a few reasons why this may have occurred and should be considered. The soldering conditions were incorrect on the first pass on the back row leading to excess solder. Close examination may show poor hole fill due to reduced wetting. This is in turn would lead to excess solder on the pad.

Incorrect heating of the joint area may have caused the solder in the hole to solidify again leading to excess volume of solder on the pad. We know from previous trails that the combination of design, cored wire and tip size and process conditions have been able to successfully solder these assemblies.

We are happy to look at robotic soldering defects for customers and provide a view to the root cause of the problem. The key to this is a good quality sharp high resolution photos. The process conditions used, make of machine, tip type and pin and PCB surface finish

Bob Willis Process Defect Video Links

The video links provided below give an insight to defects in real time and hopefully give a better understanding on how certain problems occur and can be overcome in manufacture. They were specifically produced for the 2020 **Robotic Soldering Experience** project cancelled due to Covid. These defects are also part of a much large collection of our **"Defect of the Month" 100+** series that has been running since 2011 and featured by **IPC, NPL, SMTA, SMART Group, Circuits Assembly** magazine and more recently launched with **What's New In Electronics Online** with video and online editorial content

Solder Balling in Robotic Soldering Process

Solder Tearing in Robotic Soldering Process

Excess Flux in Robotic Soldering Process

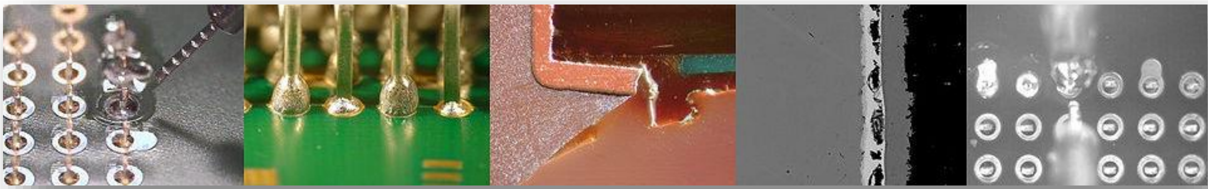
Process Defect Video Links

Flux Bubbles in Robotic Soldering Process

Fillet/Pad Lifting in Robotic Soldering Process

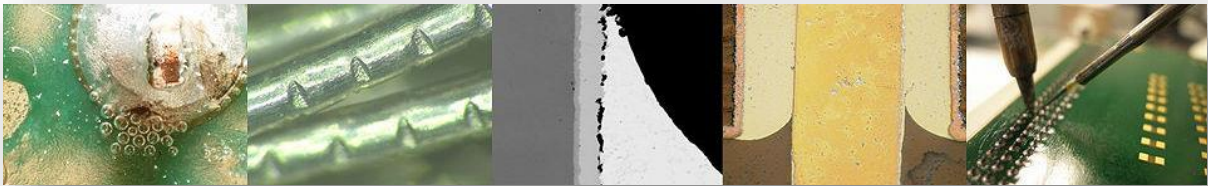
Solder Mask Damage in Robotic Soldering Process

Robotic Soldering Experience 2020 – The Project Overview



Design, component selection and good process control is very important if you are going to transition from skilled production staff to automated irons and laser beams. **"Our Robotic Soldering Experience"** continues on from **The Lead-Free Experience 1-4** concept from early 2000 with practical hands-on demonstrations with high and low temperature materials. Presentations and user experience on building boards and testing materials will be available online and to specially invited engineering guests

A number of equipment suppliers are being invited to demonstrate the use of their system to solder a specific board design featuring multiple connectors. Material suppliers could be able to offer different cored solder wires for low and high temperature soldering and participate in the online presentations and interviews



The organiser has experience in conducted trials on robotic iron and laser soldering with different alloys and produced a report with National Physical Laboratory (NPL). He has also run workshops and online webinars in Europe and the US and events at major trade shows on robotic soldering

Proposed topics covered in online presentations

PCB design rules
Programming robotic soldering systems
PCB tooling
Quality control & testing solder joints
Evaluation of cored solder wire for iron and laser soldering
Use of nitrogen to aid soldering
Extraction for robotic systems
Health and safety considerations
Soldering standards

