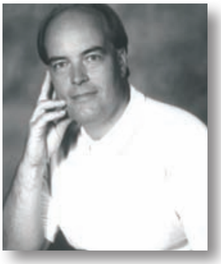
PIHR TECHNOLOGY

BY BOB WILLIS

About the Author



Bob Willis currently operates a training and consultancy business based in England and has created one of the largest collections of interactive training material in the industry. Bob is known as a specialist for companies implementing lead-free manufacture, but has provided worldwide consultancy in most areas of electronic manufacture over the last 25 years. He has worked in a wide variety of areas, including contract assembly, printed board manufacture and environmental test facilities. Recently Bob has been providing regular online training sessions and custom webinars for the industry worldwide.

His activity in international companies and conferences has earned him the SOLDERTEC/Tin Technology Global Lead-Free Award for his contribution to the industry. He has also been presented with the SMTA International Leadership Award and IPC Committee Award for contribution to their standards activity, as well as Best Paper awards from the Institute of Circuit Technology (ICT).

He has worked with the GEC Technical Directorate as Surface Mount Coordinator for both the Marconi Communications Systems and GEC group of companies. Prior to GEC, he was Senior Process Control Engineer with Marconi. Following his time with GEC, he became Technical Director of an electronics contract manufacturing company, where he formed a successful training and consultancy division. Over the years, Bob has been Chairman and Technical Director of the SMART Group and holds the title of Honorary Life Vice President for his contributions to the group since its inception.

Bob welcomes your feedback on the book. Please feel free to contact him if you have any questions or suggestions.

Bob Willis
Bobwillisonline.com
bob@bobwillis.co.uk
Tel 00 44 1245 351502

PIHR
TECHNOLOGY

by Bob Willis

PIHR Technology

BR Publishing, Inc.
PO Box 50
Seaside, OR 97138 USA

Publisher of the I-Connect007 family of online and digital magazines serving the Printed Circuit Board (PCB), Printed Circuit Board Assembly (PCBA), Electronics Manufacturing Services (EMS), Original Design Manufacturing (ODM) and PCB Design Industries.

Book Design: Ron Meogrossi
Cover Design: Bryson Matties
Edited by: Diane Neer

Contents

ADVERTISERS

Introduction

Surface Mount Technology (SMT) is now part of mainstream electronic assembly with virtually all market sectors benefiting from the use of SMT. One feature that has always proved a problem to design and manufacturing engineers is existing through-hole parts where no direct equivalent surface mount part is available. It is possible to hand solder conventional through-hole components after the surface mount assembly when reflow soldering operations are complete. However, this is time consuming, expensive and may leave more flux residues on the surface of the joints. Many through-hole parts are used as direct contact test points using bed of nail or flying probe test equipment. Residues can quickly clog test pins unless regular preventative maintenance is conducted.

SELECTIVE PALLETS

The use of selective pallets is an option for high-volume soldering of through-hole parts during wave soldering. On a double-sided surface mount product, the SMT components are protected during wave soldering by the pallet material, with only through-hole termination and pads exposed to the wave soldering process. In previous operations, the surface mount components on both sides of the board have been reflowed to make the interconnection. The selective pallet technique works, but puts many constraints on the design engineer for correct component layout and the special design of the pallet and on the process engineer for optimising the wave soldering parameters. Whenever selective pallets are used on wave soldering, there is a move away from ideal soldering conditions.

SEMI- AND FULLY-AUTOMATED SOLDERING

A wide range of automatic selective soldering equipment is available to either semi- or fully-automate soldering of through-hole leads. This does, of course, require capital expenditure on equipment and extensive engineering work on programming the systems. This option has become very popular for tin/lead and lead-free soldering, particularly with more supplier options making the market more competitive.

SOLDER PRE-FORMS

Solder pre-forms are another technique that can be employed with reflow soldering. Pre-formed alloys are available in standard shaped outlines from many suppliers in different alloys. In the early days of large telecom backplanes, the pre-formed solder donuts were used in combination with vapour phase reflow soldering. One of the limitations was the need for a liquid flux application to aid the reflow process. The backplane soldering process was not very elegant and was expensive, due to drag-out of the soldering liquid. Some companies employed the wave soldering process even with the long wire wrap spills.

PIN IN HOLE REFLOW

One method of soldering all surface mount and through components in a single operation is Pin in Hole Reflow (PIHR), or intrusive reflow soldering, which has steadily gained importance. The first presentation on the subject was by Racal at Nepcon West, USA in the late 1980s. Unfortunately, the paper was never published. Over the years, however, there have been many good papers on the subject, expanding the knowledge and practical experience in what is a very simple process to implement at minimum cost.

There are many other names given to PIHR throughout the world, including:

- Pin in Paste
- Multi-Spot Soldering
- Selective Reflow
- Through-Hole Reflow

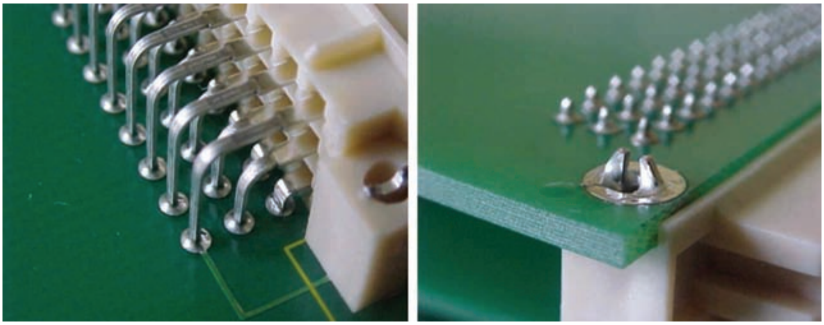


Figure I-1: Typical solder joints produced on opposite sides of a 1.6mm board produced by PIHR soldering on through-hole connectors.

In the PIHR process, all through-hole locations have solder paste applied at the same time as surface mount printing and prior to mounting the through-hole component terminations onto the plated through-hole board. This is generally a final assembly stage after all other surface mount parts have been placed. It may be done first to avoid any unnecessary movement of the surface mount parts due to placement force, board flex or vibration. When complete, the assembly is passed through a reflow process, which today would be convection or vapour phase. Other processes have been used and will be discussed in a later chapter.

Single-Sided Reflow Assembly

The basic steps of single-sided reflow assembly are outlined in Figure I-2.

Print solder paste onto surface mount and through-hole pad locations
Place surface mount components
Insert through-hole components
Pass assembly through the reflow process
Inspect the solder joints after reflow

Figure I-2: Single-Sided Reflow Assembly.

Double-Sided Reflow Assembly

The basic steps of double-sided reflow assembly are outlined in Figure I-3.

Print solder paste onto first side surface mount pads
Place surface mount components
Pass assembly through reflow oven
Turn board assembly over
Print solder paste onto surface mount and through-hole pad locations
Place surface mount components
Insert through-hole components
Pass assembly through the reflow process
Inspect the solder joints after reflow

Figure I-3: Double-Sided Reflow Assembly.

Depending on the equipment and the type of components, it may be beneficial to insert the through-hole parts prior to surface mount placement. Insertion of parts can cause movement of previously placed surface mount parts.

Simultaneous Double-Sided Reflow Soldering

A final option to the process engineer for the assembly of double-sided boards with through-hole packages is Simultaneous Double-Sided Reflow Soldering (SDSRS), which has been used on a number of production lines with tin/lead and lead-free solder paste alloys. It benefits from the use of only one reflow step, which reduces cost, production time and the impact of multiple heating operations on board assemblies. The process steps are outlined in Figure I-4.

Details on the use of SDSRS are covered in a report and CD-ROM that are listed in Chapter 10 - Suggested PIHR Resources.

Print solder face for first side surface mount parts
Dispense or stencil print UV curing adhesive
Place surface mount components
UV cure adhesive
Turn the board assembly over
Print solder paste onto surface mount and through-hole pad locations
Place surface mount components
Insert through-hole components
Pass assembly through reflow process, simultaneously reflowing all joints on both sides of the board Inspect the solder joints after reflow

Figure I-4: Simultaneous Double-Sided Reflow Assembly.

CONSIDERATIONS FOR PIHR

The following section outlines some of the advantages and disadvantages of PIHR assembly and some of the challenges that should be considered during its introduction.

PIHR Advantages

Eliminates Manual/Wave Soldering

Hand, wave and selective soldering are expensive, particularly when you consider the cost of the solder alloys. Any high-temperature soldering process with different consumable materials has potential health and safety issues that need to be considered along with the energy costs associated with these processes.

Compatible with Current Assembly Process

Running a reflow soldering process allows experimentation with through-hole reflow using a sample board and some connectors assembled manually. It is also possible to do simple trials by making a punch stencil for the through-hole terminations.

Minimum Cost of Implementation

Purchasing a new stencil to run some intrusive trials is easy and will allow experience to be gained in your production process. The through-hole apertures can be blocked again for normal production while other trials are conducted on component compatibility.

Potential for Increased Automation

Manually assembled connectors can be automated with most placement systems today without investing in odd form assembly equipment. Many connector suppliers have experience with automatic placement and placement equipment has been introduced by many suppliers.

Reduction in Floor Space

Eliminating selective or wave soldering equipment can free up floor space. Considering the potential elimination of free-standing extraction, conveyors and storage for consumable materials brings floor space requirements back to the core surface mount equipment, printer, placement and reflow system.

Reduction in Capital Equipment

When setting up a new production facility, the core surface mount equipment required for single- or double-sided reflow is a printer, placement and reflow. Provided that the design and the through-hole components are compatible with automatic placement and reflow, this will be the core requirement for small and medium volume assembly.

Potentially Only One Heating Process

On single-sided products, there is just one heating process, which reduces energy costs, but not the solderability on the PCB solder finish. If SDSRS were to be used on a double-sided surface mount design, it would be just one reflow soldering cycle for the complete assembly, which reduces costs and speeds up manufacture.

Use No-Clean Technology

Provided that the correct solder paste products are used, PIHR can provide an assembly with minimum residues left on the surface of the joints. Since the assembly and fluxes have been through the complete temperature cycle, there is a reduction in the possibility of contamination-related corrosion failures.

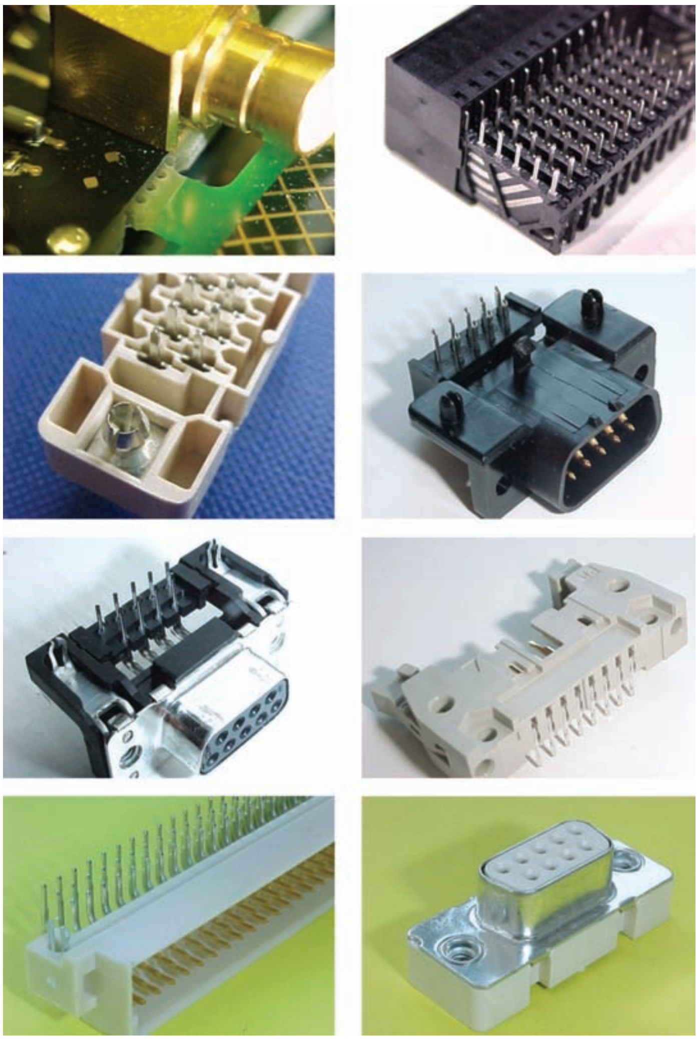


Figure I-5: Samples of different components that could be assembled using PIHR.

PIHR Disadvantages

Through-Hole Components Must be Temperature-Compatible

Components exposed to reflow must not be affected by the temperature used during soldering. The temperature requirement varies depending on if the solder alloy is tin/lead or lead-free, but a good guide is a rating of 260°C. Changing components to a higher rating may add to the cost, but it is possible to use lower temperature alloys on lower specification components. Often components selected for PIHR assembly have a higher mass, so correct profiling is important and should not be dismissed by the process engineering department.

Component Lead Lengths Need to be Defined

The length of the component lead protruding from the base of the board should be specified since it can impact the assembly process. Ideally, the lead protrusion should be as small as possible since it does not impact reliability. Excessive lead length can waste paste designed to fill the plated through-hole.

New Solder Paste Stencils Required

The stencil requirements are defined by the fine pitch requirements on the board and the volume needed for the through-hole parts. Some stencil suppliers may be unfamiliar with pin in paste, so make sure they understand that through-holes are sometimes printed over. There have been occasions when specifically designed apertures have been removed from the design file by the stencil producer.

Correct PCB Specification

It is poor practice not to specify the solder resist, mask type or solder finish, or to randomly change PCB manufacturers without considering material compatibility. Some solder masks do not have the same surface tension, which can lead to poor reflow of the solder paste across the surface of the mask, resulting in solder balling.

Changing the surface finish may change the solder joint appearance to customers, causing unnecessary concerns.

Modification of Inspection Criteria

It is easy to meet the minimum requirements of IPC 610 level 3 for hole fill with PIHR, but through-hole reflow can look different without impacting reliability. It is important that these differences are discussed with the onsite inspection department and external customers to eliminate unnecessary rejection or joint rework.

Customer Acceptance

Changing the soldering process from wave, hand or selective soldering has positive benefits, but customers must be aware of the proposed changes prior to running a product build, otherwise it will lead to long discussions. If you change a standard process, your inspection department should be provided with any inspection criteria and details should be provided to your final customers for approval.

Possible Manual Insertion

Manual insertion can be seen as a negative to many engineers, but in realistic terms, there was no option in the early years of PIHR. It was not economical to use odd form assembly equipment for small companies or it was difficult to justify the costs. SMT placement machine capability automation is now open to more users, not just high-volume users.

Other Considerations for PIHR

There are a few alternatives to adding solder paste to the through-holes for intrusive reflow soldering. The method of paste application for through-holes can directly affect the volume of solder available to form the solder joint. Each of the basic methods will be described later in this book. The methods used to adapt the traditional stencil printing process must not, however, impact the assembly yield for any other fine pitch parts on the board. In reality, a through-hole solder joint is far more robust and reliable than any surface mount joint and 100% through-hole fill and positive fillets on both sides of the board are not necessary. **PIHR**

“Although the IPC specification IPC 9501 is based on surface mount component compatibility, it provides a good reference for a testing methodology.”

CHAPTER 1

Pin In Hole Reflow PCB Design Guidelines

PIHR DESIGN GUIDELINES

Compatibility of components with reflow temperatures and solder volume are the major factors affecting the design engineer. Reference should be made to the component specifications in Chapter 2 - PIHR Component Guidelines, the solder alloy to be used, the reflow process type and the operating environment of the final product. Although the IPC specification IPC 9501 is based on surface mount component compatibility, it provides a good reference for a testing methodology. In addition, the following IPC documents are worth reviewing for additional information:

- IPC C 406 - Design and Application Guidelines for Surface Mount Connectors
- IPC SM 786 - Procedures for Characterising and Handling of Moisture/Reflow Sensitive ICs
- IPC/JEDEC 020 - Moisture/Reflow Sensitivity Classification for Non-Hermetic Solid State Surface Mount Devices

SOLDER FILLET FORMATION

Solder fillet formation is defined by the following:

- Volume of paste printed
- Plated through-hole size
- Plating type and thickness
- Component lead diameter
- Metal content of paste
- Thickness of printed board
- Printed circuit pad size
- Component lead pitch

A basic formula that is often used was originally produced by AMP Incorporated some years ago to assist design and process engineering staff when using AMP products. Its aim was also to meet the IPC inspection standards for solder joints produced by hand or wave soldering on standard 0.063" (1.6mm) thick boards. Since many applications use thinner boards and even flexibles today, the process of through-hole fill is simplified. It is also fair to say that it is very rare for plated through-hole solder joints to fail. It is more common to have the component leads break or the copper barrel pulled out of the printed board assembly during mechanical testing.

PIN-TO-HOLE RATIO AND PAD SIZE

As a starting point, a design engineer should consider using standard design rules for the pin-to-hole ratio and pad size, comparing the results and paste requirements with the formula in this chapter or with any of the other resources featured in Chapter 10 - Suggested PIHR Resources. The resist aperture size around the through pads, which is normally the pad size plus 0.004"/0.006" (100µm /150µm), should also be considered. This size may be changed to assist with paste

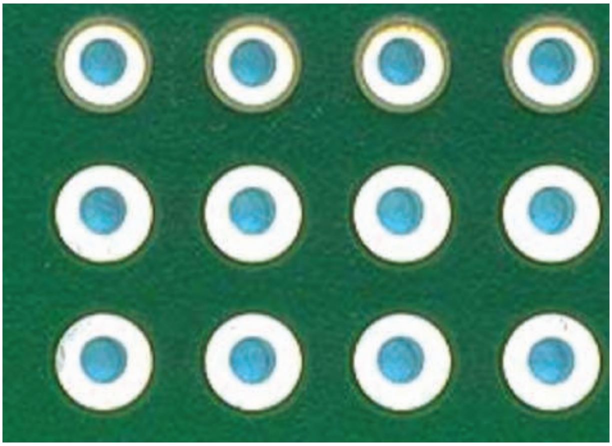


Figure 1-1: Example of copper-defined pads and solder mask-defined pads on the surface of a printed circuit board. The solder mask-defined pads are shown on the top row.

application and damming between joints for reflow. Experience has been gained over the years with using solder mask-defined and copper-defined pads, but traditional copper-defined pads are still the most common method used in industry. Just for clarity, copper-defined pads have a gap between the copper and the solder mask and are considered standard industry practice. Resist-defined pads have the mask overlapping larger copper pads to define the exposed pad area for soldering.

THROUGH-HOLE ASSEMBLY AND SOLDERING

Normal guidelines for through-hole assembly and soldering are the pin size plus 0.010” (254µm), which defines the finished hole size after drilling and plating and is the minimum recommended hole size for automatic assembly. The pad size is the maximum hole diameter plus twice the minimum annular ring plus any fabricator allowance. Care should be taken in discussing the capability of the hole size tolerance from the PCB supplier. The final hole size after plating and any surface finishing should be specified on the PCB drawing or specification, not the drill size. Tin/lead or lead-free solder leveling will have a greater hole tolerance than copper OSP, tin, silver or nickel/gold surface finishes. When solder leveled boards are specified for better long-term solderability, the finished hole size is important due to the potential variation of thickness through the hole.

SQUARE OR FLAT PINS

In the case of square or flat pins, the pin size for calculation is taken across opposite corners. There are, of course, many connectors today that have punched and formed leads rather than round or square pins. The pin type has changed over the years due to manufacturing costs. Some may be V- or U-shaped to improve rigidity, but have an open section. This is a little more difficult to calculate and will need the old engineering tool - guesstimating! Although not necessarily a reliability issue, there is a tendency to see a few more voids after reflow with V, U or kinked pins when microsectioning or examining the joints by x-ray. However, they have virtually no impact on joint strength.

THICKNESS OF THE BOARD

The thickness of the board is another factor, along with the pin length, that can be an issue in assembly. When specifying the component pin length and calculating the pin protrusion, long pins can be an assembly issue. If the pin protrusion is excessively long, paste can be displaced so far away from the joint area that it is difficult to reflow back to the joint. In some cases, much of the solder will be coating the pin and can deplete the joint area. Use the normal protrusion guidelines of approximately 0.280" (1.5mm) recommended in IPC design and inspection documents.

There is no real reason not to have shorter pins for intrusive reflow other than to meet customer inspection standards and to possibly use them for test access, which should be avoided if possible. There are many cases where products have been built with lead lengths with less than 1.5mm protrusion or no lead length at all. This may be where a common stock connector is used on a number of products for economic reasons, but the thickness of the printed board varies. Another reason may be a high frequency application where it has been shown to be beneficial not to have a solder joint/pin through the whole of the board. In this type of product, it is possible to fabricate the board with the hole countersunk on one side so the through-hole plating only extends to a selected layer on a multilayer construction, making solder joint fill easier to achieve on very thick boards.

LOCATION OF DESIGN FEATURES

The location of via holes, test points and any other design feature needs to be borne in mind. Since the solder paste will normally be printed and will extend over the pad onto the surface of the solder mask, vias may scavenge solder required for the through-hole. A test point may also allow some of the valuable solder volume to wet the pad and be lost to the joint area. Solder paste does not reflow very well over legend ink. Solder balls tend to stick to the ink when reflowed, so keep legend to a minimum or eliminate it on your design. Putting nice white or yellow boxes around every component is a thing of the past, and in most cases today, there is no space for legend.

STENCIL DESIGN

It is difficult for the design engineer to determine what the process engineer is going to do in terms of stencil design. *[Author Note: Process engineers, like me, like to play and try out different things in manufacture, so give them some room to manoeuver wherever possible.]* Design teams should visit the shop floor and watch trials being conducted to determine a no-go area for printing on future products. If this is not done, it is possible for shorts to form between topside mounted parts and vias. In the case of through-hole parts, these would not be visible.

SOLDER JOINT VOLUME

The calculations in Figure 1-2 are used to determine the solder joint volume for pin in hole reflow soldering. In each case, the solder volume is the area left after deducting the pin volume from the plated through-hole. If positive fillets are required for the solder joint specification, this additional volume needs to be added to the calculation. Different formulas have been published in technical articles over the years, many of which can be found in Chapter 10 – Suggested PIHR Resources.

Volume of Paste = (Vpth - Vpin) x 2

2	multiplication factor to compensate for shrinkage of the paste during reflow. After reflow the actual volume taken up by solder paste will be approximately 50% of its printed volume.
Vpth	volume of the through-hole cylinder
Vpin	volume of the pin cylinder

Volume of a cylinder = ∏ r 2 h

This formula does not specifically determine the solder paste volume to obtain positive fillets above the surface of the pads on both sides of the board. An alternative formula will need to be used.

EXAMPLE OF EXCEL SPREADSHEET FOR PREDICTING SOLDER JOINT QUALITY

The spreadsheet produced by Alan Hobby of DEK may be useful and a sample is provided below. It may be possible to obtain a copy of the spreadsheet by contacting DEK at http://www.dek.com.

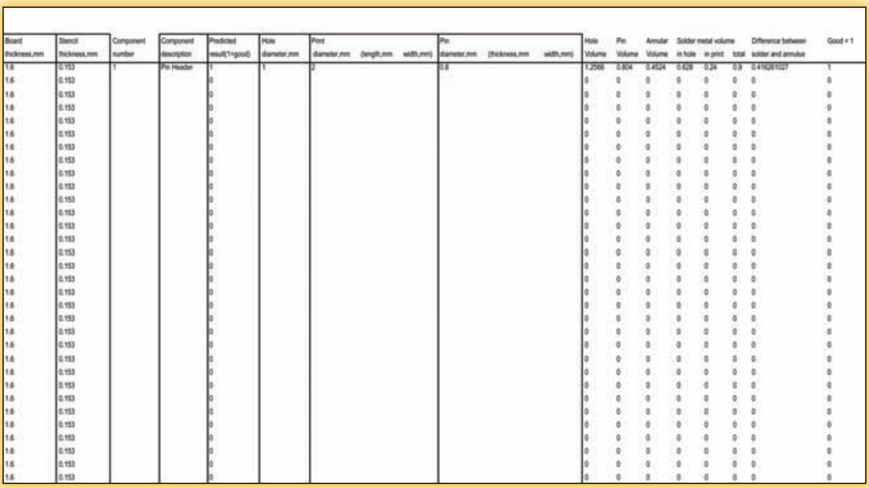
Instructions for the use of the spreadsheet were written by the author so he could easily describe the use of the spreadsheet in a SMART Group Charity Report that Alan Hobby contributed to that raised money for Children in Need in the UK and the SMTA's Charles Hutchins Educational Grant in the United States.

The spreadsheet provides an indication of expected solder joint quality. By entering details of the printed board thickness, component termination and stencil aperture size, the solder joint quality can be predicted. The following procedure was written by the report's author to assist in completing the form and to better understand the resulting information.

Column	
1	Enter the board thickness. All dimensions should be entered in metric.
2	Enter the stencil thickness you intend to use in production.
3	Enter the component reference number. This may be used for later reference to avoid reassessing components in the future.
4	Enter a description of the component.
5	This column calculates the expected solder joint quality from the information entered into the first four columns. It is based on the general requirements of IPC 610.

6	Enter the final required size for the plated through-hole.
7/8/9	Enter details of the stencil aperture size - either the diameter for a round aperture or the length and width if square/ oblong.
10/11/12	Enter either the diameter of round pins or width and material thickness in the case of square/oblong terminations.

After entering these dimensions, the spreadsheet displays the pin volume and hole volume. The calculated solder volume is provided for the hole. The print and total are calculated from the stencil thickness and aperture dimensions. The spreadsheet also displays the difference between the solder and the annulus. **PIHR**



Board thickness,mm	Stencil thickness,mm	Component number	Component description	Predicted result(1=good)	Hole diameter,mm	Print diameter,mm	(length,mm	width,mm)	Pin diameter,mm	(thickness,mm	width,mm)	Hole Volume	Pin Volume	Annular Volume	Solder metal volume in hole	in print	total	Difference between solder and annulus	Good = 1
1.6	0.153	1	Pin Header	1	1	2			0.8			1.2566	0.804	0.4524	0.628	0.24	0.9	0.416261027	1
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0
1.6	0.153			0								0	0	0	0	0	0	0	0

CLICK FOR LARGER VIEW

“Early conversions of connectors and sockets with surface mount lead terminations suffered from problems of lead coplanarity.”

CHAPTER 2

2

PIHR Component Guidelines

PIHR COMPONENT ISSUES

Many component suppliers have realised the benefits of reflowing through-hole parts rather than converting to a surface mount equivalent with either gull wing, J or other forms of surface connection. However, many design engineers have been slow to see the benefits of PIHR in manufacture. Often process engineers have been reluctant to push the process benefits in design lead companies. Early conversions of connectors and sockets with surface mount lead terminations suffered from problems of lead coplanarity. Larger components also had issues with package warpage and different expansion rates during the heating and cooling of soldering or rework operations. Differences in the dimensions resulted in pin-to-pad misalignment on the surface of the circuit board. The reduced strength of surface mount-only joints without some other mechanical support has been held as a reason for not using surface mount where the operation of sockets and connectors may put considerable strain on the smaller joints.

Large surface mount components will always suffer from coplanarity issues. Therefore a conventional through-hole part still has its place in modern designs. During normal reflow, if the board is not supported, the connector or socket may remain rigid, but the board can sag as the laminate goes through its glass transition phase, leading to large gaps between the pad and the lead. The limited solder volume provided for fine pitch connections may not be able to maintain a reliable connection. This problem is virtually eliminated with through-hole leads and PIHR.

TEMPERATURE COMPATIBILITY

The main component issue for PIHR is the higher temperature the component may experience during normal production. Generally, tin/lead reflow soldering is conducted between 210°C and 225°C and lead/free between 230°C and 250°C. Parts may be exposed to peak temperatures for up to 60 seconds. Many specifications require compatibility of all surface mount parts at 245°C for tin/lead processes and 260°C for lead-free alloys for a minimum of 10 seconds and up to 40 seconds. It is sensible to test potential through-hole parts for temperature compatibility during process development within the manufacturing operation rather than rely completely on suppliers' data sheets.

Normally, through-hole components would not be exposed to the extremes of reflow temperatures during wave or selective soldering on the top side of the board. The peak temperature on the top side of the board, between the surface of the circuit board laminate and the body of the component, would be between 120°C and 170°C for a short period of time. For those engineers moving from tin/lead to lead-free and still running wave or selective soldering, it is sensible to check that existing through-hole components can stand up to the temperatures reached during these processes. There have been many cases where design engineers did not consider the increased temperatures on the top side of the board on traditional through-hole soldering during alloy conversion. Cheap components just do not stand up to the process. Often Restriction of Hazardous Substances (RoHS) compliant does not mean lead-free compatible.

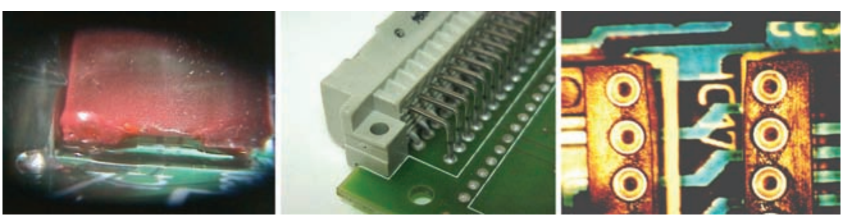


Figure 2-1: Impact of high top-side PCB soldering temperatures on the body of selected components after a traditional through-hole soldering process. It is important to check top-side temperatures during wave or selective soldering and run a temperature profile.

MATERIAL COMPATIBILITY

Component compatibility is often specified by its material type or some of its characteristics. Unfortunately, this does not always work in practice. Components may be susceptible to stress cracks built up during moulding. They may also change dimension due to the moulding procedure, the percentage of filler and many other factors. The pin terminations may float in the housings, leading to poor location with adjoining plugs, something the material type or classification would not necessarily highlight. The most common areas examined after testing are dimensional changes, visual damage, blistering, warping and mating compatibility. Each of these is compared with parts not exposed to the testing procedure.

DESIGNS AND MATERIAL COMBINATIONS

Even if the parts are material-compatible, specific designs or material combinations can introduce stress points into the moulding that can lead to cracking. High levels of moisture in plastic parts can lead to surface blistering, even if the plastic itself has a high temperature rating. It is very important to work with the supplier during the development phase of the project and make sure the supplier requirements are defined so that a cheaper supplier is not substituted for in-house or subcontract production.

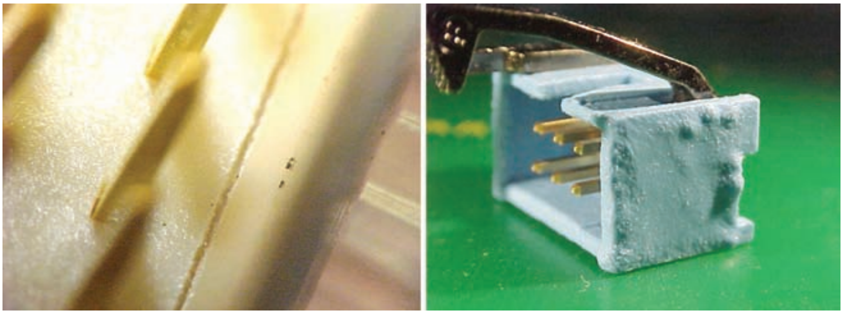


Figure 2-2: Cracking on the inside of a connector body, most likely due to the thin section in the moulding (left). Connector surface blistering can be related to moisture content or elevated temperatures (right).

STAND-OFF

Clearance under the component body should be available with some form of stand-off pip or foot. A minimum clearance of 0.010” should prevent the plastic body from contacting the paste deposit. If stand-off feet are present on the base, care should be taken to prevent them from smudging the paste deposit since this will lead to solder balling. The design of the component base and the paste deposit should be considered when specifying the stencil aperture. A stand-off is also preferred since it may allow some degree of visibility of the joint area beneath the component to improve confidence in the overall process. It will also facilitate cleaning if activated fluxes are employed during production operation.

When components are selected and the paste stencil aperture is designed, suppliers should not be changed without approval. The stand-off pins on the base of the parts can be in different positions and may not be compatible with the stencil design. It is not uncommon for professional stencil suppliers to ask for a sample of the component from new customers implementing PIHR to help them provide the best stencil design for their customers.

Ideally, all components will have a sealed body around the pins. It is possible that during reflow, flux vapors could condense into open apertures adjacent to the soldering areas, which may occur on top or

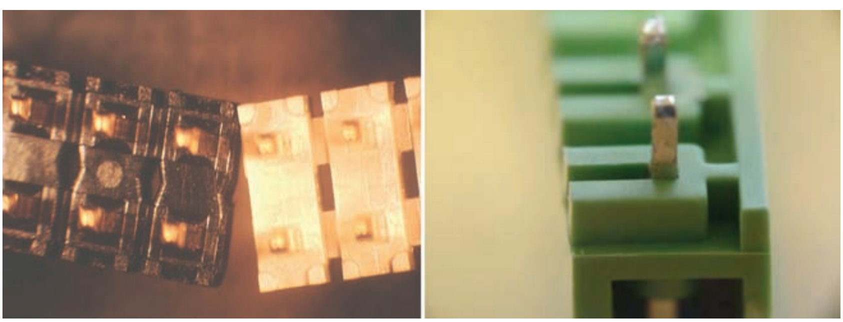


Figure 2-3: Moulding footprints and stand-offs that may require modification to the stencil design to avoid solder balls during reflow. If paste is printed under the plastic body, it will be displaced during placement and lead to micro solder balls.

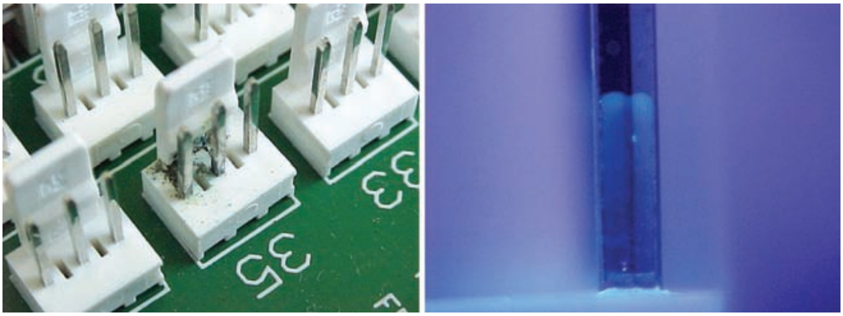


Figure 2-4: Impact of capillary action from the assembly process. Corrosion deposits on the surface of a connector after wave soldering and environmental testing (left). Conformal coating that has capillaried along the pin onto the active section of the pin (right).

bottom side access. This is not a unique issue with PIHR. It can happen in wave and selective soldering. It can also be seen during conformal coating, when the coating can capillary along the pins causing electrical opens.

COMPONENT PACKAGING OPTIONS

Component packaging options should be evaluated with suppliers for possible automation, even if manual assembly will be used initially. If the volume of production increases, it is important to know what options are available before committing to a specific part or supplier. When the technique of PIHR is first adopted, it is often used to eliminate wave soldering as a process. The next logical step is a reduction in the handling of the boards or second stage assembly. The automatic assembly of through-hole parts can also reduce the likelihood of displacing solder paste from the holes during insertion.

Often the most difficult problem with automatically placing parts is incorrect packaging used by the suppliers. Another issue is the ability to pick the part up, just like in the early days of SMT. At least connectors and sockets can be modified to allow vacuum pick-up. Either Kapton tape is applied to cover open socket pins, or a spring clip is positioned in the centre of the part to allow vacuum pick-up. These are then removed after reflow soldering.

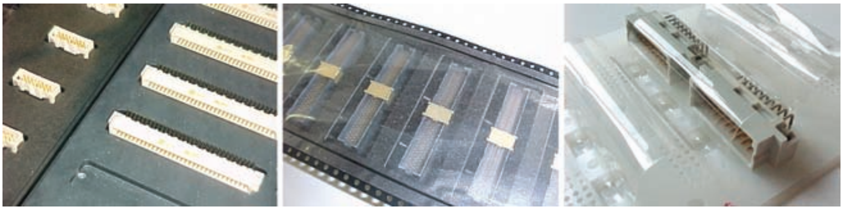


Figure 2-5: Forms of packaging for automated assembly using odd form or pick and place equipment. Custom pallets (left), tape and reel (middle) and custom tape (right) for a GPAX feeder system.

Today most components are available in packaging that is compatible with automatic assembly, like waffle trays, tape and reel, stick or custom plates. But never assume that this will be the case. Always ask the supplier. There are also specialist suppliers that can custom-make packaging and feeders for odd form parts. This may be another option when volume increases or the component supplier is unable to meet the packaging requirements

COMPONENT LEAD TERMINATIONS

Component lead terminations need some consideration in light of RoHS. Pre-RoHS, a tin/lead coating was preferred for soldering. In a lead-free world, the most common finish is tin. A gold flash over 2µm to 5µm of nickel is satisfactory for soldering, provided the gold is below 0.5µm. If brass or a brass-containing pin alloy is used as the base material, the pin must first be plated with copper or nickel to a minimum of 2µm before the tin/lead is applied. This is normally a standard practice due to zinc migration into tin/lead coatings, but some plating companies have been known to skimp on this operation. In many lead-free trials, the fast dissolution of copper to the circuit board is a concern with wave and selective soldering due to the alloy, temperature and flow rate during mass soldering. However, it is less of an issue with reflow soldering. There is generally a thicker intermetallic in a lead-free joint with high levels of tin at elevated temperature, but since the solder in reflow does not constantly wash across the surface of the plating, dissolution of the barrier or base plating is less of an issue. A guide to copper dissolution produced by the Environmental Protection Agency in the National Priorities List (NPL) is listed in Chapter 10 - Suggested PIHR Resources.

COMPONENT PIN SIZE

Generally, the minimal variation on component pin size is not a problem for PIHR, but it should be checked with the supplier. The tolerance needs to be defined in the component specification or it could change and, where design engineers have defined a smaller than usual hole size, problems could occur during automated component insertion. However, problems of lead insertion are far more likely to be associated with incorrect drilled hole size or variation of copper plating from different circuit board suppliers.

COMPONENT PLACEMENT PARTS

Some types of components, like sockets, may have solderable and non-solderable guide or retention pegs that help locate or retain the parts in position prior to soldering. If they are interference fit in any through-hole, it may cause problems during surface mount placement due to the excessive force required during insertion. If inserted after printing and surface mount, component placement parts may be displaced from their correct positions due to the force required to insert the leads. Always assess the correct design rules for active and retention pins and never assume that the supplier's design guidelines have taken all assembly operations into consideration.

LEAD LENGTH

Lead length has been mentioned since it can affect solder paste pushout. This problem is mostly overcome with the paste wicking back to the joint area during reflow, provided that the temperature profile and the time above liquidus for the solder alloy have been correctly defined. If the hole-to-lead ratio is small or the paste has dried out, long pins can displace significant amounts of paste onto the tips. As a guide, the pin length protruding from the board should be 1.0mm to

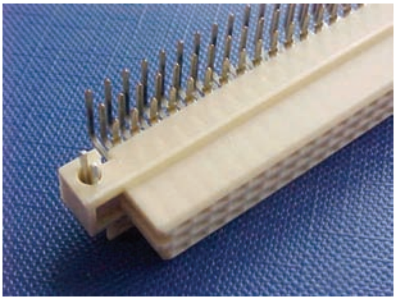
1.5mm in length. The degree of pin float should also be checked. Ideally the total circular float around the true centre position should be no more than 0.005".

PIHR APPLICATIONS

Although we have covered the design aspects relating to PIHR, it is important in this practical guide to illustrate some successful applications with specific design dimensions for selected connectors. It is fairly obvious that engineers will have to adapt their designs to the components of choice, but hopefully these examples will emphasize the points raised in this chapter, illustrate where some of the dimensions and recommendations have come from and explain why they are in place.

Design guidelines are provided for selected connectors illustrated below. Care needs to be taken on differences in tolerances and connector design with different suppliers even though the parts may meet international standards. The recommended finished hole sizes, stencil apertures and layout are based on many production trials. In each case, it is possible to print solder paste using a 0.006" (150µm) stencil to achieve solder joints that exceed the minimum IPC requirements on a 1.6mm board thickness.

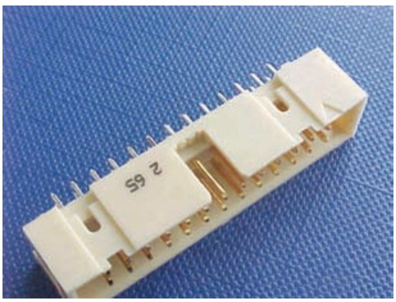
By optimizing the solder paste printing process for traditional metal squeegee blades and possibly using an enclosed head printer process, a 0.006" (150µm) stencil may be used, which will exceed the IPC



Hole Size: 0.031" (0.79mm)
Pad Size: 0.060" (1.52mm)
Resist Aperture: 0.066" (1.68mm)
Fixing Holes: 0.145" (3.68mm)
Pad Size: 0.180" (4.57mm)
Resist Aperture: 0.186" (4.72mm)
Stencil Aperture: 0.085" (2.16mm)
Fixing Holes: 0.150" (3.81mm) 0.015" cross webs: (0.38mm)

Figure 2-6: Type C DIN 41612 Connector level 3 requirements and also produce satisfactory hole fill on thicker substrates. In each of these examples, the stencil aperture is positioned over the plated through-hole. However, based on the connector design, hole fill requirements or track layout, the print aperture design or position can be modified. The recommendations have proved suitable for use with conventional tin/lead and lead-free solder pastes.

Let's look closely at the two connector designs and some of the possible challenges faced in design and manufacture. Figure 2-6 is the 96-way edge mounted connector Type C DIN 41612, which has been very popular in the telecom industry for many years. The main con-



Hole Size: 0.035″ (0.89mm)
Pad Size: 0.055″ (1.40mm)
Resist Aperture: 0.061″ (1.55mm)
Stencil Aperture: 0.080 x 0.080″ (2 x 2mm)

Figure 2-7: IDC Header DIN 41651 Connector

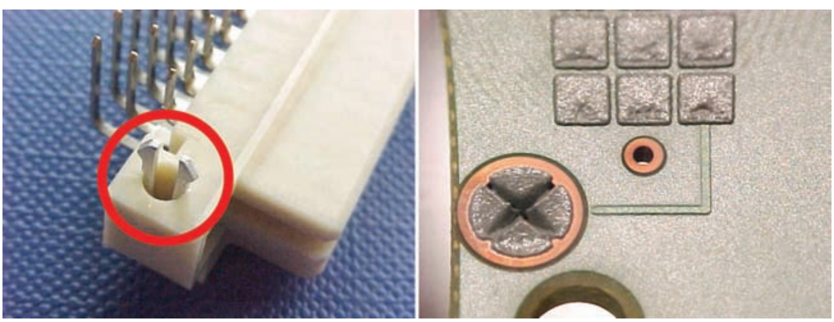


Figure 2-8: Connector hold-down barb (circled in red) and the corresponding location on the printed board after stencil printing solder paste. The solder paste deposit is split into four deposits in the through-hole and not on the surface of the pad so it is not squashed when the connector is inserted.

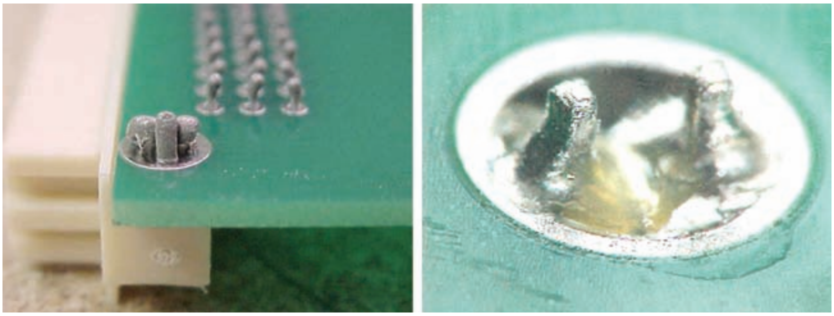


Figure 2-9: Paste displacement after connector insertion hold-down feature (left). The resulting joint is considered satisfactory and totally reliable (right).

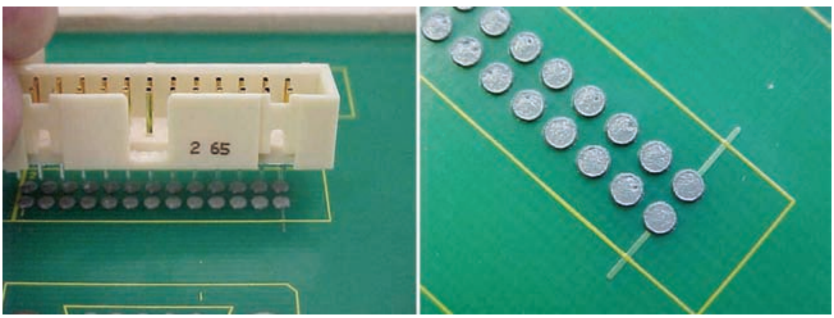


Figure 2-10: IDC post header being manually inserted after stencil printing with a sealed head printer (left). Solder paste print on the surface of the solder mask (right). In both cases, the image shows a round stencil aperture. Square apertures provide a greater volume of paste than the same size round apertures.

nector pins are very straightforward in terms of through-hole reflow. The solder paste is printed with a square aperture centrally placed over each of the plated through-holes and extending over the solder mask surface to increase the solder volume.

Connector hold-down features generally come in two options: either a retention barb (Figure 2-8) or a hole that allows mechanical fixing, nuts, bolts and crinkle washers to be used. If the barb is to be used and the through-hole soldered, then the stencil design has to accommodate this feature. You cannot simply print paste on the surface of the board since it will be displaced when the connector is inserted. The plastic body of the part will contact the solder paste on the edge of the board.

After connector insertion, some of the paste is displaced around the pin. When it is reflowed, it provides a reliable mechanical contact with the through-hole plating. It should be noted that the hole will not be completely filled with solder. There simply is not enough paste volume. However, the joint is 100% reliable, and during mechanical testing the pin normally always pulls out of the connector housing.

A couple of extra points on this connector type which is provided by a few suppliers in the industry. It is necessary to modify the supplier's recommended hole sizes for the active pins to accommodate PIHR. Check the finished hole size for the two-hold down pins. They are normally specified as an interference fit, but should not be in this case. **PIHR**

PIN IN HOLE/INTRUSIVE REFLOW CHECKLIST

A Simple Q&A to Help Get Things Right

❑ ***1. Are your through-hole components compatible with reflow soldering temperatures? Do they meet the minimum requirement of the IPC/IEC component compatibility standards of 245°C for tin/lead soldering and 260°C for lead-free for a minimum of 10 seconds and up to 40 seconds?***

❑ ***2. Can you obtain the through-hole components in packaging suitable for automatic assembly?*** Manual assembly is possible, but automation will improve consistency.

❑ ***3. Have you calculated your through-hole and lead-to-hole ratio for automatic and manual insertion?*** You may have to do this if you normally group holes to reduce the number of drill sizes in printed board manufacture. Lead size plus 0.010" is normal.

❑ ***4. What stand-off height do you have on the components? Where are the component stand-off feet located? Will they contact your paste deposit from your stencil design?*** A minimum stand-off height should be 0.010".

❑ ***5. Have you tested your solder resist with your solder paste during reflow? Does it cause solder balling?*** It is often necessary to print paste onto the resist surface to obtain the correct volume of solder, which then pulls back to fill the hole during reflow.

❑ ***6. Have you calculated the stencil thickness required to fill your plated through-holes with solder after reflow?*** The following calculation will provide a basic initial guide:

Volume of Paste = (Volume of PTH - Volume of pin) x 2

❑ ***7. Have you told your stencil manufacturer that through-hole apertures are required on your new stencil?*** Normally we tell the stencil supplier to take them out. Have you shown your stencil supplier a sample connector?

❑ ***8. Have you discussed changes to your soldering standards for PIHR assembly with your quality department, inspection staff and customers?*** You can achieve 100% fill, but positive fillets are more difficult. The joints may also look different.

❑ ***9. Have you specified your component lead lengths? Can you control them?*** Lead length control is crucial and should ideally give a protrusion of 1.0mm to 1.5mm below the board.