Suppliers supporting practical trials by the author

Henkel Ltd. Headquarters
Technologies House, Wood Lane End
Hemel Hempstead
Hertfordshire
HP2 4RQ, UK
Tel: 01442-278000

Japan UNIX
2-21-25, Akasaka,
Minato city
Tokyo, Japan
Tel: 81-335880551

mta Automation ag
Bernstrasse 5
CH-3238 Gals
Switzerland
Tel: 41 32 338 82 82

Harting Connectors
51 Caswell Rd
Northampton
NN4 7PW, UK
Tel : 01604 827500

Merlin Circuit Technology Ltd
Hawarden Industrial Park
Manor Lane, Deeside
Flintshire, North Wales
CH5 3QZ, UK
Tel: 01244 520510

PCB Laminate suppliers to Merlin Circuits for the NPL project included:
ISOLA Group
Lamar - Nelco
Ventec Europe
Park Electrochemical Corp

Apollo Seiko LTD
3969 Lemon Creek Road
Bridgman, MI 49106
USA
Tel: 269-465-3400

Apollo Seiko
Kaisertech Group
Unit 12, M3 Trade Park,
Manor Way, Eastleigh,
Hampshire, SO50 9YA, UK
Tel: 02380 650065

Promation INC. USA
North American Headquarters
9900 58TH PL. STE.#100
KENOSHA, WI 53144
Tel: 262.764.4832
www.promationusa.com

DKL Metals Ltd
Avontoun Works
Linlithgow
EH49 6QD, UK
Tel: 01506 847710

Seica SpA
Via Kennedy 24
10019 Strambino – TO
Italy
Tel: 0125 636811

Wolf Produktionssysteme GmbH
Robert-Burkle-Strasse 6
72250 Freudenstadt
Germany
Tel: 074 41 89920

Altus Group UK
Unit D, Crescent Trade Park
Moons Moat Drive
Redditch
B98 9DZ
Tel: 44 01386 791074
www.altusgroup.co.uk

NIHON SUPERIOR CO LTD
Head Office NS Bldg., 1-16-15 Esaka-Cho
Suita City, Osaka, 564-0063 Japan
Tel: +81- (0) 6-6380-1121
www.en.nihonsuperior.co.jp

Almit GmbH
Unterer Hammer 3
DE 64720 Michelstadt
Germany
Tel:+49 (0) 6061 96925 0
www.almit.de

Author's Profile

Bob Willis is semi-retired but originally operated a training and consultancy business based in England for over 35 years. Bob was a member of the SMART Technical Committee before its transition to SMTA Europe. Although a specialist for companies implementing Surface Mount and Lead-Free Technology Mr Willis provides training and consultancy in most areas of electronic manufacture. He ran theory and hands on workshops and monthly online webinars worldwide. He has worked with the GEC Technical Directorate as Surface Mount Co-Coordinator for both the Marconi and GEC group of companies and prior to that he was Senior Process Control Engineer with Marconi Communication Systems, where he had worked since his apprenticeship. Following his time with GEC he became Technical Director of an electronics contract manufacturing company where he formed a successful training and consultancy division.

As a process engineer with GEC Marconi, he was involved in all aspects of electronic production and assembly involved in setting up production processes and evaluating materials; this also involved obtaining company approval on a wide range of Marconi's processes and products including printed circuit board manufacture. During the period with Marconi, experience was gained in methods and equipment for environmental testing of components, printed boards and assemblies with an interest developed in many areas of defect analysis. Over the last 35 years he has been involved in all aspects of surface mounted assembly, both at production and quality level and during that time has been involved in training staff and other engineers in many aspects of modern production.

Bob Willis has been involved with the introduction and implementation of lead-free process technology during the introduction of RoHS in Europe. He received A SOLDERTEC/Tin Technology Global Lead-Free Award for his contribution to the industry, helping implementation of the technology. Bob has been a contributor to Global SMT magazine during its launch. He was responsible for co-ordination and introduction of the First series of hands-on lead-free training workshops in Europe for Cookson Electronics during 1999-2002. These events were run in France, Italy and the UK and involved lead-free theory, hands-on paste printing, reflow, wave and hand soldering exercises. Each non-commercial event provided the first opportunity for engineers to get first-hand experience in the use of lead-free production processes and money raised from these events was presented to local charity.