❑ ***10. Do you know how strong solder wave and hand-soldered joints are?*** They are no different than through-hole reflowed joints, but remember! Someone will ask you!

If the answer to any of these questions is NO — STOP and ask some more questions. Good luck with your Pin In Hole Reflow process.

3

> "Printing twice onto a board without separating the PCB and stencil is a way of increasing the paste volume in the hole."

CHAPTER 3

Solder Paste Application Methods

INTRODUCTION

The most common method used in Pin in Hole Reflow (PIHR) manufacture worldwide is printing solder paste into plated through-holes and over the pad surface during normal surface mount paste printing. This could be on side one or two of the board depending on other design factors. The size of the aperture in the solder paste stencil may be adjusted to allow paste to fill the hole, cover the pad or extend over the surface of the solder mask to obtain the solder volume required.

If the stencil and board are correctly positioned, and all the apertures are gasketed to pads or the solder mask surface, a double print stroke could be undertaken, but is never an ideal option. Printing twice onto a board without separating the PCB and stencil is a way of increasing the paste volume in the hole. During the second print stroke, no more paste is applied to the surface pads, but paste will flow farther into the through-hole barrel.

The downside of this process is that paste may be forced under the stencil between the stencil and substrate, which may impact fine pitch printing, causing wet paste shorts and affecting product throughput on the production line. Therefore, through-hole component insertion is conducted either manually or automatically before the whole board assembly passes through a reflow soldering process.

SINGLE AND DOUBLE STENCIL PRINT

Since it may be difficult to obtain the volume of solder for a through-hole component if the hole-to-lead ratio is excessive, a double print stroke operation is possible. If the hole-to-lead ratio is large when the first engineering checks are conducted in New Product Introduction (NPI) or at the quotation stage in subcontract, discuss the incorrect hole size with the designer. It is easy to change the hole size before the boards are fabricated and in nearly all cases it will not impact other processes, electrical requirements or reliability.

For this type of double print operation, the lowest pressures should be considered during printing or the pressure should be changed on the second stroke, since paste will be displaced under the stencil. It is important to remember that the metal content of paste is only half the volume of the deposit during printing. Alternatively, an initial printing operation may be conducted to force paste into the holes only. A second printing operation is then conducted to add more paste to the hole, pad and resist window, as well as surface mount parts. This increases the solder volume in the through-hole joint. The second print operation would also be used to apply paste for the standard parts. This operation requires two printers and two stencils and is not an engineer's first process choice.

Some companies have reversed the operation by doing standard surface mount printing operations first. After the first print, a second print is applied to the through-holes only. This requires a stepped stencil. Etched recesses are produced on the bottom of the stencil to pre-

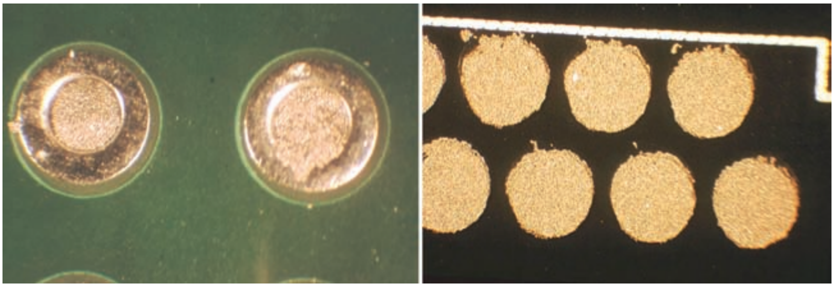


Figure 3-1: Through-holes filled using a double print stroke during paste application (left). Round paste deposits that have been over printed (right), viewed from the base of the board prior to inserting the component leads and showing 100% fill. The round paste deposits show evidence of paste squeeze-out under the stencil due to the double print stroke.

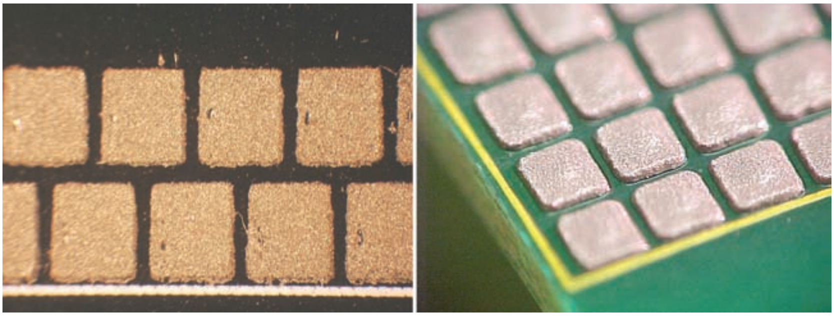


Figure 3-2: Staggered square paste deposits (left) that can increase the volume of solder left after the reflow soldering operation compared to a round deposit. Square aperture print (right), in this case for a five row connector.

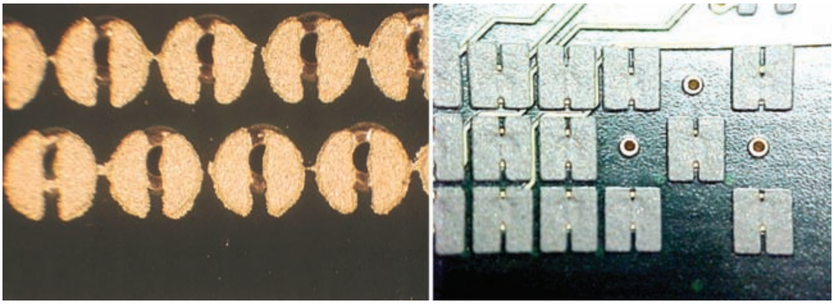


Figure 3-3: Split paste deposit design (left). Two completely separate half-moon deposits on opposite sides of the plated through-hole pad (right). Both deposits are joined together, straddling over the surface of the through-hole. Both of these aperture designs provide a point of reference for manual lead insertion or alignment check on a placement process.

vent the first print from being smudged. The specification of the print thickness for the first print must be carefully considered, making sure the recess fill maintains a non-contact gap with the first paste deposits. This is unnecessarily complicated, so it is much better to change the specified finished hole size in the design to optimise solder fill.

Theoretically, round split paste deposits are used to aid in the manual assembly of components. This print technique was conceived so operators could see where the holes are in relation to the paste deposit.

However, it is not recommended due to the solder balling that may occur during reflow. It may be better to only use this technique selectively on corner pads to aid in manual alignment. During reflow soldering of split deposits, if the solderability of the surface finish is poor or slow to wet, solder particles may not wet back to the through-hole or pad surface. The same is true if the design of the pad surface is small in relation to the hole size or the total surface area of the aperture provides too much paste volume for the surface area.

In terms of process control, the amount of solder paste required for through-hole soldering is much greater than for surface mount. After conversion to PIHR, paste can easily increase in weight to double or even more than in the original design. The increased paste weight will require more frequent checks on paste volume and bead size on the stencil surface since they will be depleted during printing.

There may be situations where the flow of solder paste needs to be restricted when printing into large plated through-holes. If the viscosity of the paste changes during production on holes over 3mm, some of the paste volume may fall out. If there is a great deal of hole size variation in the design, you can also see more paste penetration, hence the reason to use good stencil design techniques to restrict paste volume.

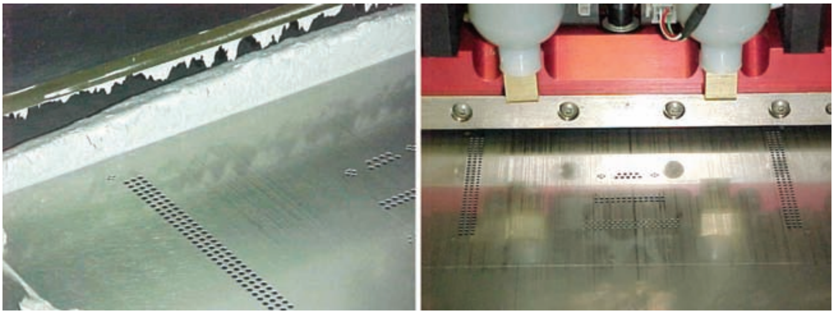


Figure 3-4: Typical solder paste bead in front of a metal squeegee blade (left). If through-hole parts were positioned on the right side of the board for the print stroke, this is where you might notice changes in the width of the paste bead over time. Sealed head printers monitoring the pressure inside the print head and via software adjust the process parameters or warn the operator to top up the paste (right).

Figure 3-5: Support block showing an open aperture positioned below the surface of the board for components and one open section to avoid paste build-up during printing for PIHR.

It is also possible to block the bottom of the hole with either a magnetic support pin or part of a support plate used on double-sided board designs with pre-mounted components or on thin substrates. Care must be taken on the surface that is directly in contact with the paste. The material used should be selected to avoid paste sticking and being pulled out of the large hole, resulting in a build-up that will contaminate other board surfaces. When printing boards with solid support blocks, an alternative to this is recessing, which removes the area directly below the holes.

Recessing keeps any paste that may be protruding prior to connector insertion from contaminating the printer support block. The recessed areas need to be checked regularly for paste contamination. *[Author Note: As a general comment to engineers using vacuum tooling on PIHR, consider the exposed holes on the board. Will they allow the*

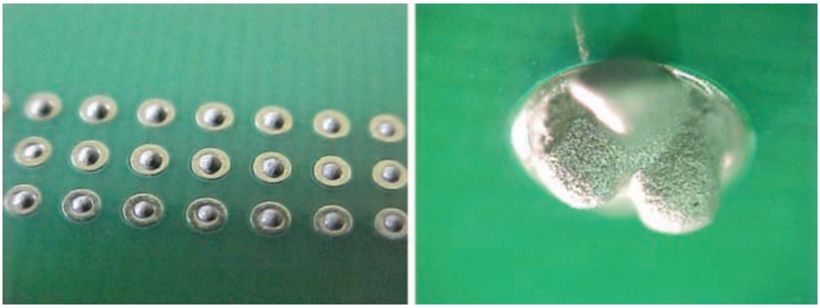


Figure 3-6: Examples of paste protruding below the surface of the board after the printing operation and prior to component insertion. Paste in 0.9mm holes (left). 3.5mm fixture location holes (right).

benefits of vacuum to be lost when holding the board flat or will the vacuum take paste where you don't want it? Also, remember to check any support tooling on your placement process as part of your regular preventative maintenance schedule.]

To prevent paste from seeping out during printing, the support feature needs to be 2mm to 3mm larger than the hole. For small volume production, Kapton tape placed on the support pin or blocks has worked. *[Author Note: Please remember that these techniques are manufacturing solutions to process issues and not something you would design into a process. It can work and overcome today's problem, but don't let it become the standard for your manufacturing process.]*

Any automatic printer with programmable paste dispense capability will need to be reprogrammed for the through-hole parts. In the case of semi-automatic or manual printers, the operator will need to top-up the paste on the stencil more frequently. It is important for operators to look at the solder paste bead size as it rolls across the surface of the stencil pattern. Most paste volume will be used where a connector or multiple connectors are located in the path of the squeegee blade. However, this should not be a problem with solder paste dispensing operations or where sealed head printers are being used.

With a good design optimized for intrusive reflow, the standard stencil printing operation with a single stencil and a metal squeegee blade works very well. The use of sealed head printers have provided enhanced performance for hole fill particularly on thick boards over the standard 0.063” (1.6mm). Sealed head printers were introduced by DEK and MPM to improve paste open life on the stencil, reduce paste waste and aid in hole fill. The DEK system is referred to as ProFlow™ and the MPM head is called the Rheometric Pump (Rheo Pump™). Other printer suppliers have offered sealed head options, but MPM and DEK have the greatest market share and considerable process experience with different suppliers’ paste products for sealed head application.

During any printing operation an additional problem may occur. Under-stencil contamination may be due to the higher pressures used to help hole fill. If the stencil aperture for through-hole parts is larger than the through-hole pad, it will contact the resist surface, which may or may not provide a positive gasket. In any printing process, if all of the stencil apertures do not fully gasket to the surfaces on the circuit board, some level of paste will squeeze between the stencil and the substrate, which may lead to paste on the base of the stencil and require extra under-stencil wipes. When first introducing PIHR, there is a tendency to increase print pressure on metal blades or place greater pressure on the sealed head printing process in an effort to improve paste filling the through-hole. Care must be taken with this since it

Figure 3-7: X-ray photographs show the degree of fill before (left) and after (right) lead insertion. X-ray used before through-hole assembly to assess the volume of solder paste fill after printing is recommended to check for paste voids or measure the variation in the printing process.

will also lead to wet paste shorts and stray solder paste deposits on the surface of the board.

Solder paste fill of the plated through-hole can be assessed by looking at the top and the bottom of the board after printing. *[Author Note: But, if available, x-ray is a more elegant technique that I recommend for process optimisation.]* X-ray is beneficial if the body of the part covers the paste deposit. It would show evidence of paste displacement that is not visible on optical check images taken on a DAGE system

SOLDER PASTE DISPENSING

Solder paste dispensing allows a greater volume of paste to be applied than stencil printing. The only limitations are the need for a dispensing system and the speed of dispensing compared with printing. The board will be printed in any case, unless dispensing is used as a standard process. Depending on the number of terminations, the throughput may be affected in an in-line process.

Dispensing paste has always been a benefit where small volumes of any one design are produced or where there are many hundreds of different designs that will inevitably require the equivalent number of stencils. Dispensing is also flexible since a simple program change is all that is required to change the volume of the paste, rather than changing the stencil thickness or aperture size. Stencils can be relatively inexpensive, but it still takes time to receive a new order which can hold up production. The innovation provided by Mydata with the My500 jet dispensing system offers the ability to paste the complete assembly and the flexibility to change paste volume in seconds.

Paste dispensing may also be used when the component has already been inserted into the board. Camalot Systems has demonstrated a number of applications where paste dispensing after insertion is possible. *[Author Note: Camalot Systems first illustrated this many years ago with high speed video and was featured on my first PIHR video. (For more information, visit BobWillisOnline.)]* In this case, a special nozzle was placed over the pin and the paste dispensed around and into the plated through-hole.

PIHR can be used for single-sided boards where the density of surface mount parts is high or the yield from a wave soldering process has proven to be less cost effective. The board or panel is first printed with paste and surface mount components are placed and reflowed. The boards are then inverted and paste is applied to the surface of the board between the components previously soldered. The board is then reinverted and the through-hole components inserted into the through-holes before passing through a reflow oven to form the solder joints.

Paste can be applied using multiple needles or dispensed through a single needle onto the surface of the joint. Alternatively, a piston dispenser system may be used with multiple nozzles onto all terminations in one operation. A piston system is preferred instead of pressure, due to the possible blocking of nozzles. In a piston system, the air pressure applied to all the remaining nozzles increases, providing excess paste to other joints. The same type of multiple nozzle system has

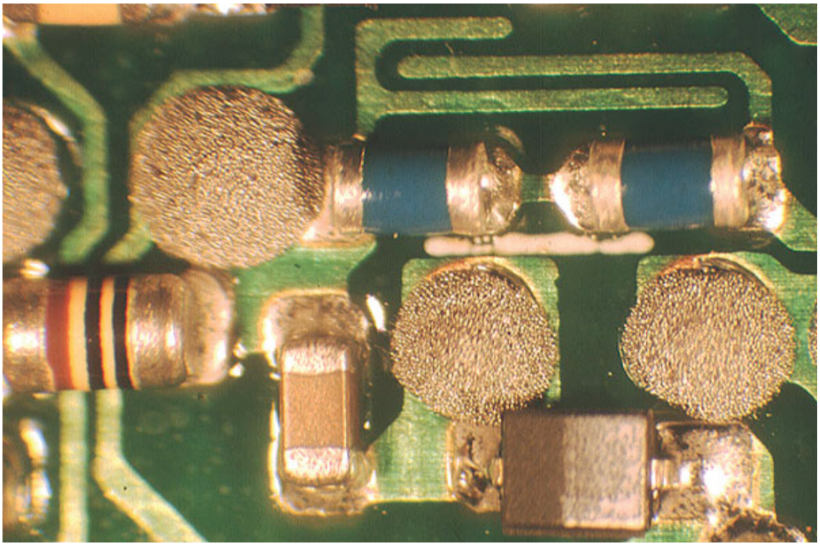


Figure 3-8: Solder paste that has been dispensed onto a single-sided board after initial reflow of the surface mount parts. This operation allows either manual or automatic insertion of through-hole parts prior to the second reflow operation.

been employed with a paste roller. In this case, a roller, resembling a paint roller, forces the paste through a series of nozzles onto the board surface. This is a simple process and has worked well in one UK-based company that produces audio boards for the automotive industry.

In the case of single-sided boards, the calculation for paste volume is basically a cone shape for the joint minus the volume of the lead. The size of the cone is taken from the pad diameter.

LD	=	Diameter of pad
CH	=	Height of cone
L	=	Lead diameter

Since the normal lead length is between 1.0mm and 1.5mm, the estimated height of the cone should be 0.5mm to 1.0mm. Ideally, the lead length should not be greater than 1.5mm due to concerns that pins will displace the paste too far away from the joint area. The lead dimension in a process like this is important for joint reliability on single-sided

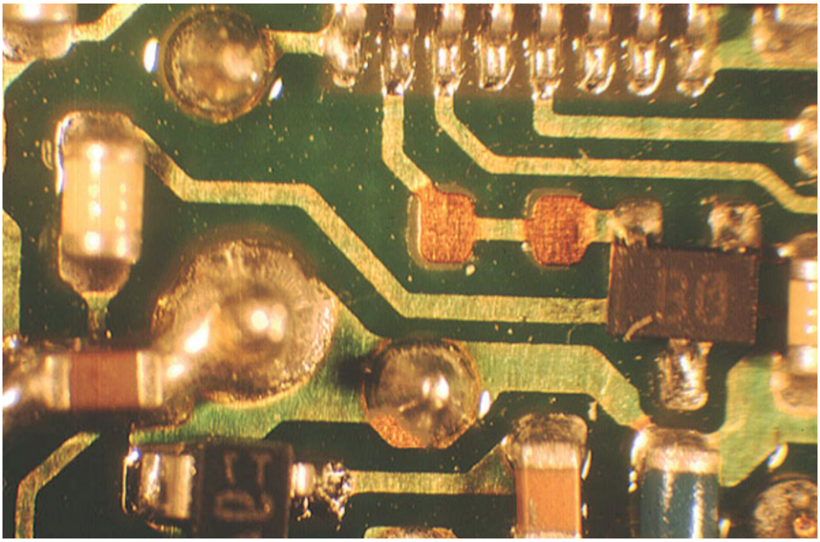


Figure 3-9: Single-sided through-hole joints formed after paste dispensing and successful reflow on a consumer audio product.

boards. The joint is formed between the pad and the surface of the pin and is much weaker mechanically. If the leads are too short, it could form a domed button joint, which could be an open circuit, especially if the components inserted are preformed and the tips of the leads are allowed to form thick oxides, making then difficult to solder.

A further variation in of the paste dispensing process was developed by Sony in Japan and the equipment sold under license to other companies. *[Author Note: During a visit to Japan on lead-free technology, I was introduced to the multi-spot soldering process, which had originally been used by Sony for the first DISCMAN products.]* The steps for the Sony process are illustrated in Figure 3-10

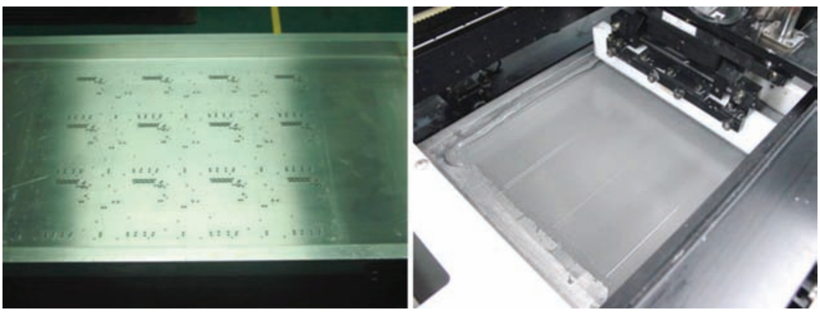


Figure 3-10: Solder paste stencil plate (left) and inline solder paste (right) printing for Sony's multi-spot soldering process.

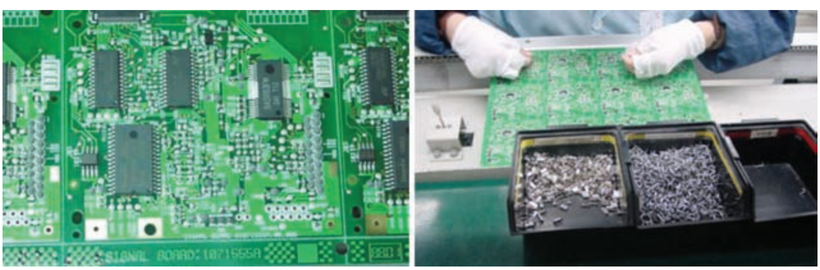


Figure 3-11: (Left) Multi-panel assembly after printing, with solder paste deposits visible over through-hole locations. (Right) Manual flow line assembly station with the preformed components in linbin storage containers ready for assembly.

Figure 3-12: (Left) Hold down/support fixture being used on through-hole components to hold the board rigid. (right) Reflow step where hot air is blown directly onto the through-hole lead and paste deposits to allow reflow.

with photographs from a production facility operated by an engineer attending one of my PIHR workshops in the US.

SOLDER PREFORMS

Another method of soldering through-hole leaded components uses specially designed solder alloy preforms in the same pattern as the stencil that would apply to the board. Solder of the appropriate alloy is formed into "doughnuts" that are linked together as a preform. This method provides 100% metal transfer as opposed to paste, which may only be 50% by volume.

During assembly, the preform is positioned on the component pins or on the board surface and held in place with a flux. The preform may be placed onto the pin ends if the component has already been fitted to the board. Most preforms do not include a fluxing agent, so this needs to be introduced as a second stage operation. Some suppliers of solder preforms or solder bricks offer flux coating and can add different activity fluxes as required for assembly. It is important that they are coated evenly and that this does not impact manual handling, automatic pick up or vision recognition if placement systems are being used.

As an alternative to using liquid flux with solder performs, solder paste has been used. Trials run by one company attempted to improve solder joint quality by providing extra solder using a preform with the paste provided by the fluxing agent. During assembly, paste was applied as part of the standard surface mount process. The preform was then applied to a Pin Grid Array (PGA) and the part inserted into the board prior to reflow. Trials were conducted with clean and no-clean paste, along with nitrogen. The results were similar to results I have experienced, where solder joints continually scavenged solder from adjacent joints, resulting in both excess and insufficient joints mainly due to the linked preform not breaking evenly during reflow. It is well-known that not all joints on a component reflow at the same time. The trials did not result in a stable process and are fully detailed in an article by Hewlett Packard (see Chapter 10 – Suggested PIHR Resources).

Soldering with preforms is hardly new. It was used by many telecommunication companies in the 1980s for soldering backplane assemblies before the introduction of press fit connectors. Using preforms was one of the first methods of pin in hole reflow assembly. The most common reflow process used in those days was vapour phase, which required extremely large systems. Unfortunately, the most common application for PIHR is socket or connector reflow, which tends to trap a great deal of fluid, which can make the process expensive.

Selected connector suppliers have also looked at introducing preforms on their connectors. In this case, they are often referred to as integrated performs. Integrated preforms may have a standard solder volume and may not suit all hole sizes or different board thicknesses.

SOLDER BRICKS

In recent years, solder material suppliers have produced solder bricks that are supplied in tape and reel packaging like chip components and can be pick and placed like any other surface mount component into a solder paste deposit. Like performs, they are 100% alloy and are used to increase the metal volume on individual through-hole locations where it may be difficult to achieve the required volume of solder. In my opinion, there is very little difficulty changing hole size to improve hole fill. However, there is justification on thick boards or

Figure 3-13: Solder bricks packaged in tape (left) and reel (right) format for automatic placement. These are available in different solder alloys from a number of solder suppliers. Indium Corporation uses the term Solder Fortification™ for their solder product.