He co-ordinated the SMART Group Lead-Free Hands on Experience at Nepcon Electronics 2003. This gave the opportunity for over 150 engineers to process four different PCB solder finishes, with two different lead-free pastes through convection and vapour phase reflow. He also organised Lead-Free Experience 2, 3 + 4 in 2004-2006. He has run training workshops with research groups like ITTF, SINTEF, NPL & IVF in Europe. Bob has organised and run three lead-free production lines at international exhibitions Productronica, Hanover Fair and Nepcon Electronics in Germany and England to provide an insight to the practical use of lead-free soldering on BGA Ball Grid Array, CSP Chip Scale Package, 0210 chip and through hole intrusive reflow connectors. This resulted in many technical papers being published in Germany, USA and the United Kingdom. Bob also defined the process and assisted with the set-up and running of the first Simultaneous Double Sided Lead-Free Reflow process using tin/silver/copper for reflow of through hole and surface mount products.

Bob also had the pleasure of contributing a small section to the first Lead-Free Soldering text book “Environment - Friendly Electronics: Lead-Free Technology” written by Jennie Hwang in 2001. The section provided examples of the type of lead-free defects companies may experience in production. Further illustrations of lead-free joints have been featured in her most recent publication “Implementing Lead-Free Electronics” 2005. He has helped produce booklets on X-ray inspection and lead-free defects with DAGE Industries, Balver Zinn, SMART Group and SMTA

Mr Willis led the SMART Group DTI Lead-Free Mission to Japan and with this team produced a report and organised several conference presentations on their findings. The mission was supported by the DTI and visited many companies in Japan as well as presenting a seminar in Tokyo at the British Embassy to over 60 technologists and senior managers of many of Japan’s leading producers. Bob was responsible for the Lead-Free Assembly & Soldering "CookBook" CD-ROM concept in 1999, the world’s first interactive training resource. He implemented the concept and produced the interactive CD in partnership with the National Physical Laboratory (NPL), drawing on the many resources available in the industry including valuable work from NPL, ITRI and the DTI. This incorporated many interviews with leading engineers involved with lead-free research and process introduction; the CD-ROM reached its 3rd edition during and after the introduction of lead-free in 2006

Bob has travelled in the United States, Japan, China, New Zealand, Australia and the Far East looking at areas of electronics and lecturing on electronic assembly. Mr. Willis 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 was course leader for Reflow and Wave Soldering Workshops in the United Kingdom for Speedline Technology. Mr Willis was a IEE Registered Trainer and has been responsible for training courses run by the PCIF originally one of Europe's largest printed circuit associations. Bob has conducted workshops with all the major organisations and exhibition organisers World Wide and is known for being an entertaining presenter and the only presenter to use unique process video clips during his workshops to demonstrate each point made. Bob has written three eBooks which are free to download on line, Design & Assembly with Pin In Hole Intrusive Reflow, Package On Package Design, Assembly and Inspection and this his third title

Mr. Willis was Chairman of the SMART Group, European Surface Mount Trade Association from 1990-94 and was elected Honorary Life President and held the position of SMART Group Technical Director, he has worked on BSI & IPC Standards Working Parties. He is a Fellow of the Institute Circuit Technology, a NVQ Assessor, previous member of the Institute of Quality Assurance and Society of Environmental Test Engineers. Bob Willis written regular features for AMT Ireland, Asian Electronics Engineer, Global SMT and Circuits Assembly the US magazine. He also is responsible for writing each of the SMART Group Charity Technology reports, which were sold in Europe and America by the SMTA to raise money for worthy causes. Bob ran the SMART Group PPM Monitoring Project in the United Kingdom supported by the Department of Trade and Industry. He was coordinator of the LEADOUT Project for SMART Group. LEADOUT was one of the largest EU funded projects, he coordinated European funded projects TestPEP, uBGA and ChipCheck with the SMART Group team of engineers

In September 2015 Bob was voted the Best Speaker at SMTA International Conference in Chicago, USA. Part of the prize money was donated to the SMTA Charles Hutchins Fund. More recently he was presented with a special award for support to industry in Europe. Bob has created more process defect guides originally for SMART Group/SMTA Europe which are free to download. They include **Guide to Conformal Coating Defect**, **Guide PCB Surface Finish Defects**, **QFN/LGA Process Defect Guide** plus **Cleaning and Contamination** with Global SMT. More recently a **Guide to Solder Paste Printing & Defects** with WNIE

Bob closed his business in 2022 and is semi-retired spending more time on holidays, diving, playing football, flying drones and relaxing. He still does a little training and consultancy to keep his hand in the game. This book is his final process guide with all of the money raised going to charity

Bobwillis.co.uk

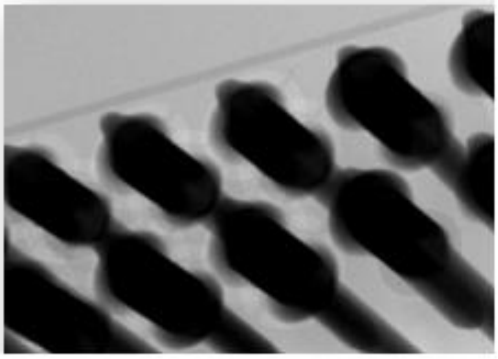
Low Temperature Solder Joint Inspection Criteria

Satisfactory

Solder joints produced with Tin/Bismuth/Silver (SnBiAg) with a robotic soldering process at 240°C. Although the solder does not cover all of the pad surface, this is perfectly acceptable solder joints

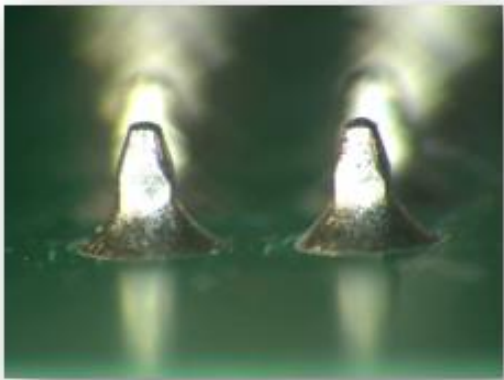
Satisfactory

Satisfactory joints formed with SnBiAg solder alloy during robotic soldering. X-ray inspection was used to show the degree of the through hole fill on a copper OSP finish printed board assembly



Satisfactory

SnBiAg solder joint produced with a robotic iron soldering process at 240°C. The solder has formed a perfect solder joint with a concave fillet formation



Produced by: Bob Willis @ bob@bobwillis.co.uk www.bobwillis.co.uk

Our inspection wall chart set are all available FREE to download via the smta.org website as part of training media package sponsorship

PCB Surface Finishes & Solderability Standards & Text Books

IPC-4552 Specification for Electroless Nickel/Immersion Gold (ENIG) Plating for Printed Circuit Boards
IPC-4553 Specification for Immersion Silver Plating for Printed Boards
IPC-4554 Specification for Immersion Tin Plating for Printed Circuit Boards
IPC-4555 Specification for Organic Solderability Preservative (OSP) for Printed Circuit Boards (Committee Draft)
IPC-4556 Electroless Nickel/Electroless Palladium/Immersion Gold (ENEPIG) Plating for Printed Circuit Boards

IPC J-STD 003 Solderability Tests for Printed Boards
IPC 600 Acceptability of Printed Circuit Boards
IPC 610 Acceptability of Electronic Assemblies

In addition, specifications for solder levelled & OSP boards are still required for the industry. IPC have drafted documents but have never passed out of the committee stages. The following are text books or free guides on printed board manufacture, design and assembly which include details on the use of area array packaging

Pin in Hole Intrusive Reflow Desing, Assembly & Defect Guide
Bob Willis - (Free Download)

Package On Package Assembly Inspection & Quality Control
Bob Willis - (Free Download)

Solder Paste Print Inspection & Defect Guide
Bob Willis - (Free Download)

Lead-Free Defect Guide 3
Bob Willis - SMART Group (Free Download)

Conformal Coating Inspection & Defect Guide
Bob Willis (Free Download)