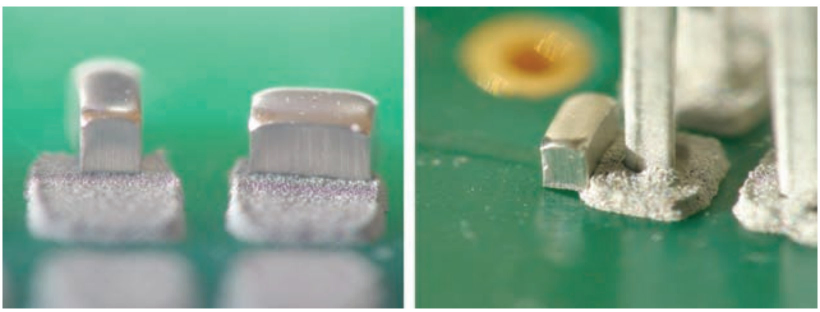


Figure 3-14: Possible placement positions (left) and the position of the paste/brick prior to the reflow soldering process (right). During preheat and prior to reflow, the flux vehicle does capillary under the brick and assists the brick pull into the joint.

where contract manufacturers cannot get approvals from customers to make value added changes to design.

As product design become denser, solder paste stencils become thinner, making it difficult to provide the volume of solder to selected joints on mixed technology boards. Heavy or tall components may be liable to stress in final operation, so providing more solder to selected surface mount components can be beneficial in an automated process. The use of solder bricks to add a greater volume of solder to these joint will increase joint strength, which is a good reason for the term solder fortification.

The parts are placed into the solder paste so that the tack of the paste can retain the solder bricks in place. Their position in the paste depends on the connector pitch and other features on the design of

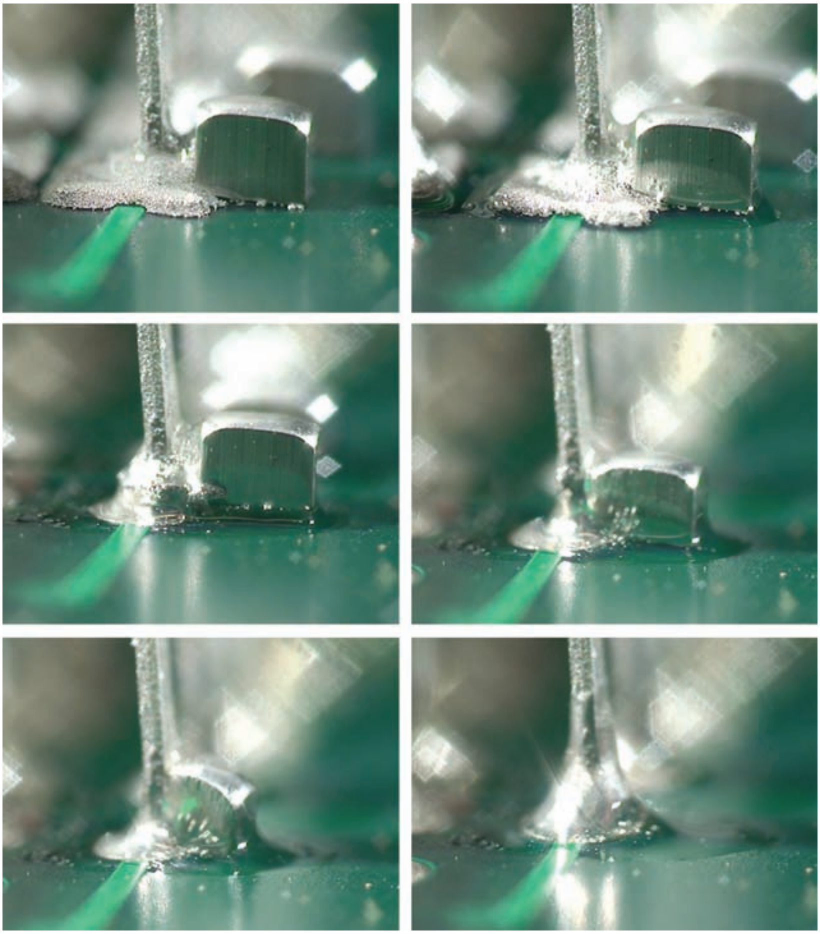


Figure 3-15: Sequence of reflow (left to right) taken from the author's video simulations of the reflow sequence using lead-free solder paste and solder brick on a through-hole connector. When the paste reflows, the solder brick is slowly pulled into the joint. Depending on hole size and paste volume, you can often see the solder brick being pulled down the hole. In recent years, there have been more examples of optical video simulations, and now x-ray is available. *[Author Note: I have used these for training workshops. Video is also used for product marketing, Alpha Metals features some good video clips on YouTube.]*

the board. It may not be possible to place these under the body of a component due to the standoff height or the moulded stand-offs on the component. Trials have shown that the position and angle is very forgiving provided that the reflow profile is correct.

SOLDER PASTE

Solder paste selection is based on the smallest lead pitch on the design rather than a pin in hole reflow product. Higher metal content, between 90% and 92% metal by weight, is preferred for both fine pitch and through-hole parts. A low residue paste is also beneficial if cleaning is to be avoided. No-clean production makes up the majority of the industry, but in the last couple of years cleaning has become popular partly due to the increasing use of conformal coatings. Cleaning reduces the possibility of test probe failure if through-hole pins are to be probed during in circuit test. A low-temperature metal is preferred for conventional component compatibility. The temperature variation between the solder types tin/lead/silver Sn62Pb36Ag2 and tin/lead Sn63Pb27 is relatively small, between 179°C and 184°C. Recently there has been renewed interest in low-temperature lead-free alloys, below 140°C, for commercial products to take advantage of the possible reduced cost of non-high-temperature components, particularly interface connectors. This may involve running a two-step reflow process with one side built with tin/silver/copper or tin/copper and the other side or selected components built with a low-temperature alloy like tin/bismuth/silver to allow through-hole components not suitable for traditional reflow to be used.

The alloy would typically be reflowed between 160°C and 190°C, depending on board mass and normal design constraints, but can eliminate issues associated with selective, wave or even hand soldering low specification components. *[Author Note: Trials have been run reflowing traditional and through-hole components in air and have been successful. But, remember that any trace of lead on the joint surface when using tin/bismuth/silver must be avoided to prevent creating a low-temperature interface that would reflow again at lower temperatures of 100°C.]*

The use of lead-free solder alloys has pushed the reflow temperature requirement up, forcing design and process engineers to look more closely at component compatibility. As seen in the past, through-hole leads have normally only been exposed to selective and wave soldering where the temperature on the top side of the board would be much lower than if parts were run directly through a reflow process. The most common lead-free alloy is tin/silver/copper, which has a reflow temperature of 217°C to 221°C, depending on the alloy mix. More recently, tin/copper pastes have been offered to the market with a reflow temperature of 227°C. The tin/copper family of materials has the benefit over most tin/silver/copper alloys of being more ductile.

Another requirement is to reflow without significant slumping of the paste. Hopefully most pastes fit the bill today. Another slightly different requirement is for the paste to be printed or dispensed onto laminate or solder resist to increase the volume available to form the joints. Some pastes and solder mask combinations may leave solder particles behind during reflow due to oxidisation, contamination from the resist or the mobility of the paste across the solder mask surface. Ideally, the more active the flux the better/faster the paste coalesces during reflow. The effects can be seen in in Chapter 9 – PIHR Manufacturing Defects – Causes and Cures.

Ideally, paste reflow trials should be conducted on resist surfaces to determine the reflow capability. Normally this is not something the paste or resist manufacturer would take into consideration during formulation, so it may be a new issue they will need to consider, hopefully before everyone starts complaining.

Reflow trials in nitrogen have shown some benefits to the flow of paste across the resist surface, but this will not be the most common reflow process used in manufacture. Based on the last survey of companies conducted on clean and no-clean, only 7% of companies were using a no-clean and nitrogen soldering process.

A final requirement of the paste is a long tack life, which is in line with all surface mount requirements. If the paste dries out quickly, there is a tendency to increase the amount of paste displaced during

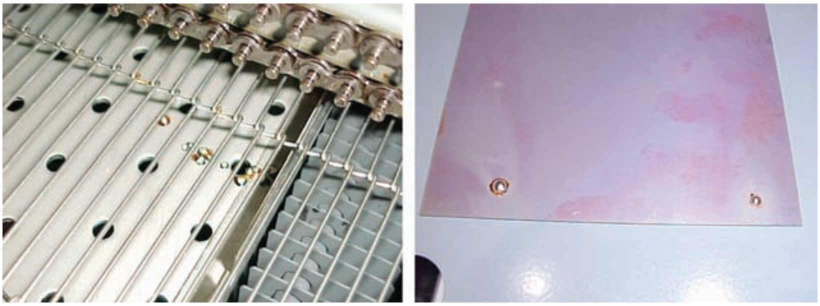


Figure 3-16: Paste lost in the early stages of reflow (left). Test results on a copper surface placed under an assembled board when passed through the reflow system (right). In this example, paste was lost from the connector mounting clips at the end of the connector rather than from the functional pins.

pin insertion. This is particularly true when boards are lowered onto connectors pre-mounted upside down in a pallet. The paste can be displaced as the dried paste deposit breaks away from the hole and lead to shorting or solder balls after reflow. As previously mentioned, a paste should not slump to meet its standard fine pitch surface mount printing requirements in manufacture. A further requirement is that the paste does not drip from the connector pin surface as it passes through the reflow oven since it will decrease the volume of solder alloy to form the joint and increase the contamination in the ovens for process technicians to remove during maintenance.

Some solder paste will be displaced on the pin during insertion and is then free to drip off during the preheat or soak period, hence the requirement to minimize all contributing factors. It's useful to gain an understanding of the paste performance and its tendency to drip. To do this, it's fairly easy to simulate reflow with a rework system and videotape the performance of some preassembled parts. Alternatively, it's possible to use a video simulator to conduct a reflow trial and monitor the performance of different paste products. A final trick is to run sample boards through a reflow system and position thin copper laminate or copper foil under sample boards as they pass through the process in order to easily see in what positions paste is lost.

SOLDER PASTE STENCIL

The stencil type and specification is defined by the surface mount and fine pitch components used, not by the PIHR process. If the design is limited to 0.050" lead spacing, a 0.008" stencil may be easily used in production to accommodate both process requirements. If 0.020" parts are incorporated in the design, a 0.006" stencil will normally be specified or a stepped foil used. A stepped stencil may be appropriate for a combination of PIHR and surface mount where all surface mount parts are at 0.006" stencil thickness and the through-holes are at 0.008". this approach is more likely to damage the squeegee blade with only selected raised areas.

The increased solder volume required for the through-hole parts is achieved through printing into the holes and over the pad, laminate and solder mask surface. In this case, the component lead pitch may become an issue. It can limit the degree of oversized printing since it tends to bridge between adjacent deposits. When paste bridging occurs prior to reflow, it can cause scavenging from adjacent pin deposits during reflow. The same issue is seen with standard fine pitch and can also occur with linked preforms. As the paste coalesces together, if one area reflows earlier than another, the paste may be drawn to that pad, resulting in different solder volumes in the joints. This can easily happen on standard reflow of fine pitch surface mount parts and is often the cause of solder shorts.

A variety of apertures may be used for PIHR to obtain the correct solder volume without shorting. Round, square and oblong are the

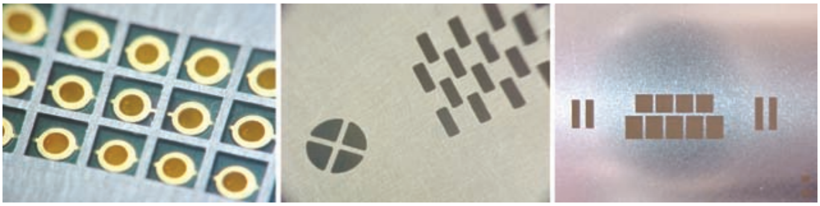


Figure 3-17: Examples of different stencil apertures aligned with the printed board prior to printing.

most common shapes, but triangles have also been suggested to meet the requirements of multiple row connectors. It is relatively easy to modify solder paste volume for components with two rows of leads. It becomes more challenging with three and four rows as the centre pins are restricted in terms of access.

Different aperture shape trials have shown that paste coalescing on solder resist can be improved with paste activity or nitrogen. The solder mask also has an impact, a fact that is highlighted again to make sure engineers specify the mask after confirming test results. As an example, a no-clean paste is better during reflow with a round aperture, rather than a square one. Solder at the corners of the deposit can separate and may form solder balls. This is not the case with water-soluble paste with greater activity. Using oblongs with rounded corners fares well with no-clean and increases volume over round deposits.

Wherever possible, use the stencil thickness to increase the paste volume first, then look at the paste area. But always consider the connector design on the base of the device. Not all connector suppliers are the same and if you don't designate a specific supplier, the difference in the connector housing can result in failure in your process. Also remember that your purchasing team is trying to save the company

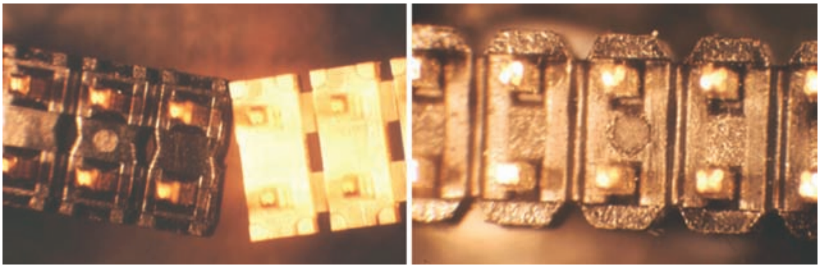


Figure 3-18: Three different IDC post header designs that may not be interchangeable in a process when a stencil design has been defined. If the process engineer knows in advance that there is the possibility of two suppliers, it may be possible for a different stencil to be designed. Like most people, the engineer doesn't have a tool kit that contains a crystal ball!

money, so make them aware of the significance of changing suppliers based solely on cost!

There are different choices in the stencil technology used today for fine pitch surface mount assembly. The choice of foil is dependent on your preference for your standard surface mount designs, unlike pin in paste assembly, where in most cases, the apertures for through-holes are massive compared to AμBGA. The most common stencil technology today uses either laser cut stainless steel or electroformed nickel. Both work well, provided that the stencil is designed correctly and the supplier produces the stencil correctly using suitable quality materials. After incorrect design, the most common cause of problems with stencils in manufacture is how we poorly handle these precision tools.

Some further details on the design of stencils for fine pitch technology and PIHR are provided in the IPC 7525A along with aperture calculations to achieve through-hole joints to meet other IPC assembly and soldering standards in line with IPC 610E. Alternatively refer to Chapter 10 -Suggested PIHR Resources. **PIHR**

"Often handling will be split between small passive and active components, with a dedicated placement system for fine pitch placement requiring greater accuracy."

CHAPTER 4

4

Automatic and Manual Component Placement

Traditional surface mount placement can be accomplished using a standard pick-and-place system. Both passive and active components will be placed by one machine in small and medium volume. In medium to high volume, two or more machines will be used for placement. Often handling will be split between small passive and active components, with a dedicated placement system for fine pitch placement requiring greater accuracy. Through-hole parts to be automatically assembled are inserted prior to or after surface mount placement.

In a PIHR process, through-hole components can be assembled by one of the following:

- Manual insertion
- Automatic odd form insertion
- Fine pitch surface mount placement

MANUAL INSERTION

Manual insertion of through-hole components into wet solder paste deposits is perfectly feasible and may be far more flexible and cost-effective than machine-based insertion. Manual placement is possible even on large connectors and sockets, with a little operator experience. It is possible to use some of the alignment methods discussed in Chapter 1 – Pin In Hole Reflow PCB Design Guidelines and Chapter 3 – Solder Paste Application Methods.

If the connectors are to be inserted manually on an in- or outfeed conveyor, it is useful to have a board support system in place. Often engineers have calculated the pin-to-hole ratio as tight as pos-

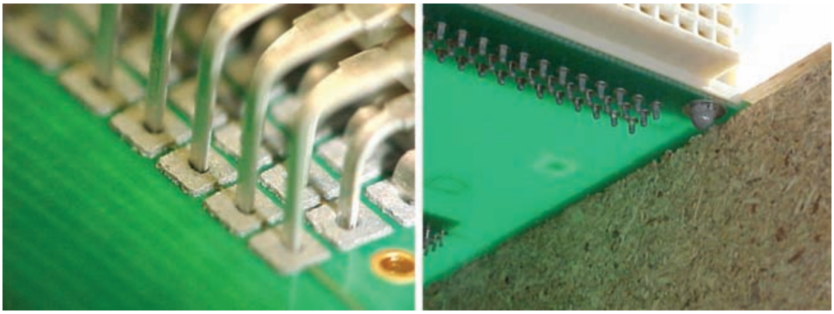


Figure 4-1: Manual insertion of connector after printing of solder paste (left). Protruding connector pins on the base of the printed board assembly prior to reflow (right). Paste is visible on the connector pins and the hold-down feature.

sible to aid in solder joint formation, which is a bad practice. Normally, the correct finished hole size specification from the PCB supplier for manual or automatic assembly is the maximum pin size plus 0.010". However, if engineers have not considered variation in hole size, pin size or errors in pin location, each can affect insertion force of the component, possibly causing displacement of surface mount parts.

A guide plate/support plate positioned over the board and conveyor during loading can also be beneficial. Operators can rest their hands on the plate and it prevents other components from being dislodged during insertion. The guide plate has cut-outs for each of the through-hole parts that aid in insertion alignment and location. A simple check for solder paste presence in the hole or over the pads can also be beneficial prior to part insertion. There will be no other opportunity for paste inspection after placement. A light box can be placed under the board location as the board is positioned against the conveyor stops. If paste is not present, light shines through the array of connector holes. This technique has been used for a number of years on single-sided and two-layer surface mount boards to inspect for shorts on fine pitch boards after reflow or wave soldering. A light placed under the board makes shorts easily detectable on flow line assembly after the soldering operation.

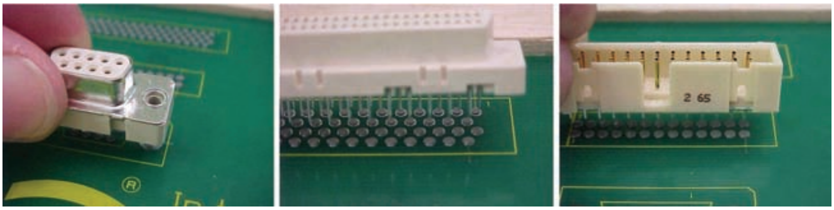


Figure 4-2: Operator manually placing three different connectors into a test board with no guide, except for the legend on the surface of the board.

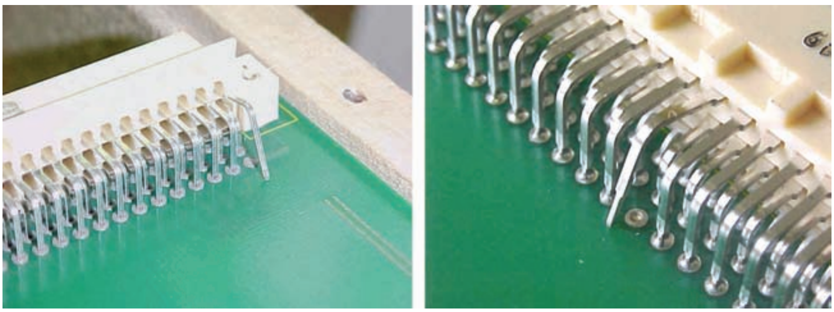


Figure 4-3: Incorrect insertion of pins (left). Connector after reflow with one misaligned pin (right).

Another benefit of the top plate guide or inspection plate is avoiding the possibility of displacement of other components that have already been placed on the surface of the paste, which is certainly a possibility with a hand, an operator's overall sleeve or the antistatic wrist strap.

Care needs to be taken to insert all of the connector pins into the plated though-holes. With experience, this is not difficult to perform reliably. However, if component pins are damaged or displaced due to poor handling or inadequate packaging, pins can be inserted incorrectly (Figure 4-3). Experience shows that misalignment of connector pins is far more likely to be caused by poor transport packaging or shop floor kitting prior to presentation to the assembly staff.

AUTOMATIC ODD FORM INSERTION

Although manual assembly is not difficult, many engineers have turned to automation for through-hole insertion in the PIHR process since it eliminates manual assembly issues. It is fair to say that most companies that adopted the assembly process in the early 1980s to 1990s were not able to justify automatic assembly and only used manual techniques. Even component suppliers needed to justify the use of SMT packaging with significant orders to cover the extra packaging costs or at least recoup the investment in the tooling costs.

Although vacuum pick-up tooling may be used on small parts, most automatic insertion systems for PIHR were robotics insertion systems, specifically designed for the purpose, with special clamp tooling. Examples of machine suppliers involved in odd form insertion were Universal Instruments and PMJ. The term odd form also became a common term associated with the parts that would be considered for through-hole reflow to differentiate them from surface mount.

Work undertaken by GPAX resulted in a tape-and-reel system being introduced for large through-hole components. The system has been adopted by companies like Universal Instruments, Quad and Panasonic. Prior to that time, plastic stick feed, tray and machined metal fixturing were commonly used with vibrator feeders. Traditional radial tape-mounted components can also be handled, but this requires a cutting/tape separation operation on the machine. Insertion speed of over 1,500 components per hour can be achieved. Again, the component design may make pick-up difficult, necessitating correct design for

Figure 4-4: Close-up view of component insertion with odd form insertion systems supplied by PMJ.

manufacture by the supplier and care during component specification by the user. If little thought is put into selection of the components, significant problems will occur in assembly.

Due to the size of many of the through-hole parts, there is a limit to the number of parts held on an odd form insertion system. In reality, there are very few large through-hole parts to insert on modern surface mount products, so machine capacity is not normally an issue. Most odd form assembly machines could develop insertion forces of 10kg, which is excessive force not required for this type of process and, if used, would displace other surface mount components.

FINE PITCH SURFACE MOUNT SYSTEMS

Thanks to the innovation of the majority of placement system suppliers, odd form assembly is not the technical challenge it was during the introduction of PIHR. The packaging of connectors or other through-hole parts should be compatible with automatic assembly, but this is still an issue. Packing must be designed to eliminate connector pins. Damaged existing packaging is mostly unsuitable. The packaging must present the part so that it can be easily picked up and inserted into the plated through-holes. The pin position is important for automatic insertion of the connector and it must not become damaged prior to insertion. Ideally, the hole size in the PCB will be just 0.010" to 0.015" larger than the maximum pin diameter and therefore requires an accurate positioning of the pin. Any less than 0.010" will make automatic insertion difficult. Today most placement systems have very effective vision systems to detect defective pins and position the connector for insertion.

There are three methods that are used for component feeding that are compatible with most modern surface mount placement systems:

- Dedicated component trays
- Tape-and-reel
- Solder support pallets

Dedicated Component Trays

Standard surface mount placement systems can select components from specially designed component trays, provided that there is the space and capability for standard fine pitch waffle trays. The dedicated tray is created for the through-hole connector by machining or injection moulding a sheet of material, then pre-mounting the components to the tray during kitting. The trays are then mounted to a placement system in a single position or mounted in a waffle tray feeder when different connectors are required for a single board design.

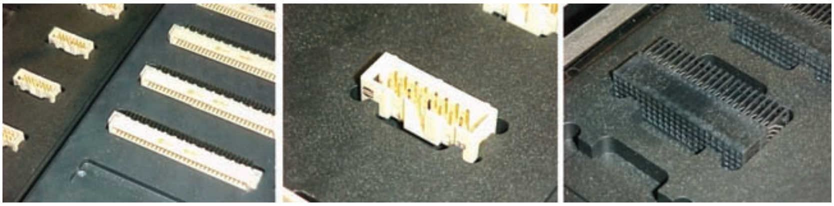


Figure 4-5: Dedicated trays produced for selected connector designs, featuring 16-, 96- and 110-pin devices with two, three and five rows of connections.

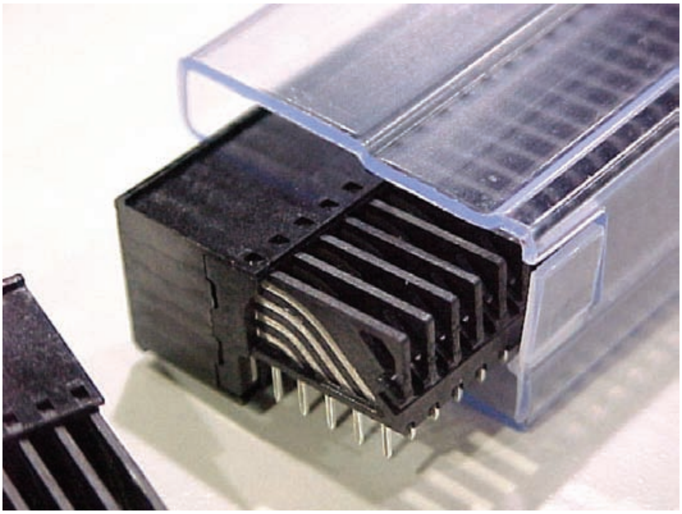


Figure 4-6: Components are available in tube feed for either vibratory or special indexing feeders.

Tape-and-Reel

Tape-and-reel is the most common format for volume applications when placing surface mount, depending on the size of the through-hole part. However, as the connectors or sockets get bigger, it is not suitable unless a special type is used. GPAX is a tape system most often used for odd form parts if standard tape-and-reel is not large enough to accommodate through-hole parts. The tape can also be used on pick-and-place systems if a feeder is available. GPAX has been offering dedicated feeders and tape systems for many years since the start of PIHR. The GPAX system is the most common method used on high volume odd form systems. It should be borne in mind that as the components get bigger, the number of components per reel gets smaller, and the space taken up on the placement table increases. This is not ideal if you are running out of feeder location space.

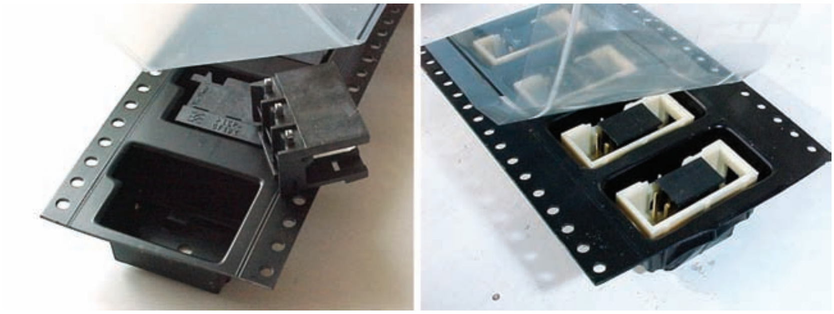


Figure 4-7: Standard tape-and-reel packaging examples provided by selected suppliers.

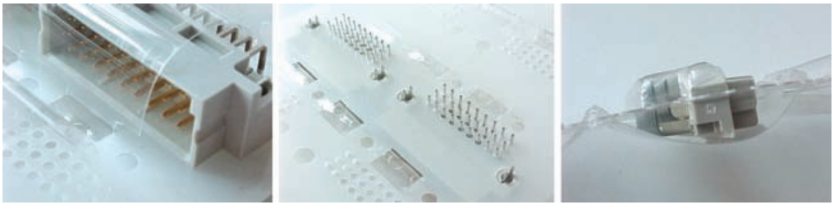


Figure 4-8: Connectors in GPAX tape-and-reel format.

If you are looking at odd form assembly, speak directly with your placement machine supplier who will now have a great deal of experience with placement of through-hole parts and will offer many different solutions for pick-up of the parts. There are basically two vacuum methods used with conventional surface mount and special gripper tools (Figures 4-9 and 4-10).

There are two visual methods of checking a connector for odd form automatic assembly: viewing the body outline and imaging the lead position. In the case of viewing the body outline, it is important that the leads are accurately positioned in relation to the body of the part. If the leads are deformed or damaged, the machine would still attempt to insert the part into the board and damage the parts and board. This was the position in the 1990s, during the introduction of automatic placement of odd form parts.

If vision is used to check the lead position, which is common today, it is the tip of the pin that becomes the key feature and nothing must affect its view. Most right angled connectors would be unsuitable for this process unless the vision system was specifically able to see the tips since there would be a reflection from the pin and the tip.

Mechanical alignment was used in the early days of odd form and on some placement systems. The part is picked and placed into

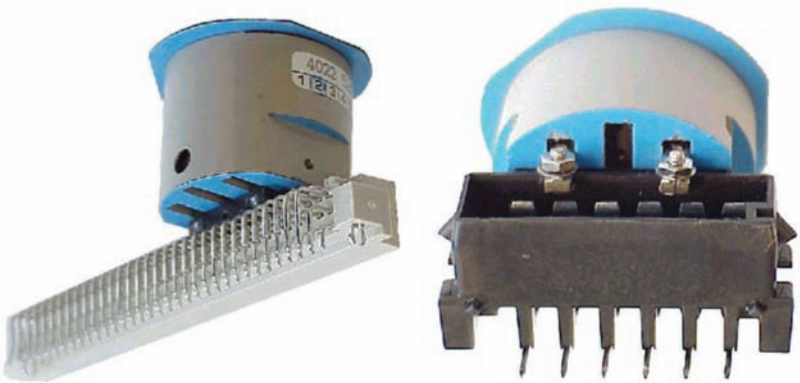


Figure 4-9: Close-up view of Philips/Assembleon odd form gripper tool being used with two different connector types.

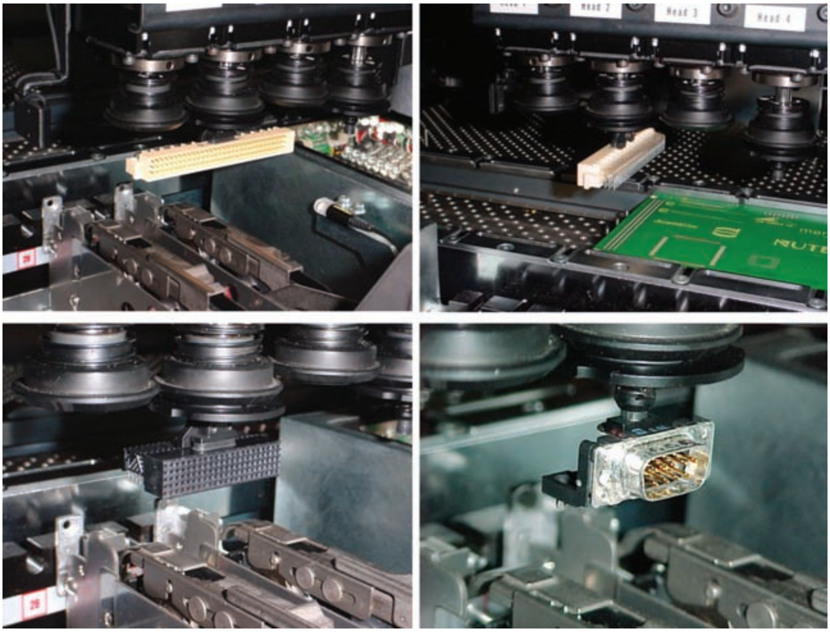


Figure 4-10: Standard vacuum pick-up tools being used with pick-and-place connectors during machine placement trials on the author's training board. In each case, the connectors are being placed on a Panasonic system.

Figure 4-11: Three different connectors that have been picked from their packaging and are being visually checked for alignment with the corresponding through-holes on the surface of the board. The connector image on the right is too long to be checked in one operation.

a machined locator plate that maintains an accurate pick-up position for insertion into the board. Mechanical alignment was time-consuming and created damage to parts and pick-up tools, but worked in practice.

If through-hole parts are inserted prior to insertion of the surface mount parts, the height of the conventional parts and proximity to surface mount parts need to be considered. Placement of the surface mount parts may have restricted access. The placement speed can also be affected since the height above the board may need to be adjusted to clear the conventional components. This is not a problem if the conventional parts are inserted after the surface mount parts are placed. On modern systems, this should not be an issue.

Solder Support Pallets

A solder support pallet is generally only required if connectors or other large components are to be assembled on the alternative side to surface mount components. In this case, a one-paste print process is conducted and all the surface components are automatically placed on the board surface after the through-hole connectors have been assembled. An alternative is to print all surface mount parts, then place the board in the pallet over the pre-mounted connectors. Paste can then be automatically dispensed over and around the projective pins.

Using a pallet, the leaded connectors are pre-located on the pallet and the board is lowered and located onto the pallet. The weight of the board and subsequent components guarantees that the connectors are flush with the base of the board. Also, the connector base design is less critical since it will not directly contact the paste deposit. The pins then pass through the through-holes with the paste around the tip on the pin. All other surface mount parts are automatically placed before the pallet passes through the reflow oven for soldering. This technique provides some protection to the connector in terms of peak temperature. It is possible to have the large parts pre-loaded on the pallet, with the board passing the pallet through the placement system. This may help avoid problems associated with manual location of parts and misplacement of surface mount components prior to reflow.

To aid in alignment of the board to the pallet, it is advisable to have some tapered tooling pins inserted in the pallet. The tapered tooling pins should align the board before the connector pins contact the base of the board. This will facilitate printed board loading and the pins may be left in place or removed before proceeding. The connector pin length is critical for PIHR. Connectors with a large amount of pin float can be a real problem for lead location and should be avoided. Some active components with built-in filters or capacitors can have a wide pin location tolerance.

Care must be taken during reflow oven profiling, since a pallet will always have some effect on the circuit board temperature, possibly delaying reflow. When profiling, the peak temperature at the base of the connector should be checked to ensure complete reflow of the pin and solder pad in this area. This type of build with a pallet would not be suitable for vapour phase soldering (VPS) due to the level of solvent trap in the pallet, which would make draining difficult and increase manufacturing costs.

Design engineers will realize a spinoff benefit of this type of design and method of build. Normally, the benefit of this type of board, with surface mount and a large number of connectors, perhaps for a back plane application, is the ease of layout. If the product is to be reflowed as opposed to wave soldered, the layout is simplified. During wave soldering, there are many more restrictions on design in terms of parts that can be used and component orientation.

ODD FORM COMPONENT CHECK LIST

As a simple reference, Figure 4-12 shows a connector and a list of the points to consider with any part for a new PCB design. Each point should be considered when looking at your placement system's capability or the suitability of the connector for automatic placement. The simplest approach is to show your placement technicians or your supplier since they will have the most practical experience. This should be standard practice during the initial New Product Introduction (NPI) initial design concept review, not just prior to production!

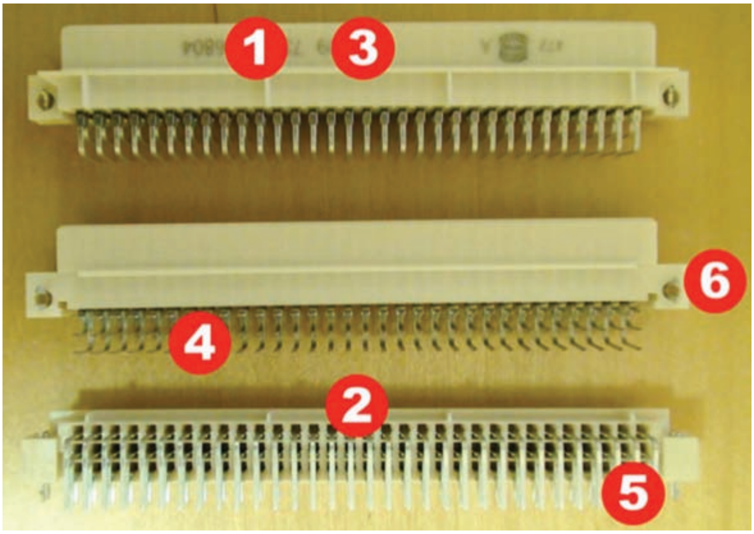


Figure 4-12:

1. Centre position for pick-up must be smooth, flat and rigid for vacuum tools. Ideally, there should be no logos, markings, cut-outs or plastic dimples.

2. Pick-up position will ideally be the centre of gravity for the component.

3. For ease of insertion, the pick-up position should be above the pins and hold-down features.

4. Check the shape and dimensions of the pin tips for automatic visual recognition. Look for good contrast between the pins and the non-reflective background or body.

5. The pins should all be one height and held coplanar with the surface for placement.

6. Hold-down features should be avoided. If required, make sure the hole in the PCB is the correct size for placement, not the size specified by the connector supplier for standard use.

In addition to the checkpoints illustrated in Figure 4-12, check the component weight and vacuum tool capability at the placement speed required for production. Remember that connector packaging must be suitable for accurate and repeatable pick-up.

Select the correct pick-up tool for the connector mass, weight and shape.

Check that alternative suppliers also meet these requirements before purchasing order parts for production to make cost savings! **PIHR**

5

> "There are a number of solder alloy options when selecting lead-free solders, some of which can be processed below the reflow temperature of common tin/lead solders."

CHAPTER 5

Reflow Soldering

Through-hole reflow can be successfully conducted by convection or vapour phase reflow with either tin/lead or lead-free alloys. Since many products today are designed for double-sided reflow, theoretically through-hole components can be soldered on one or both sides of a through-hole design. There are a number of solder alloy options when selecting lead-free solders, some of which can be processed below the reflow temperature of common tin/lead solders. Although most through-hole parts have a higher mass than other surface mount components using modern reflow processes, the change in temperature (ΔT) between small and large body components can still be kept to a minimum.

VAPOUR PHASE REFLOW

Vapour Phase Soldering (VPS), also referred to as condensation soldering, has been around in the industry for many years and was one of only two serious options during the early introduction of Surface Mount Technology (SMT). In the early days, SMT engineers had the option of brown/black belt conduction for single-sided products, infrared or VPS. The first reflow system used by many engineers was VPS because of the simplicity of the process and process set-up that reduces the problems of accurate loading, board conveyor positioning, profiling and overheating of flux residues and components.

A well-designed board provides higher soldering yields. A poor design, particularly on passive components, amplifies the number of lifted and tombstoned components. All vapour phase systems can show a difference in component lift due to the fundamental nature of the process. As the vapour condenses on the surface of the board and

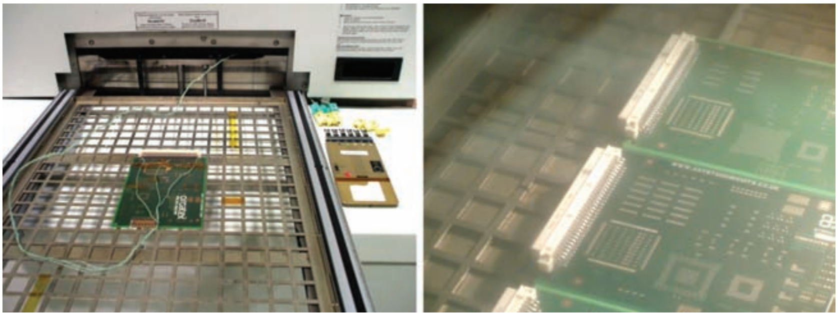


Figure 5-1: PIHR demonstration board being reflow soldered in VPS. Board used for profiling with an ECD M.O.L.E.® (left). Three boards during reflow (right).

turns to liquid, component movement can occur. Is this a reason to dismiss VPS? No. It is often just an excuse to stick with the poor design. *[Author Note: A lead-free trial run comparing VPS with convection showed an increased number of lifting defects with VPS. If you only looked at the total PPM levels, you might say that VPS was the cause, but the defects were all on one component size. This illustrates the need for good documentation during design reviews and New Product Introduction (NPI) at contract assembly companies.]*

One lead-free process defect that has been apparent with a number of lead-free trials on 0201 chip resistors, capacitors and µBGA, but is not seen with VPS, is incomplete reflow. When soldering 0201 chips on boards with a wide range of other components, including QFP, BGA and through-hole connectors, there can be a large ΔT. To optimise the process and decrease ΔT, the surface of the board, including the 0201 paste deposits, will often be at elevated temperatures, just below reflow, for long periods. This can cause flux exhaustion prior to reflow, leaving incomplete coalescence of all the solder balls (Refer to Chapter 9 - PIHR Manufacturing Defects - Causes and Cures for images of this situation.) The same effect can be seen when incorrect paste and printing parameters print a minimum paste volume, resulting in a lack of flux vehicle. After reflow, the result is what engineers refer to as graping. *[Author Note: I refer to this as warting.]* With modern VPS, this does not occur, but it did happen with older VPS processes when a secondary vapour layer was used and the condensing secondary fluid caused the flux to be separated from the paste.

When we consider PIHR specifically, there are a few potential issues seen with VPS, like trapping of solvent either in or under sockets or connectors, which is possible and has been seen on open and vacuum systems. The material loss is minimal, but difficult to estimate without running medium- to high-volume trials. It is true to say that any engineer running VPS may note solvent leakage from under area array or low stand-off packages or on thin PCBs mounted on a support carrier plate for soldering.

In convection reflow systems, solder paste can be seen dropping off the tips of pins. Paste that has been displaced on the tips of the pins can fall off before it reflows. The same is true with VPS. As the vapour condenses on the surface of the pins, paste can drop into the sump or onto the carrier, which reduces the solder volume to form the joint and can be random in nature. The same is true when the connector is mounted on the board with the pins facing up. The paste can separate from the pins and drop onto the board surface, possibly resulting in solder balls around the pins.

There is no evidence that VPS soldering provides a better wetting performance with tin/lead and lead-free alloys and on different solder

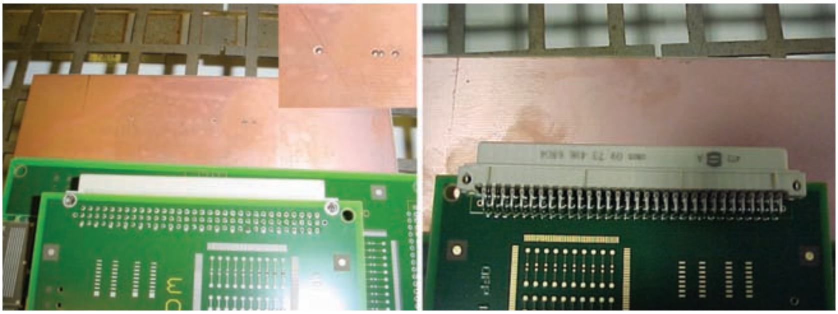


Figure 5-2: Copper foil being positioned under connectors during VPS to look for possible paste drop off. Paste would be very easily detected on the copper surface after reflow soldering (left). No drop-off on the copper after soldering (right).

finishes when compared to convection in air. However, the results can become blurred when compared with nitrogen convection. It is often stated that VPS soldering is an inert soldering process, which is true when the board assembly enters the vapour layer. Prior to that, or when the board in some designs is not in a vapour during cooling, this will not be the case.

As a process, VPS has always been an option in the industry and there will be new engineers who select the process. There will also be existing engineers who will move back from convection to VPS as a solution for lead-free. Some machines also incorporate a vacuum chamber system for VPS, which is used to reduce void formation and fluid loss.

In the original batch vapour phase system, the primary fluid was boiled and generated a vapour layer. In order to contain this vapour within the size constraints of the holding tank, it was necessary to cover the vapour to prevent it from escaping. This was done with a secondary fluid that also produced a vapour layer that boiled at a lower temperature. The condensing coils used cold water running through at a very low flow rate to recondense the vapour. The fluid then circulated back through filters. Above the primary coil, another coil was used to reduce the loss of the secondary blanket of vapour sitting on the top of the primary fluid.

In operation, a basket/carrier loaded with assembled boards would be lowered through the secondary layer and into the primary. After the work reached reflow temperature, the basket would be raised. On the way up, it was allowed to dwell in the secondary blanket in order to contain primary vapour and fluid. Ideally, the boards or basket would be slightly angled to allow condensed fluid to run off. The basket would then be raised out of the machine and allowed to cool. Prior to this operation, the assembled boards had to go through a preheating operation, normally using a separate oven, to reduce the possibility of thermal shock and the amount of volatile material in the solder paste. Today, preheat is incorporated into the VPS systems to more tightly control the profile of the board assemblies. Both batch and in-line sys-

tems are available today and, although the fundamentals of the process remain the same, there are many innovative features that make modern VPS cost-effective and reliable for small- and medium-volume users.

The following is a listing of some materials that were originally available for use in vapour phase units.

Manufacturer	**Liquid type**	**Boiling point**
3M	FC70	215°C
3M	FC5311	218°C
Montedison	Galden LS	228°C
Montedison	Galden HS	255°C

Today, the supplies are limited and the primary material for lead-free may be either 230°C or 240°C Galden material available in the UK from Solder Connection or worldwide from Solvay Solexis.

Using more than one vapour phase system, it is feasible to carry out sequential soldering processes at different temperatures. This may be a benefit to some companies and for non-PCB applications. The liquids required cost in the region £70 to £80 per kilo (approximately $110 to $125). Due to the cost, it is imperative that the machine used with these fluids be designed to minimise vapour or fluid loss from the system. Today, a typical batch process is quoted with a running cost of between £2 to £3 per hour (approximately $3.15 to $4.75). This means that the PCB design should not have any fluid traps that could carry this extremely expensive fluid out of the system. Remember that the only reason for the secondary layer on older machines was as a "sacrificial blanket" to prevent loss of the primary vapour. In the batch and in-line vapour phase systems of today, systems do not have a secondary vapour blank.

From a profiling point of view, the goal is to solder joints with the minimum ΔT across the board and at the lowest peak temperature. It is important to minimise the time joints are in a liquid state and control the speed of temperature rise to reflow. Gone are concerns of board damage due to excessive temperatures. Care needs to be taken on fixturing of small or light boards on carriers. If they are lost into the boiling sump, there is no way to retrieve them until the fluid cools. In the case of VPS designs that employ a heated plate for vaporising the fluid, as opposed to the traditional sump systems, retrieval may be easier, but is still not recommended.

There is a difference in the ΔT on the board surface and under components, but it is very small at peak temperatures, provided that time is allowed for the profiles to converge. The initial temperature rise through preheat does have recordable differences, just like convection, but the differences can be smaller depending on the machine design and the care taken by the process engineer when profiling the systems.

Cooling can cause a delay in the process, just like standard convection reflow. However, new developments in liquid cooling may benefit the industry and provide a solution to fillet/pad lifting that can occur on lead-free through-hole joints. Through-hole fillet lifting is probably the only new process defect associated with lead-free. It does not seem to affect reliability, but it is difficult for quality staff to accept its existence.

CONVECTION REFLOW

During reflow soldering, a solder joint is formed between a component termination and the pad on the surface of a printed board. In the case of a through-hole, the joint is also formed with the through-hole plating. During process setup, it is important to consider that a high percentage of the solder paste is inside the plated through-hole during the soldering process. Today, terminations on a component or connector are normally tin- or gold-plated, providing an easily solderable surface finish.

Most chip terminations are metallised silver palladium with a nickel layer and tin on the surface. A leaded part will either be copper, steel, kovar or brass, with a barrier layer of copper or nickel and a final tin coating. The most popular surface finishes for printed boards are gold over nickel, solder levelling, copper (OSP), silver and tin. Alternative solderable coatings have been introduced to resolve the problem

Figure 5-3: BTU Convection reflow system used on the SMART Group Lead-Free Experience technology feature and typical of the systems used in medium- and high-volume production.

of uneven surface coatings seen with poorly solder levelled circuits.

To reflow a board, solder paste is applied to each of the pads on which a component is to be soldered and forced into selected plated through-holes. Components are then placed into the paste surface to assist in wetting the side of the terminations and through-hole leads pushed into through-holes. The paste provides some adhesive qualities during transportation of the board to prevent surface mount components from being lost prior to soldering. As the paste is reflowed, it changes into a liquid state. A reaction takes place between the tin and the surfaces being soldered. The tin from the paste and the copper or nickel on the lead or circuit pads form another alloy referred to as an intermetallic. The new alloy is part of the joint that forms during any soldering operation and must form if a reliable joint is to be produced.

The effectiveness of the reaction depends on the solderability of the surfaces being joined. To make the process feasible, a fluxing agent is incorporated into a paste to clean the lead, pad and solder particles that make up the paste. When the solder is in a liquid state, there should be nothing to prevent the solder reaction from taking place. It is possible to use low-activity or high-activity fluxes in solder paste formulations, but that depends on if cleaning is to be conducted after assembly.

Controlling the temperature, preheat, atmosphere and solderability of all of the materials being joined is the job of the process engineer when setting up the convection reflow process.

CONVEYOR SYSTEMS

There are two types of conveyors for transporting board assemblies through the reflow process. The type of conveyor system selected by a manufacturing engineer depends on the number of boards to process per hour, the number of different board widths and the need for single-sided or double-sided reflow soldering.