QFN LGA Assembly Inspection & Defect Guide
Bob Willis - SMART Group (Free Download)

PCB Surface Finishes Inspection & Defect Guide
Bob Willis - SMART Group (Free Download)

Cleaning & Contamination Defect Guide
Bob Willis - Global SMT (Free Download)

Printed Circuit Handbook (7th Edition & the PCB Manufacturing Bible)
Clyde Coombs, Jr. - McGraw Hill

Flexible Circuit Technology 4th Edition
Joe Fjelstad – BRP (Free Download)

Comprehensive Guide to Design, Manufacture of Printed Board Assemblies
Bill MacLeod Ross - Electrochemical Publications (Finishing Publications Limited)

SMT for PC Board Design (2nd - 3rd Edition)
James Hollomon - Prompt Publications

Practical Guide to Soldering PCBs with High Temperature Solder Alloys
Good Practice Guide - Chris Hunt & Bob Willis NPL (Free Download www.npl.co.uk)

A range of text books are available to purchase from the SMTA and can be seen on line at www.smta.org SMTA also hosts Bob's webinars, photo library, inspection posters FREE to download. The collection of material is available for use by engineers and was kindly sponsored by **Humiseal UK** in 2023 for Prostate Cancer UK

Online Charity Webinars Robotic Soldering Inspection & Defects Solutions

Bob Willis – Bobwillis.co.uk

During the session Bob Willis will highlight the benefits and show the process used in both contact and laser soldering to exceed the requirements of IPC 610 criteria. He will illustrate the common problems and how to overcome them with good practice. In addition, Bob will show some of the reliability test results with standard lead-free, Low Temperature Solder LTS and higher temperature alloys. If you have any specific questions or existing production issues you would like addresses in the session, you would be able to send questions in advance

Delegates will also benefit from a copy of the report on High Temperature Robotic Soldering produced with Dr Chris Hunt from NPL and a recent colour inspection guide to inspection standard for Robotic Soldering with different solder alloys, also available from SMTA

Our session will also showcase the experience gained with **"Robotic Soldering Experience"** a hands-on production feature bringing together the most well-known companies in robotic soldering their materials and experience which was created by the presenter

Topics covered

Why use robotic soldering
Introduction to contact and Laser soldering
Design, PCB and component requirements
Solder wire considerations to aid soldering
Process performance during soldering
Benefits in using nitrogen
Considerations for planned maintenance
Common issues with materials
Reliability testing and inspection results
Process defects and common causes

During and after the session there is time for Q&A with ample time for all delegate questions to be answered. However, if a delegate has a process example, they would like covered in the event it will need to be provided in advance of the session

Who should attend?

This technical session is ideally suited to design, production and quality engineers looking at future technology and maintaining a company technology roadmap. It’s vital to subcontractors to be up to date with new technology like robotic soldering and its possible implementation along with material and equipment requirements for future customers

The following are gratefully acknowledged, for sponsoring this guide and raising money for charity

Are you at risk of prostate cancer?

1 in 8

In the UK, about 1 in 8 men will get prostate cancer at some point in their lives.

Prostate cancer is the most common cancer in men in the UK.

Over 50 years old

Prostate cancer mainly affects men over 50 and **your risk increases with age**.
The average age for men to be diagnosed with prostate cancer is between 65 and 69 years.

Family history and genes

You are two and a half times more likely to get prostate cancer if your **father** or **brother** has been diagnosed with it, compared to a man with no family history of prostate cancer.

Your risk of getting prostate cancer is higher if your **mother** or **sister** has had breast cancer.

Ethnicity

Black men are more likely to get prostate cancer than other men. In the UK, about 1 in 4 black men will get prostate cancer at some point in their lives. If you're black, you may be more likely to get prostate cancer if you're aged 45 or over.

PROSTATE CANCER UK

Speak to our Specialist Nurses
0800 074 8383* | prostatecanceruk.org

Check your risk in 30 seconds:
prostatecanceruk.org/risk-checker

* Call our Specialist Nurses from Monday to Friday 9am - 5pm, Wednesday 10am - 5pm. Calls are recorded for training purposes only. Confidentiality is maintained between callers and Prostate Cancer UK.