The choice of conveyor affects the price of the machine and its ability to work as part of an in-line process. There are two options from most manufacturers: a pin or belt conveyor or a combination conveyor system, normally referred to as combo, with both belt and pin transport. The original designs produced some years back were rumoured to be produced so that if the board fell off the pins, it would fall onto the mesh belt and not be lost.

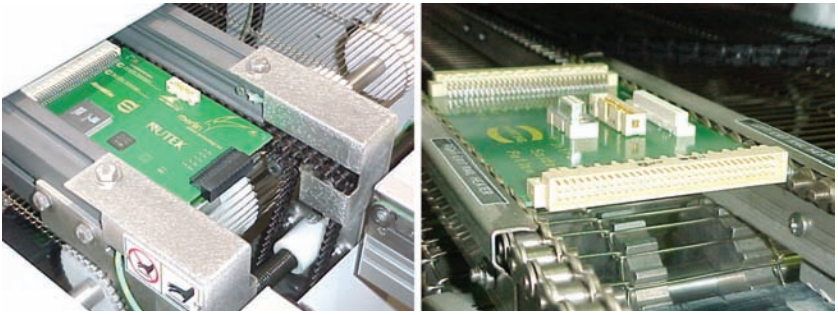


Figure 5-4: PIHR test board running through an Electrovert reflow oven on pins. The system is fitted with a combination of pins and a mesh conveyor belt.

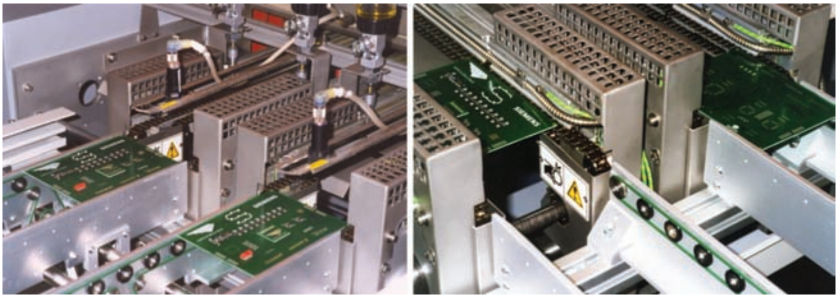


Figure 5-5: Dual lane production line set up by the author with Speedline Technology and Siemens. Boards entering the reflow oven (left) and exiting the soldering process prior to inspection (right).

The industry trend in the 1990s was for two pin conveyors to be incorporated into a single reflow oven. The dual lane process is particularly interesting for high-volume companies with limited factory space or a limited mixture of board widths. It also lent itself to companies producing one board width in high volume and a variety of other widths in smaller volumes. The smaller volumes could be processed down one lane and the high volumes down the other fixed lane. The only process parameter that could be varied independently was the speed of the conveyor since it is not possible to change the zone setting for each conveyor. This was a bit of a challenge for the process engineer with two profiles to juggle. Fortunately, with this technology, standards have been defined so that reflow system communications are compatible with placement equipment and in/out feed conveyors.

PIN TRANSPORT

Pin transfer has become the most popular conveyor because it is more versatile for both in-line and batch production. Using a linked mesh system would be fine for batch production, but not ideal for in-line operations. A linked mesh system has constraints for boards that are double-sided and PIHR, where the paste will be displaced from pins.

The pin transfer system provides the best solution for in-line production. The most common type is a linked chain with 5mm pins pro-

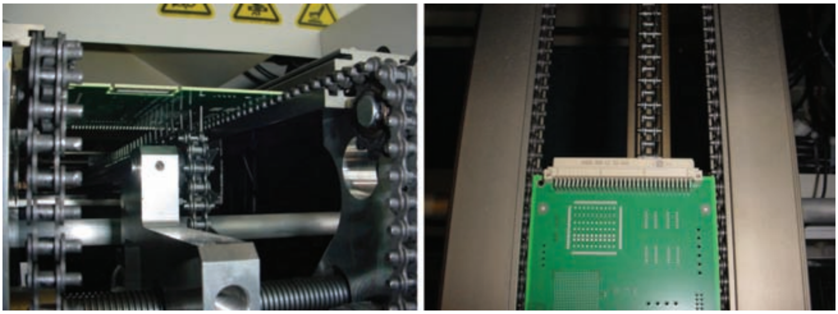


Figure 5-6: It is possible to still use centre board support on boards that are correctly designed or on thin multi-panels to provide support. The images show a board being supported with pin support on a BTU chain mechanism. Care should be taken in positioning the assembly to avoid contact with components on the base of the board or with the pins on through-hole connectors.

truding to support the edge of the board and the board just sits on the surface of the pins for transportation through the process. Alternative edge-clamping systems are offered by two reflow suppliers. In edge-clamping systems, the board enters the reflow oven and the edges are clamped on each side, limiting warping and twisting of the board as it passes through the reflow process. Thin boards of less than 1mm or standard 1.6mm boards that have copper balance problems can benefit from this type of conveyor. The board expansion in the x and y direction is not restricted, The clamping technique utilises the normal 5mm clearance designed into the edge of all good designs. The clamping design can also reduce sag of the board during soldering.

Some reflow oven suppliers have noted that the conveyor section heat sinks some heat from the board, just like a board placed into a soldering pallet drops the temperature by 5°C to 10°C. To compensate for this, heaters have been built into the conveyor section in specific locations.

LINKED CHAIN

The mesh belt has been used for many years in the industry for adhesive cure and single-sided reflow applications. It was also used for reflow in the printed board industry for fusing tin/lead. The belt provides support for the whole board as it passes through the process and reduces the possibility of warping. The weight of the board holds it flat as it passes over the glass transition temperature of the base laminate. Mesh or linked chains are not suitable for double-sided reflow or PIHR.

MACHINE ZONES

The number of zones in an oven is dependent on the throughput requirement of the process. Selecting a seven-zone over a ten-zone system provides the engineer with more control of the process, but at the speed required for the number of boards to be processed per hour.

An oven normally includes preheat, soak, reflow and cooling zones, just like the stages of a temperature profile. There may be one or two preheat, two soak and one reflow zone with a forced air cooling section. More substantial systems with seven or more zones incorporate more reflow and dedicated forced cooling zones.

Preheat

When the printed board loaded with components first enters the reflow oven, it is at ambient temperature, around 18°C to 20°C. Initially, the preheat section will increase the temperature of the assembly. The rate of temperature rise will depend on the oven settings and the thermal demand of the printed board assembly. Some component suppliers limit the component specifications to a rise temperature of 3°C to 4°C per second.

During preheat, any volatile material in the paste can be driven off and the initial cleaning action of the flux contained in the paste can start. Some materials do not activate until they reach substantially higher temperatures. This is dependent on the material and it is important to seek advice from the supplier. During preheat, every effort is made not to create too much of a temperature differential on the board surface since it may be difficult for the differential to be overcome during the soak period. PIHR creates a larger range of temperatures on the board assembly. In most cases, the through-hole part around the pins/connector will be the coldest part of the board. A temperature rise in the initial preheat zone can be between 80°C and 150°C.

Soak

The soak period allows all temperatures on the surface of the board to normalise. It is inevitable that some termination areas on the surface of the board will heat up more quickly than others during preheat. With the greater sophistication of reflow technology, the surface of the board can see limited temperature variations. But at the component termination interface, temperatures will vary, particularly under connectors or area array components. Under a plastic ball grid array, the differential temperature can be 5°C to 10°C. Under a through-hole connector body, the temperature can be 10°C to 15°C higher than the surface of the circuit board.

Reflow

The reflow zone is where the board temperature moves over the liquid temperature of the solder paste. The aim of the reflow zone is to allow reflow in the shortest time possible to produce reliable joints, normally within 30 seconds, but it can be up to 60 seconds. The time is dependent on the temperature differential of the board and joints entering the reflow zone. The job of the process engineer is to reduce the difference in ΔT during preheat and soak prior to entering reflow. Otherwise, the peak temperature will always be harder to compensate.

Cooling

When all of the solder joints have been formed and the board moves out of the heating zones, the board will begin to cool naturally and the solder will solidify. To speed up cooling, blowers are used to blow ambient air. In recent years, air has been substituted with nitrogen and refrigeration systems have been introduced to speed up the cooling cycle.

Some engineers suggest that you cannot conduct lead-free reflow reliably in a reflow oven with fewer than five zones, but that is not true. If the board assembly can be reflowed with the minimal differential temperature across the board, not subjecting the components to a thermal gradient outside their specification and have the joints in a liquid state for a period of less than 30 seconds, any number of zones is acceptable. The limiting factor may be the throughput speed. Single-zone reflow processes are sold on table top batch ovens. Tower reflow systems were effectively single-zone systems. Two- and three-zone ovens are capable of providing reliable reflow on double-sided assemblies, BGA and PIHR, albeit with reduced throughput speed compared to a larger oven. However, the limited number of zones reduces the engineer's flexibility when profiling a complex board design.

Machine Extraction

During reflow, there will be fumes coming from the paste, the printed board substrate and any masking on the surface of the board. This may or may not be harmful, but it is certainly unpleasant and must be extracted. The extraction rate for any process is dictated by the supplier in the machine specifications and should be followed. Care should be taken that the extraction rate does not vary or deteriorate over time since it can affect the temperatures in an oven. If any maintenance is done on the extraction, profiles should be run to confirm that the work has not caused any changes in the process.

Extraction ports are positioned during the preheat and reflow zones, or just before, in the final soak zone. Initially, vapours from the paste are seen in the preheat phase. Additional fumes are seen just before reflow of the paste from any material coming from the laminate or the solder mask. Some laminates will give off fumes between 80°C and 140°C. There may be specific issues, like laminates that give off different vapours that can condense out and drip down from exhaust ports. An example of this is the phenolic resin associated with paper-based substrates used during wave soldering in nitrogen. The phenolic resin causes a problem for the extraction in wave soldering, but these laminates are not normally used in a reflow process, even with low-temperature solder alloys.

Fumes from the surface of boards after reflow can impact other down-line processes. In high volume, some companies have reported problems and downtime on Automatic Optical Inspection (AOI) due to contamination caused by vapours given off from the surface of hot boards as they pass through the inspection stage and condense on the optic systems. *[Author Note: This may be possible when there are high-level volatiles. I am investigating this further.]*

Cooling

Cooling systems vary from basic fans that blow ambient air onto the surface of the board to assisted cooling. There are high-volume convection systems that increase the cooling rate or nitrogen systems that cool in an inert atmosphere. There are a number of practical reasons for a board assembly to be cooled before it exits the oven. Manual or automatic handling may be an issue due to the residual heat in the board. Operators would need to wear gloves for manual unloading. The solder must not be in a liquid state as the board exits the oven and moves onto a conveyor since this could cause SMT part displacement. If the board is being fed directly into a second paste printing or dispensing operation, a warm board will affect the paste, possibly causing it to slump on fine pitch areas.

Cooling the joint quickly after reflow can reduce the intermetallic at the joint interface and surfaces yet to be soldered. The thicker the intermetallic, the weaker this area can be. The reflow period should be less than 30 seconds in a liquid state. Maintaining the board at high temperatures for a long period of time will degrade the solderability of the PCB pads and terminations. Cooling the surface quickly to below 50°C is beneficial, particularly for copper boards coated with Organic Solderable Protectors (OSP). Through-hole boards that are going to be subjected to a final soldering operation may experience problems with hole fill due to the changes in wetting.

Nitrogen

Nitrogen use has become popular over the last few years, particularly when using low-residue solder pastes and copper OSP circuit boards. Nitrogen is used to displace the oxygen and open up the process window during reflow soldering. Oxygen levels in reflow soldering at or below 1,000 ppm can improve soldering performance and can be economically viable if copper boards are being used. Copper boards with OSP are cheaper than gold, silver or solder levelled, and the cost savings can offset the cost of using nitrogen, making the quality and process improvements obtained with the use of nitrogen free, or nearly free. Using nitrogen allows ultra-low residue pastes to be used in lead-free, which improves in-circuit test probing. Reducing the residues on through-hole pins can facilitate direct contact with PIHR contacts and reduce test probe cleaning.

The SMART Group Lead-Free Experience Report listed in the Chapter 10 - Suggested PIHR Resources covers the use of convection and vapour phase reflow soldering on difference printed board finishes and different pastes. It includes parameters used and inspection results obtained with x-ray and microsections for surface mount and through-hole reflow. **PIHR**

95

"Peak temperature and time at peak temperature should be investigated closely due to the differing requirements of conventional and surface mount components."

CHAPTER 6

6

Temperature Profiling for Pin In Paste Reflow

INTRODUCTION

In the case of through-hole components, there has been little change in the basic philosophy of correctly profiling a board for production. However, there may be a considerable change depending on your current practice. In the case of connectors, sockets, transformers and other through-hole parts, it is the temperature under the parts that may be the significant factor. Like Ball Grid Array (BGA) and other area array packages, the component body will mask the solder joints to some degree, which is why it is necessary to monitor temperature under the parts during profiling. Through-hole parts will most commonly be reflowed when on the top of the board. This chapter includes a guide to setting up a reflow oven for production and monitoring changes based on process trials.

Peak temperature and time at peak temperature should be investigated closely due to the differing requirements of conventional and surface mount components. To aid in setting successful process parameters, profiling equipment like the ECD M.O.L.E.®, KIC or SolderStar profilers should be used with the maximum number of thermocouples available to accurately monitor process conditions. Four probes may be used for normal surface mount parts, with a minimum of two used specifically for through-hole parts. Separate experiments may be conducted in the initial setup for large connectors with additional monitoring positions.

Let's consider a large connector with three rows of pins (Type C DIN Connector) illustrated in Chapter 2 - PIHR Component Guidelines and shown below. If the connector is mounted on the edge of the board

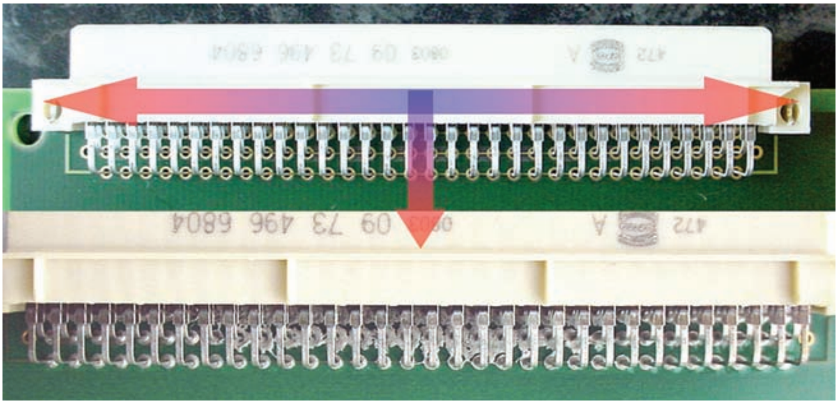


Figure 6-1: Connectors with three rows of pins.

and spans the complete width of the assembly, approximately 90mm, we would ideally like to know the temperatures during reflow in a minimum of three positions.

The connectors with three rows of pins shown in figure 6-1 can be thermally demanding during intrusive reflow, hence the need for correct profiling. Theoretically, the connector body and pins have different thermal demands, indicated by the range of temperatures illustrated on the arrows from red to blue. Pins close to the body of a connector and at the centre of the body will be the last to reflow in

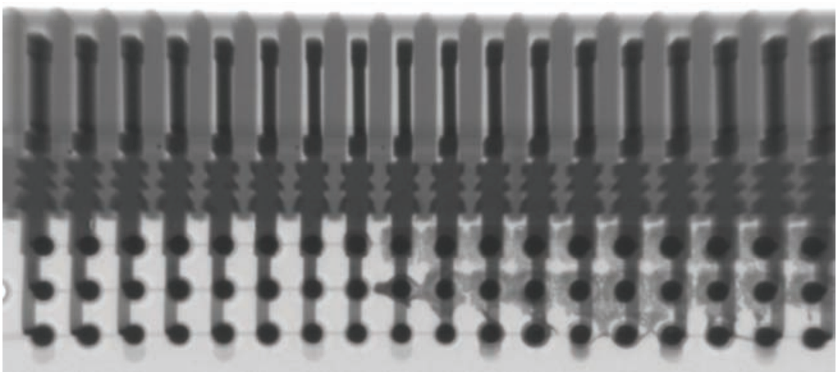


Figure 6-2: Reflow results shown on x-ray images prior to optimising the profile for the connector.

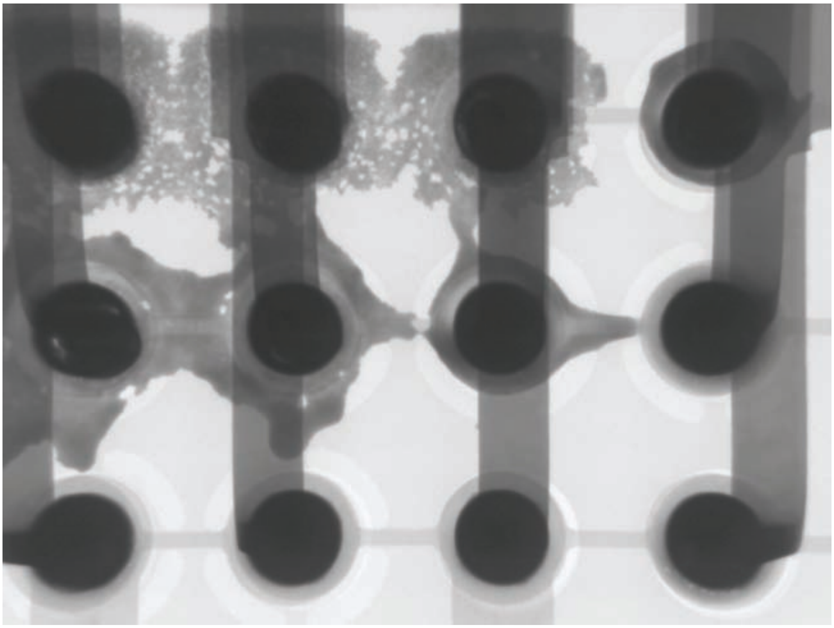


Figure 6-3: Close-up x-ray image taken around the centre section of the connector shows incomplete reflow of the solder paste on the surface of the board. Through-holes on the bottom line have filled.

a tin/lead or lead-free process, which is perfectly illustrated with the incomplete or non-reflow of paste in these areas shown on the bottom connector in figure 6-1.

The x-ray images in figure 6-2 show evidence of incomplete reflow of the solder paste in the centre of the connector and on the inner row of pins. It is a great example of incomplete reflow of paste on this type of connector design. The image clearly shows differences in reflow temperature, with the temperature seen by the board assembly decreasing from left to right. The x-ray image of the connector and incomplete reflow of the paste indicates incorrect profiling procedures used on the product.

There are two reflow soldering processes: convection with air or convection with nitrogen. Standard convection in air is the dominant process used in the marketplace. Vapour Phase Soldering (VPS) has come back into favor due to the introduction of lead-free soldering.

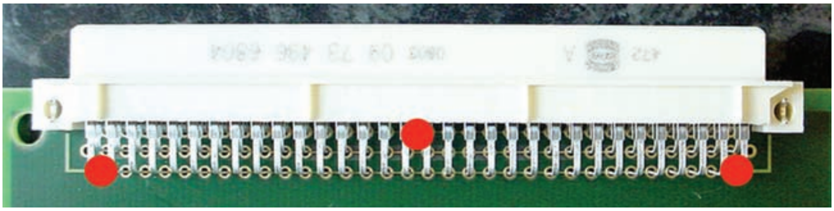


Figure 6-4: Connector showing monitoring of three locations with thermocouples mounted in the positions indicated by the red dots. This should provide the change in temperature (ΔT) for the joints on the connector.

Using a fluid that boils at 230°C, it is feasible to successfully reflow both tin/lead and lead-free assemblies using a Sn/Ag/Cu (SAC) alloy in one VPS system. This is an ideal situation for many small- and medium-volume companies with minimum engineering support. It is also a good option for small and medium contract assembly companies.

It may seem that with VPS, you can't overheat an assembly. So why bother profiling a board? Isn't this just lazy engineering, since the board assembly will still have a ΔT during preheat and cooling? Depending on mass, the board will still take time to get to reflow temperature, particularly if the mass is greater than the vapour capacity. It is fair to say that profiling a VPS system is more difficult based on different system design and vapour locks. Fortunately, some suppliers have correctly designed their systems to be compatible with external profiling equipment, so all reflow data can be collected and retained on a single system.

TEMPERATURE-RELATED ISSUES

Let's take a look at a typical connector previously shown in this chapter (see Figure 6-1) and discuss the temperature-related issues that should be considered by process engineers.

Temperature on Top of the Connector Body

Depending on the performance of the convection system, the top surfaces and edges of the connector can experience higher temperatures. This can be an issue on older reflow systems. Since the surface being measured is away from the bulk of the board, the difference in actual mass being heated results in higher temperatures. This can result in cracking at high stress points in the moulding or blistering on surfaces due to the moisture content in the plastic.

Temperature on Connector Pin

The pin protrudes from the base of the board and may have paste on the tip after insertion. The pin is generally the first surface to reach reflow temperature, which is why you can see incomplete reflow (graping) or a solder ball on the tip of the pin if the process is not correctly set up. Having a small mass that is away from the board surface allows this point to increase in temperature faster.

The difference in temperature is the reason for the solder ball on the tip and incomplete reflow in the hole. The reasons for incomplete reflow on the surface of the pin and reflow in the hole are the same as the reasons for surface graping. *[Author Note: I refer to this condition as spots or warts.]* The paste has been allowed to reach a temperature just below reflow and remain there for a long period of time, causing complete exhaustion of the solder paste flux. The resulting graping is due to the impact of a very long soak profile on a no-clean paste.

Temperature Along the Connector

On long connectors, there will be a difference in temperature between the centre pin and the end contacts. On a three-row connector the outer rows will reach reflow before the centre row. On the connector in Figure 6-1, if you profile the pin contacts close to the body of the connector, the temperature will be lower, lagging behind the row of pins on the rows farthest away from the connector body.

SETTING UP REFLOW PROFILES

Reflow soldering is a relatively simple process. Solder, in the form of solder paste, is heated along with component and printed circuit terminations. Depending on the alloy, the solder paste particles become a liquid at either 179°C to 184°C for tin/lead or 217°C to 227°C for lead-free alloys. When the solder is in the liquid state, a solder joint will form between the two surfaces, provided that both surfaces are solderable.

The speed of wetting depends on the lead and PCB coating and the solderability of that coating. It is necessary to heat up and cool down the assembly in a controlled manner. It is also necessary to maintain the solder joints in a liquid state to eliminate voiding and form a true intermetallic bond with the base materials. *[Author Note: Further details on reflow and profiling are contained in my CD-ROM, "Introducing Pin In Hole/Intrusive Reflow Soldering."]*

Board Support

The printed circuit board should remain as flat as possible throughout the first and second soldering operation. The peak temperature and maximum duration at peak temperature of any component should not be exceeded. Many parts have a peak temperature limit of 220°C for tin/lead assembly and 250°C for lead-free processes. Most high-temperature connectors are rated at 260°C.

With modern convection ovens, the convected air or nitrogen should not disturb components or cause the printed board to flex. Ideally, all reflow ovens should be able to adjust the level of convection rates to minimise component movement, which can occur with thin boards of 1mm or less that need to be supported. If the board warps and the through-hole connector stays perfectly flat, the gap will be obvious in quality inspections. With long connectors, the same problem has been seen in the past on wave and selective soldering, where the connector does not sit flat on the surface of the board.

Any board support should be fully adjusted to meet the board requirements. It should need the minimum clear area on the base of the board. Variation on conveyor width should be checked on entry, exit and in the centre of the oven, as well as when cold and at operating temperatures to make sure that distortion of the board is not due to conveyor pinching.

Machine Parameters

Initially the temperatures of the separate zones are based on an existing profile for a similar board design. The speed of the conveyor is adjusted to the desired assembly throughput. This may be limited by the type and length of the unit.

Profiles and Thermocouples

Thermocouples are fixed to the printed board surface and the component terminations; ideally, directly in contact with the pad surface. If they are placed on the top of the terminations, it may affect the readings. Ideally, high-temperature solder alloy is used to fix thermocouple beads to solderable areas. With intrusive reflow, the thermocouple beads can be positioned in the top of the plated through-holes.

The second method of choice is aluminum tape, which has been demonstrated to give the second best process performance and repeatability. Aluminum tape is used to fix thermocouples to the surface of connector body. In this case, we look at the actual temperature seen by the connector to make sure the plastic body is not seeing temperatures outside the supplier's specification. After any adjustment to the oven, it is necessary to wait until the oven stabilises. The speed of stabilisation and its repeatability over a number of profiles are the marks of a

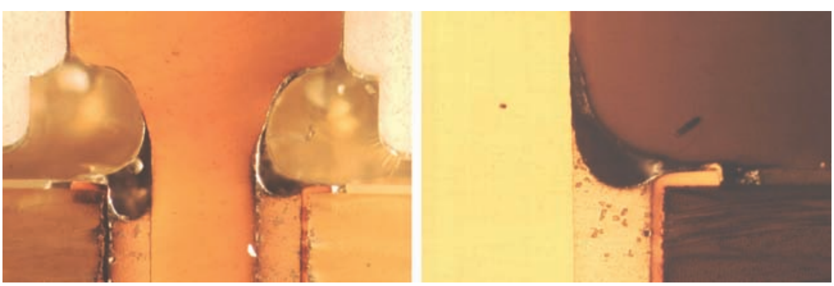


Figure 6-5: Microsections showing satisfactory solder joints formed with no-clean solder paste during convection reflow soldering in air. The examples both demonstrate 100% solder fill of the plated through-hole, but no topside fillet. These examples exceed the IPC 610E requirements.

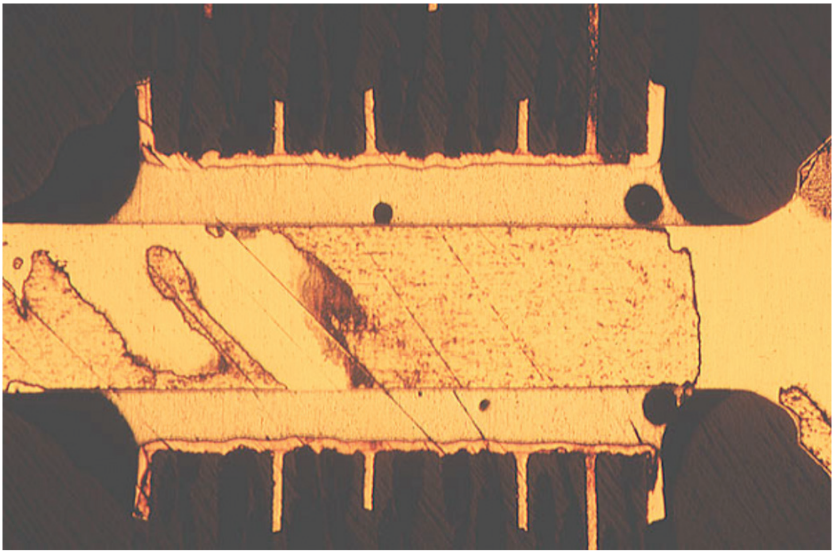


Figure 6-6: Microsection showing a satisfactory through-hole joint on a six-layer board with minor voiding. Solder fill after reflow of the paste is greater than 100%. The hole-to-lead ratio is quite large. If this were optimised with the same process parameters, it would result in positive solder fillets. This example exceeds the IPC 610E requirements.

good reflow oven. This should be part of the initial oven evaluation and should be understood by production staff.

Next, the first board with thermocouple leads attached can be passed through the oven and the temperature profile analysed. Adjustments can then be made to the zone temperatures and conveyor speed to obtain the correct profile. The desired profile is a combination of recommendations from the solder paste manufacturer, the component supplier's guidelines and the impact on the solderable finish. All surface finishes are affected to some degree by high temperatures. The correct temperature profile can eliminate solder balls and significantly reduce flux residues on many low-residue pastes.

All profiles should be developed on a fully populated board to guarantee that the correct conditions are achieved. If the boards are to be processed in or on support pallets, they should be used during profiling. The pallets contribute to the mass and therefore affect temperature rise on selected areas in contact with the board by as much as 10°C.

When a profile has been established, the board should be run through the oven again to monitor the profile. But this time, load the oven in front and behind the profile board to determine the thermal loading and the degree to which the temperature drops. Final setting changes may then be made to the oven zone temperatures.

Final Trials

When a temperature profile has been established and the board has been run in production with satisfactory soldering results, the following information should be retained:

- Solder temperature in each zone
- Speed of the conveyor
- Extraction rates
- Board loading

A temperature profile should be run regularly to build up a picture of the process stability. The frequency may then be adjusted depending on the repeatability of the results. Alternatively, a constant monitoring system may be considered for full tractability of the reflow process.

Further trials should also be run with the desired profile and production paste to determine the degree of slumping of the paste since it will affect solder shorting. Also consider the amount of paste dripping during the early weeks of production. Lower the temperature of the final zone to just below the reflow temperature of the paste. Pass a fully populated board through to examine the board on exit. Check the amount of slumping on fine pitch, under BGA devices and on chip components. This test is very useful to understand many of the causes of solder beading on chip devices.

Even with the best and latest convection oven technology, there will be a difference in peak temperature or duration between different board assemblies. If there is no difference, you are not conducting your profiles correctly. Don't ever be a lazy engineer with a single profile. Be a great engineer and learn more about your process.

PROCESS TRIALS PROCEDURES

Production engineers often conduct standard trials on reflow ovens during product assessment, machine approval or in-process set-up. The following trials may be of value and are also used by machine suppliers during equipment development and production audits.

Temperature Uniformity

Measure the surface temperature on an assembly or, better yet, on a blank laminate test board to obtain any variations across the complete conveyor belt width to show any peaks or low points between the centre and the edges of the conveyor. Ideal test results have temperature variations of less than 5°C.

Thermal Loading

First a temperature profile is produced as a reference, using up to six thermocouple probes soldered to the assembly - three or four on the top and at least two on the bottom. The oven is then thermally loaded with products. Alternatively, copper laminate or steel sheets may be substituted to fully load the oven. During loading, another profile is taken to compare the temperatures in this simulated production test. If only top side heating is being considered on the oven, all the probes are placed on the top of the board. Through-holes will have an impact on the results obtained. Ideal test results have temperature variations of 5°C to 10°C or less.

Temperature Stability

Surface temperature on an assembly or test board should be measured, checking variations across the complete belt width. Repeating this trial periodically throughout the day in production shows an oven's control system, even with a varying environment. The test should be run with one set-up, but may be run with different board types. Ideal test results have temperature variations of less than 5°C.

Throughput Speed

Adjustments are made to the conveyor speed for the maximum envisaged circuit board throughput requirements. The goal is to achieve the preferred temperature profile for the most complex product. The tests must consider the paste or adhesive requirements.

Over the last few years, a great deal of experience has been gained in processing tin/lead 63/37 and lead-free solder alloys like tin/silver/copper with successful results. Recently, as tin/copper pastes have also entered the reflow market, successful trials have been conducted with tin/copper/nickel, although with a higher liquidus temperature of 227°C, as opposed to 217°C to 221°C for the tin/silver/copper family of materials.

Sample Process Results

Some process results are provided here for future reference. Reflow soldering was conducted with two processes: convection in air

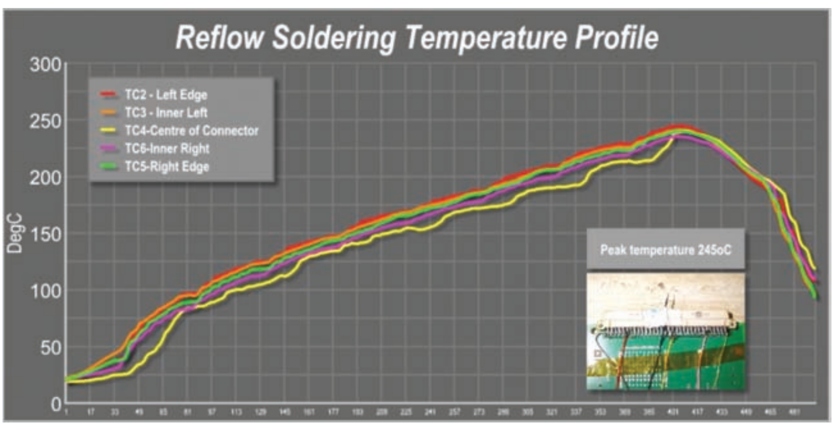


Figure 6-7: The reflow profile of a connector that achieved a maximum temperature of 245°C when processed in a BTU Pyramax™ 100 convection reflow oven. The profile was conducted on a six-layer 1.6mm thick board. The same design was used in the production trials.

on a BTU convection system and an ASSCON VP 6000 vapour-phase reflow soldering system. Both processes are commonly used for lead-free manufacture in Europe. Figure 6-7 shows the results obtained on both convection and VPS.

The reason for using a 240°C fluid in the example in Figure 6-8, as opposed to the more common 230°C fluid, is the reflow temperature of the tin/copper/nickel alloy. SAC alloys are 217°C to 221°C, whereas tin/copper/nickel reflows at 227°C. Some engineers like to use the same alloy type in reflow, wave and selective soldering. Tin/copper/nickel has been used for many years in wave and selective processes in the UK and mainland Europe. Any tin/lead assembly that has been successfully designed for manufacture, where component specifications have been correctly assessed, will still reflow reliably at 240°C.

Figure 6-8: ECD reflow profile illustrating the results achieved on thermocouples placed across the length of the connector and processed through the ASSCON VP 6000 vapour-phase reflow system running a 240°C fluid at the manufacturing facility of ACW in Wales.

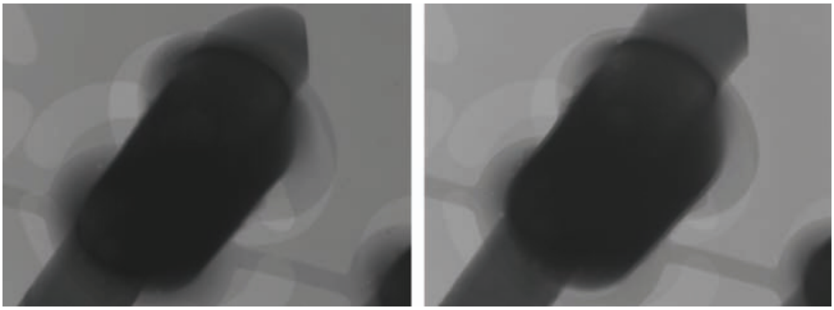


Figure 6-9: X-ray images show 100% fill on connector pins on the six-layer 1.6mm boards. Convection reflow with Hot Air Solder Level (HASL) (left). Vapour phase with a silver PCB solder finish (right). The x-ray images show virtually no void formation and positive solder fillets on the top and bottom of the through-hole barrel.

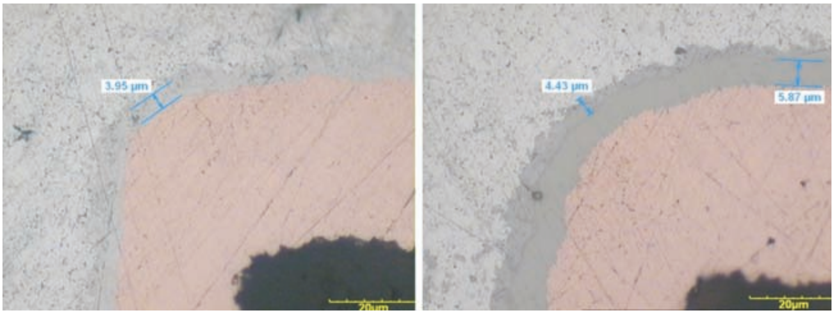


Figure 6-10: Microsection on connector pins after intrusive reflow with tin/copper/nickel paste. OSP board (left). Gold over nickel finish (right). The microsections were taken so the copper plating thickness could be measured after soldering to examine the difference in copper dissolution between SAC and tin/copper/nickel alloys. The solder joint strength and long-term reliability of PIHR are no different than with conventional wave or manual soldering operations. The joints also easily meet the visual requirements for national and international standards like IPC 610E.

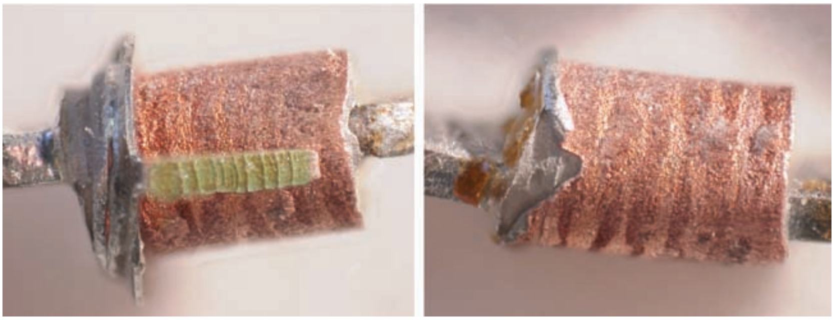


Figure 6-11: Typical examples of through-hole pins after pull testing from an intrusive reflow production board. Comparing solder joint pull strength between samples produced by wave (left) and selective (right) soldering show little practical difference. The example shows the copper barrel/solder joint removed from the board. Depending on the pin or connector design, it is more common for the pin to break or pull out of the housing during testing than the solder joint fail.

The results speak for themselves. The PIHR process is not that difficult. You just need to have a good design, a reliable process and final inspection. The peak temperature for lead-free is satisfactory and lower than some of the profiles we see featured in the industry. Although the reflow temperature of tin/copper/nickel is higher than SAC, the peak temperature can be the same with the exception of vapour phase reflow. **PIHR**

> **“If you spend a little time designing and correctly implementing PIHR, you can produce through-hole joints with positive solder fillets on both sides of the board.”**

CHAPTER 7

Solder Joint Inspection and Quality Control

Some companies have put off using the PIHR process due to variation in solder volume seen after reflow soldering during initial trials. Often the final joint shows a solder fillet flush with the board surface on both sides. In some cases, there is not enough paste to fully fill the plated through-hole. However, if you consider specifications like IPC 610 and the minimum requirement for level 3, which is the highest level for high reliability applications, these requirements can be easily met with intrusive reflow assembly.

JOINT RELIABILITY

Most specifications today are practical in their approach and have a minimum solder fill of 75% of the board thickness. Two European companies have a specification for intrusive reflow that requires 50% of the board thickness. In reality, there is little difference in the reliability of these joints. The IPC 610D, released in 2005, specifically covered through-hole reflow requirements. The latest version IPC 610E tends to treat joints produced with reflow the same as with other processes, which is very sensible.

In practical terms, if reliability testing were conducted on the joints in Figure 7-1, none of the above examples would fail if correctly soldered. Measuring pull strength is impractical for most pin in hole applications. Most joints will develop pull strength figures in excess of 6 to 10 kg in line with wave solder fillets. By way of comparison, a surface mount gull wing lead will be between 800 g and 2.2 kg based on lead pitches of 0.020” to 0.050”. If you spend a little time designing and correctly implementing PIHR, you can produce through-hole

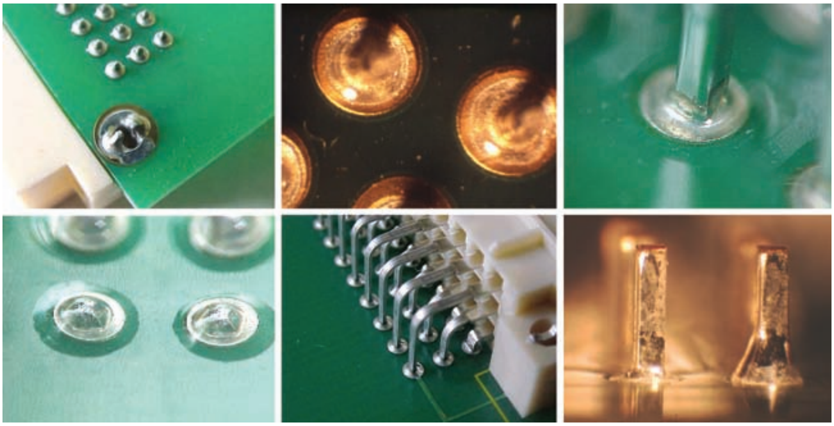


Figure 7-1: Sample joints.

joints with positive solder fillets on both sides of the board. In practical terms, many quality engineers would be hard-pressed to tell the method of manufacture.

The best and most practical approach for qualifying a process is to conduct testing with some microsections of selected joints to provide confidence and evidence of complete reflow with full wetting of the lead and plated through-hole. Performing an x-ray is a good way to assess the joints, the degree of voiding and solder fillets directly below the connector body. X-ray is an ideal means of monitoring hole fill in production and in-process monitoring of the total process.

INSPECTION CRITERIA

The following inspection criteria are in line with the requirements of IPC 610, but are expanded upon in selected areas to cover typical features of the PIHR inspection requirements.

Target - Class 1, 2, 3

To meet the target criteria for Class 1, 2 or 3, there must be 100% fill of the plated through-hole (Figure 7-2). Ideally, there should be evidence of positive solder fillets on both sides of a typical 1.6mm printed board and evidence of lead protrusion on both sides of the board.

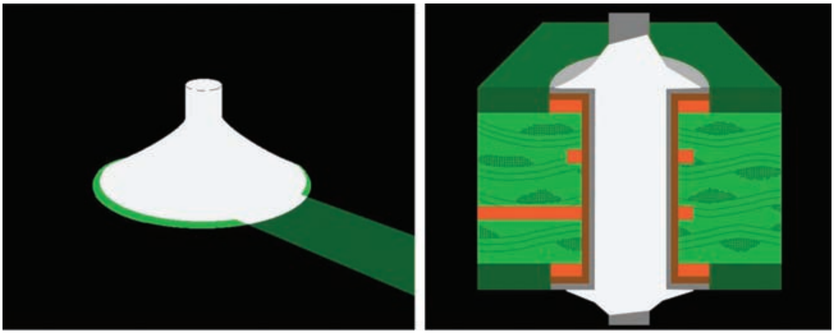


Figure 7-2: Target – Class 1, 2, 3 - 100% fill of the plated through-hole.

Acceptable - Class 1, 2, 3

To meet the acceptable criteria for Class 1, 2 or 3, there must a minimum of 75% fill of the plated through-hole. (Figure 7-3) A maximum total solder depression of 25% of the PCB thickness is acceptable and does not require rework. This can be made up from a sunken joint on both sides of the board, but the total fill must be ≥ 75% of the board thickness.

The opposite side of the board from the application of solder paste may have the original surface finish on the pad still visible, provided that there is evidence of good wetting around the pin in the plated through-hole.

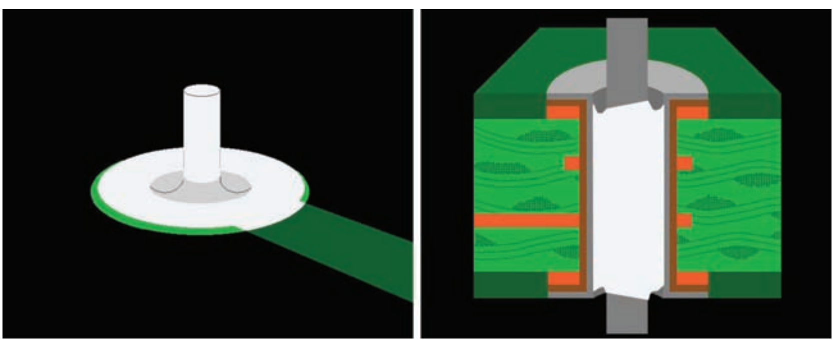


Figure 7-3: Acceptable – Class 1, 2, 3 - minimum 75% fill of the plated through-hole.

The component's protruding pins will ideally be coated with solder after reflow. However, due to lack of solder volume, there may be evidence of the base material visible on the pin. This is acceptable, provided that there is evidence of good wetting between the pin and the plated through-hole.

If the component lead does not protrude from the hole by design, it is acceptable, provided that there is evidence of good wetting

Where visual inspection of topside fillets is not possible, x-ray can be used. X-ray may also be used to confirm or measure the fill percentage of the plated through-hole.

There may be an increase of flux residues in a no-clean process after reflow of the paste deposit around the pad and on the solder mask area.

Defect - Class 2, 3

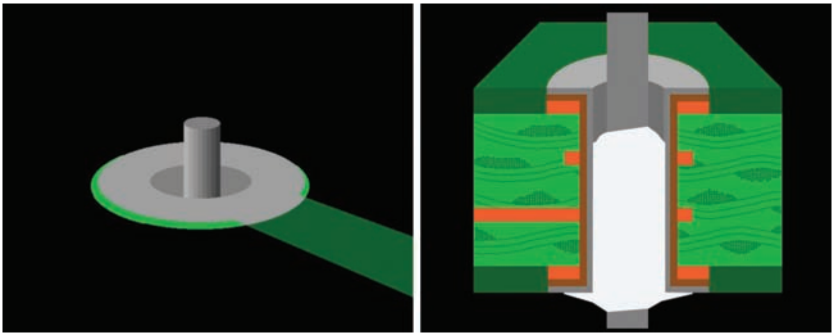


Figure 7-4: Defect – Class 2, 3 - solder fill < 75% of the board thickness.

METHODS OF INSPECTION

IPC 610 indicates that the target for solder vertical fill on plated through-holes is to be 100%. The acceptable level for classes 1, 2 and 3 is to be at least 75% fill, and anything less is deemed as unacceptable. Visually confirming the actual fill of anything less than 100% is difficult since you cannot see down the barrel of the hole to get a level. The

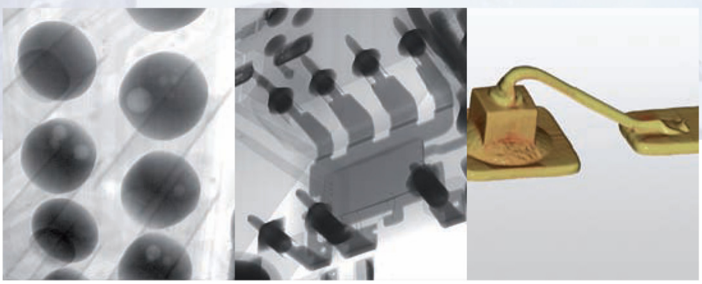
Nordson DAGE X-ray Inspection Systems

Crystal Clear X-ray Inspection

Focussed on the Electronics Industry

No Compromise in Analytical Quality

Ready for the New Technologies



Nordson DAGE XD7600 Diamond FP X-ray System

Unique 3Mpixel Radiation Resistant Flat Panel Detector

100nm (0.1μm) Feature Recognition unique Sealed-Transmissive X-ray Tube

Up to 10W of Power at Sub-Micron Feature Recognition

Rapid Switching between 2D and High Quality Fast 3D CT Mode

Now Available with X-Plane™*
CT Analysis Without Cutting the Board

www.nordsondage.com | pihrsales@nordsondage.com

*Patent Applied for

Nordson DAGE

top side of the joint is often obscured by the body of the through-hole component being soldered.

An alternative technique to qualify joint strength is to pull the pin in a controlled manner and detect the force required until it is removed from the hole or the pin breaks. Unfortunately, this method requires destructive examination of the sample. X-ray inspection offers a way of non-destructively investigating PIHR joints and, thanks to developments in x-ray inspection techniques in recent years, allows the percentage fill of these joints to be calculated.

Recent developments in x-ray inspection include:

- Increased use of open-style x-ray tubes for electronic applications. This makes much greater magnification available for joint inspection compared to using closed x-ray tubes.
- X-ray system design improvements that permit oblique angle views of joints without compromising the available magnification by tilting the detector rather than the sample.
- Digital x-ray imaging as a standard feature of x-ray systems enables far better visual separation of similarly dense features.

These x-ray developments allow even a relatively inexperienced operator to quickly assess and quantify the PIHR quality within

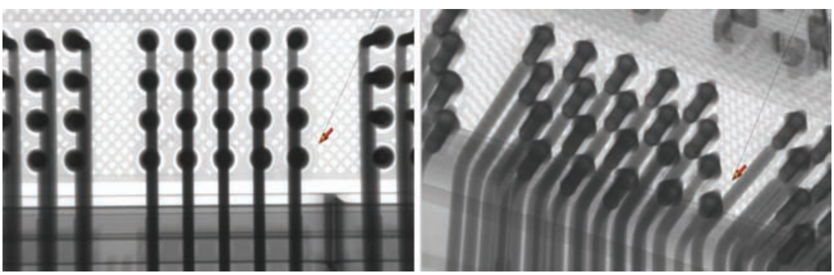


Figure 7-5: Normal view (left) and oblique x-ray view (right) of the same through-hole joints. The oblique x-ray view shows variation in hole fill that is not obvious in the normal vertical image.

production. They make it possible to see the individual PIHR joints at good magnification, resolution and contrast sensitivity. Achieving this at an angular view is vital for PIHR investigation, since the view from directly above will not show details through the hole and may result in missed voiding. X-ray has been used successfully for many years to examine intrusive reflow and selective solder joints in automotive applications as a process check. This technique has become popular because many large connectors obscure the view of the joints.

X-RAY INSPECTION CASE STUDY

[Author Note: The following is a case study from my personal experience.]

During four years of the SMART Group Lead Free Experience, a hands-on production feature run during exhibitions in the United Kingdom and Germany, I used a digital x-ray inspection system for joint analysis. The system had an open x-ray tube with sub-micron resolution that provided 16-bit grayscale sensitivity with an x-ray image size of 1.3 megapixels. The x-ray images were acquired at 25 frames per second. The system was able to provide oblique angles of up to 70° at any point 360° around any position on the test board without compromising the available magnification. This was achieved by tilting the x-ray detector instead of tilting the board. Software on the x-ray system provided a measurement of the fill percentage of the PIHR joints. It also allowed automatic inspection of selected joint areas on each board to be saved as video files for later comparisons with other inspection techniques.

X-ray inspection allowed modifications to be made in the printing process and reflow to adjust paste volume and voiding during production. This was particularly useful because different surface finishes and solder paste alloys were being implemented on each line. Measurement of hole fill with wet paste and displacement can also be monitored and pin insertion can be accomplished non-destructively. (See Chapter 3 – Solder Paste Application Methods)

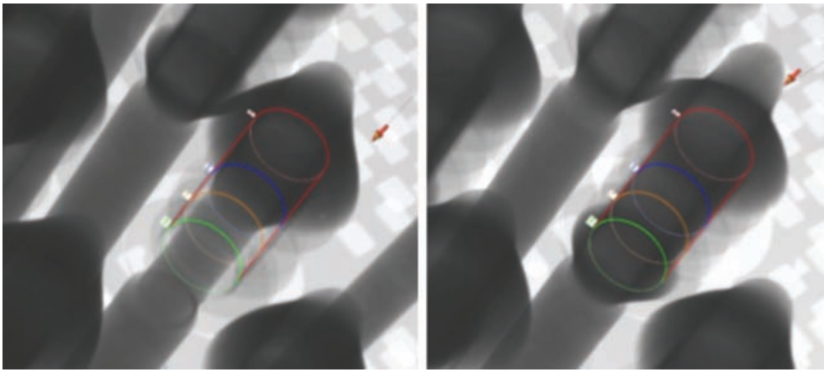


Figure 7-6: X-ray images showing fill percentage measurement on two through-hole joints in the same connector during process optimisation. The partially filled joint (left) has substantially less than 50% fill, as the scale shows.

Figure 7-6 illustrates how through-holes can be measured using x-ray images. On-screen cursors highlight the through-hole joint and allow for additional lines to be added at different percentage levels through the hole. In the figure, 0%, 50%, 75% and 100% levels are shown, but extra levels can be added as required. In the figure, the partially filled joint is substantially less than 50% filled and would therefore be unacceptable according to the IPC 610 criteria. The software allows the fill cursors to automatically change their orientation in agreement with a change in the oblique angle view selected. It can also change their relative size as the magnification is altered. In this way, analysis can be made at different oblique views if, for example, other components on the board interfere with the view of the joint under investigation.

The full colour report on the SMART Group Lead Free Experience project plus a variety of technical articles on inspection of PIHR joints with x-ray are listed Chapter 10 - Suggested PIHR Resources. X-Ray Inspection Criteria and Common Defect Analysis, by David Bernard from Dage, is a particularly useful reference book that features a guide on PIHR.

VOIDING

Incomplete reflow can normally be easily detected by inspection staff. Some degree of voiding may be seen in joint areas due to difficulty in the volatile agents of the paste escaping from the through-holes. It is far easier for these agents to escape during normal reflow of paste on the surface of a board. The gelling agents used in paste may also be difficult to fully displace during the soldering operation.

Minor voiding has no effect on solder joint reliability and has very little effect on pull strength. Voiding is a process issue that can be improved through correct profiling of a new board design with the paste supplier's support.

Small or large voids can be present after soldering if the reflow process is not considered properly. Correct profiling can virtually eliminate voiding in the PIHR process. X-ray inspection is a method that may be used to examine through-hole joint, as is microsectioning.

It is important during the initial PIHR trials to involve quality engineering to establish inspection criteria for production staff. This should be part of a training exercise implemented prior to production to eliminate debates on differing inspection criteria when a product is live on the shop floor or with a contractor. Is this a familiar story? Don't let it be in your company. Photographs included in this book, along with hundreds more available on CD-ROM, may be useful in setting inspection criteria. The CD-ROM and other inspection criteria for this process are available from IPC, SMTA, SMART Group and others.

IN-CIRCUIT TESTING

In terms of test access, joints that have been soldered by PIHR can be treated like any other through-hole soldered joint. They may be used as test access points during in-circuit testing. They may be probed with suitable test pins to facilitate test access. The only possible consideration is the amount of flux residue on the surface of the joint. There may be no more flux than on hand soldered joints, but there may be more than those with a wave or selective soldered termination using a no-clean flux.

Increasing test pressure or using different test pins improves test probe penetration. Test fixtures and probes should be regularly maintained in production with a set period for operation or a defined number of test cycles. They should have a fixed maintenance schedule based on time or throughput. Maintenance and refurbishment should not be based on test failures or no-faults found, since this situation will always occur during busy production times. Planned maintenance should be what it says – planned.

The amount of paste remaining on the pins depends on the paste used and the pin direction. If the pin is facing down during reflow, the flux residue can flow down the pin. If the pin faces up, the residue tends to flow down with the paste during reflow. This will depend on how much paste is displaced from the hole during pin insertion.

There are potential differences in the level of residue left on the surface of the board based on a combination of the following factors:

- Paste type
- Paste volume
- Reflow temperature
- Air or nitrogen
- Vapour phase reflow
- Clean or no-clean process

8

> "Since there is little difference between well-engineered intrusive joints and those produced by hand, selective and wave soldering, they will be equally reliable."

CHAPTER 8

8

Reliability of Through-Hole Reflow

INTRODUCTION

It is fair to say that through-hole solder joints that are correctly designed and manufactured do not fail. It is also fair to say that plated-through solder joint failures are very uncommon. Since there is little difference between well-engineered intrusive joints and those produced by hand, selective and wave soldering, they will be equally reliable. The long term reliability of through-hole solder joints with different alloys has been demonstrated by the UK National Physical Laboratory (NPL) in Teddington, London. NPL ran a lead-free soldering project that allowed customers to compare the results from lead-free production after long-term reliability testing. NPL has run many projects, funded by the UK Department of Trade & Industry (DTI) in collaboration with industry, to evaluate the joint reliability of various lead-free solders.

[Author Note: I support the NPL Team and the NPL Defect Database, which is a free online searchable source of process defects and suggested solutions, including information and support on PIHR. In addition to providing references for process defects and improvements, it includes all of NPL's process and reliability reports.]

PIHR PROCESS ISSUES

In PIHR processing there are some specific process issues that may occur that may not cause reliability issues on products, but are important process indicators. These issues can be eliminated by modifying process conditions or making minor design changes. One of the goals of this book is to highlight known issues, explain why they occur and help to overcome them. It is inevitable that additional production issues will occur, but we have tried to cover the most common and likely scenarios, including:

- Voiding in joints
- Variation in solder volume
- Incomplete reflow of solder
- Pad or fillet lifting on lead-free
- Flux residues
- Component damage

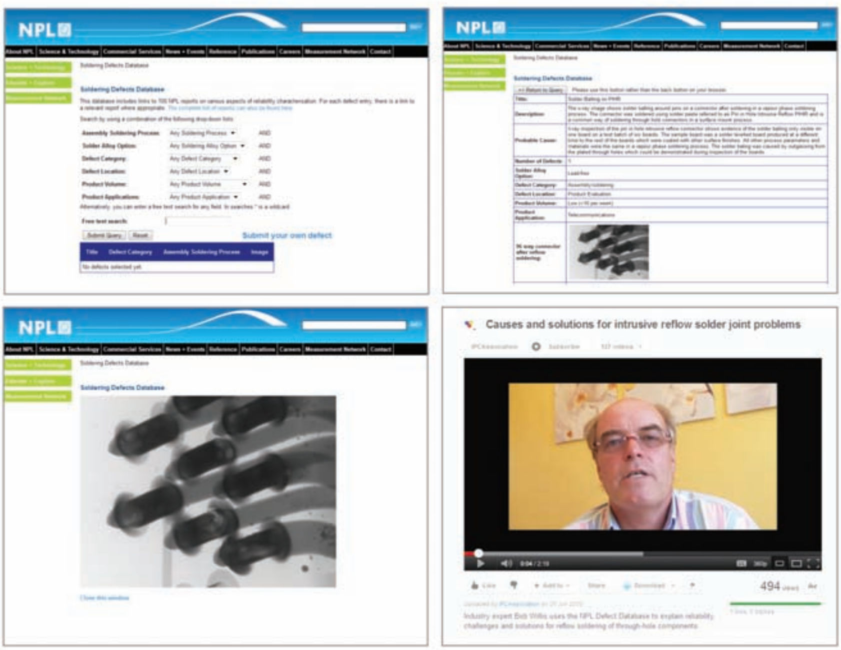


Figure 8-1: Sample screens taken from the NPL Defect Database during a search for process issues associated with PIHR production. Links are provided to reliability reports and in some cases videos from the IPC Defect of the Month library.

RELIABILITY TESTING PROJECT

Reliability testing was conducted on through-hole joints and lead-free alloy combinations on an existing NPL test board and featured a Harting 96-way edge connector (Type C DIN Connector) with a high-temperature body. The testing program involved a great deal more than just through-hole reflow joints with lead-free alloys and can be reviewed in "Reliability of mixed lead and lead-free alloys: simulating the effect of reworking electronic assemblies," which is listed in Chapter 10 – Suggested PIHR Resources.

The reliability project included 145 boards with a mixture of the following lead-free alloys: SnAgCu, SnAgBiCu, SnCu, SnPbAg and a standard tin/lead control. The test board consisted of a wide selection of surface mount components, as well as the PIHR

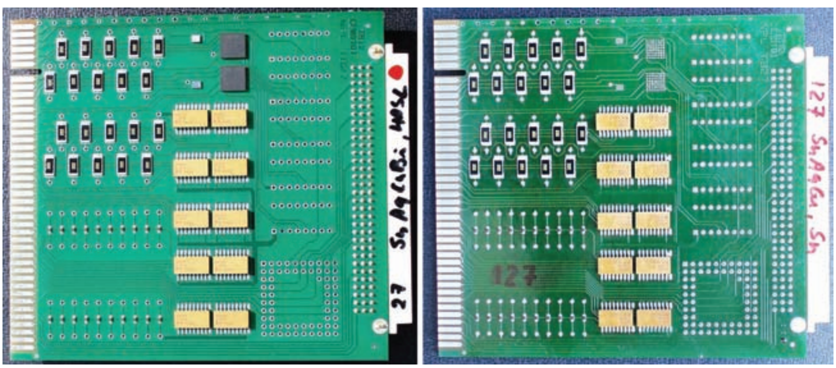


Figure 8-2: Test boards with connectors and a range of other surface mount components included in the NPL reliability study.

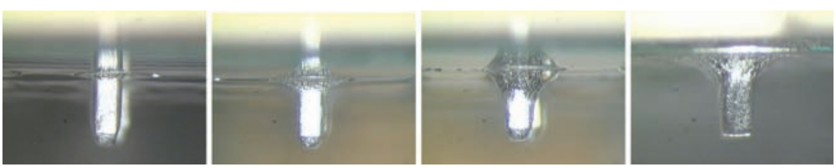


Figure 8-3: Through-hole joints after reflow and different solder volumes.

connector. The assembly process featured a standard process of stencil printing, placement and reflow in air on a convection reflow oven. The samples went through 2,200+ cycles of thermal testing, ranging from -55°C to +125°C, with dwell times of 40 minutes. None of the test boards exhibited any electrical failure. The NPL PIHR reliability project also involved the use of different stencil apertures in the printing process to obtain a range of solder joint volumes on each connector.

By producing solder joints with varying volumes, the NPL project produced a fair representation of joints that may be likely in production. It also covered the basic range of criteria illustrated in IPC 610E. As a result, the final reliability results of the report are more meaningful when compared to simply using the visual or x-ray joint assessments that are typical in the industry.

Figures 8-4 through 8-9 show PIHR microsections taken from the NPL project and illustrate some of the different joint structures produced. The sections are magnified approximately 100 times. In each figure, the joint between the connector pins is on the left and the copper-plated through-hole is on the right. **PIHR**

Figure 8-4: SnAgCu solder on Cu PCB finish

Figure 8-5: SnAgBiCu solder on Cu PCB finish

Figure 8-6: SnAgBiCu solder on tin/lead HASL

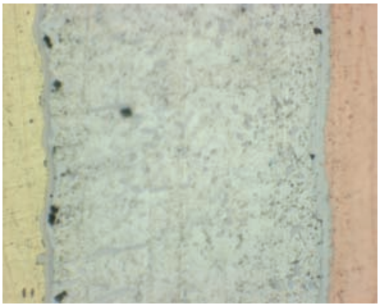


Figure 8-7: SnAgBiCu solder on tin PCB finish

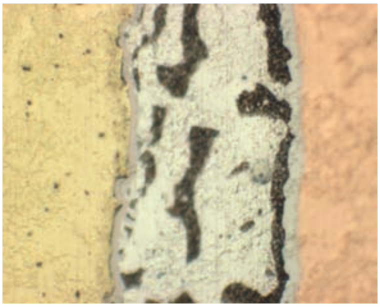


Figure 8-8: SnPb solder on Cu PCB finish

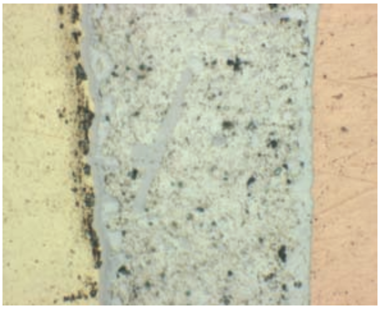


Figure 8-9: SnAgBiCu solder on silver PCB finish

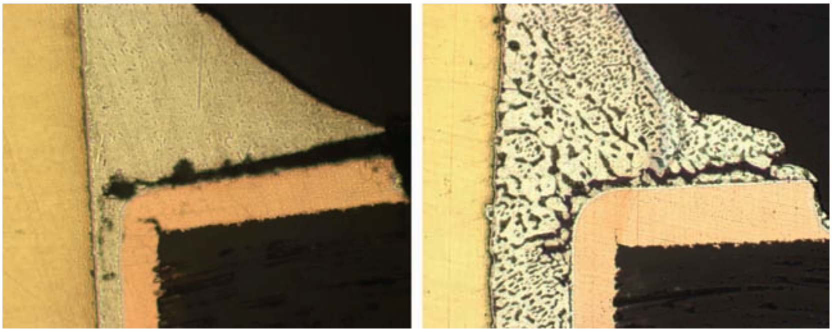


Figure 8-10: Microsections of through-hole joints after thermal cycling. The image on the left was produced with SnAgCu tin/silver/copper and the one on the right with SnPb tin/lead. In both cases, they show evidence of expansion of the board against the solder joint, leading to some separation of the topside solder fillet, but no electrical failure.

“By showing some of the common process issues, we hope to help engineers avoid the same process problems in their own facilities.”

CHAPTER 9

9

PIHR Manufacturing Defects — Causes and Cures

INTRODUCTION

In manufacturing processes, defects can occur. Things are no different with PIHR. Process problems can be caused by design, material selection or process set-up. This chapter provides photographs that illustrate some of the most common process issues seen during the introduction of PIHR and their potential causes. By showing some of the common process issues, we hope to help engineers avoid the same process problems in their own facilities.

ROUND PASTE DEPOSIT

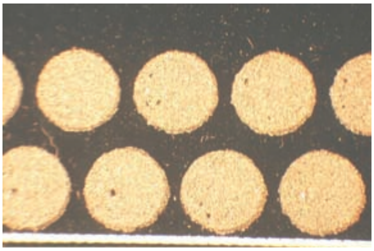


Figure 9-1: Staggered round paste deposit for through-hole soldering.

The deposit is satisfactory in terms of print quality and, provided that a suitable volume of paste is made available to the joint area, should produce a reliable joint. Care needs to be taken when overprinting paste onto the resist surface. The solder mask must be compatible to avoid solder balls during reflow.

SQUARE SOLDER PASTE DEPOSIT

Generally, a square deposit can produce more solder balls than a round deposit if the mobility of the paste across the resist surface is

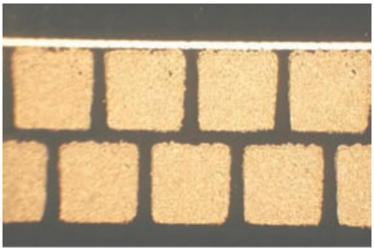


Figure 9-2: Square deposits used as an alternative to round to increase the volume of available solder after reflow.

poor. The solder balls are normally seen at the corners after reflow and are more common with low-residue, no-clean flux systems. The greater the flux activity, the less the problem of solder balling is seen in manufacture.

SPLIT ROUND DEPOSIT

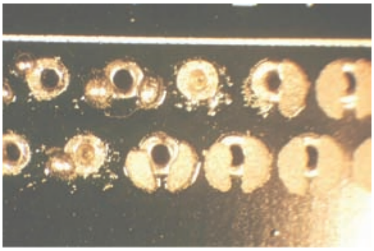


Figure 9-3: Split prints used to make insertion of the pins easier for manual assembly staff.

Since the hole is visible after printing, insertion should be less of a problem during pin location. Unfortunately, all of the trials conducted have shown that the paste does not always reflow back evenly and it is very difficult to eliminate solder balling. The use of split prints also reduces the amount of paste available to form the joints on the through-hole parts. The stencil aperture does not allow paste to enter the hole.

PASTE DISPLACEMENT

During connector insertion, it is impossible to eliminate contacting the solder paste deposit. After reflow, the result is smaller joints and a lot of solder balls. When designing the paste deposit aperture in the stencil, there should be no contact between the paste and component body. The body design or the stand-off feet should prevent

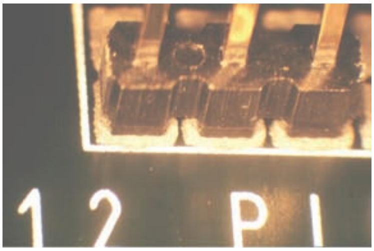


Figure 9-4: Poor post header and paste deposit combination.

paste displacement. Ideally, the component stand-off should be more than 0.010". Using a different stencil aperture design may eliminate the problem, but there will always be some parts that may not be suitable for the PIHR process.

PASTE DISPLACEMENT

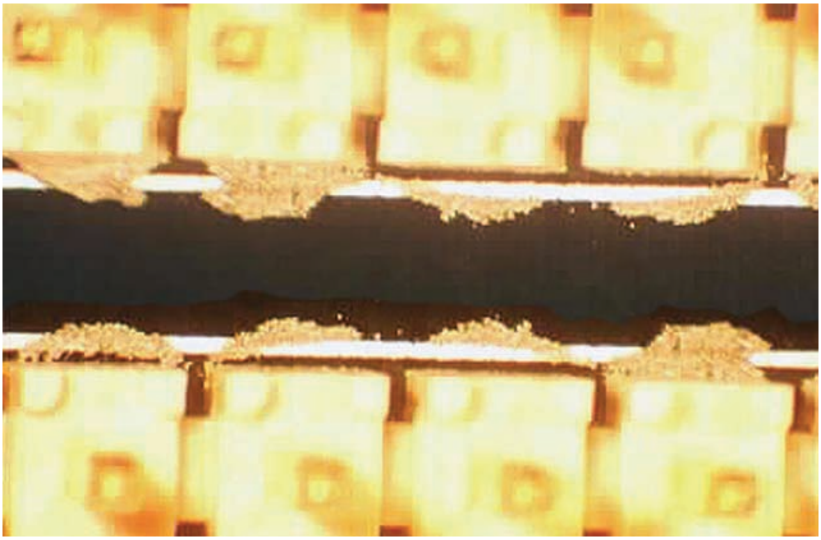


Figure 9-5: Minor paste displacement after component insertion, resulting in solder balls.

Minor paste displacement after component insertion will result in solder balls. It is only through correct aperture design and comparison with connectors that this problem can be eliminated.

PASTE DISPLACEMENT

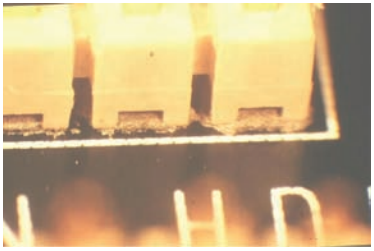


Figure 9-6: Close-up of stand-off feet on connector with minor paste displacement.

Minor paste displacement around the stand-off feet on a connecter will result in solder balling during reflow. When designing the paste deposit aperture in the stencil there should be no contact between the paste and component body. The body design or the stand-off feet should prevent paste displacement. Ideally the stand-off should be more than 0.010".

CONNECTOR BASE DESIGN

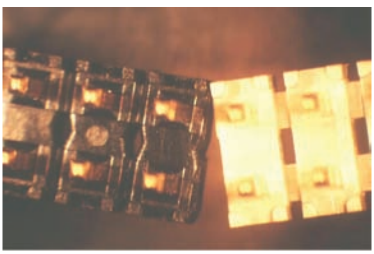


Figure 9-7: Two examples of post headers showing different base designs with slightly different stand-off feet.

The use of different stand-off feet with different base designs can cause process problems. Care must be taken when selecting the initial supplier and even more care if alternative vendors are required. Simply changing the part supplier without analysing the impact on production can result in process problems. But, there's nothing new there! If alternative suppliers are considered, they should be selected based on a common company standard.

SOLDER BALLING

There are a variety of types of solder balls that can be produced after reflow soldering. Paste is displaced by the connector and it is

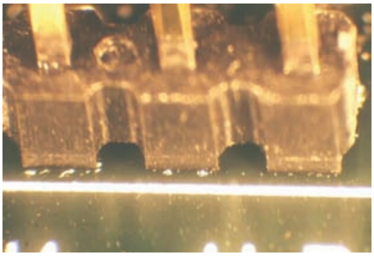


Figure 9-8: Types of solder balls that can be produced after reflow soldering.

prevented from reflowing back to the through-hole area to form the joint. It can be argued that if the product is to be cleaned, this is not an issue, but it will reduce the volume of solder at the joint interface.

SOLDER BALLING

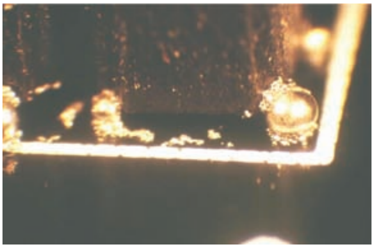


Figure 9-9: Solder balling caused by a combination of poor paste aperture and component design.

If there is a combination of poor paste aperture and component design, when the paste makes contact with the body it will not reflow back to the through-hole. If alternative component suppliers are introduced after a stencil design has been defined, this may be the result. Care must be taken on component designs from second source suppliers and on the initial stencil design.

POOR WETTING

Poor wetting on the pad surface may be caused by solderability issues with the printed board. Solderability of boards can vary depending on the surface coating. Normally gold and tin/lead have a shelf life in excess of 12 months. A copper OSP coating, however, may be between 3 and 6 months. Poor wetting can also be caused by insufficient paste in the through-hole during reflow. In figure 9-10, paste has been printed to the opposite side of the board and the component

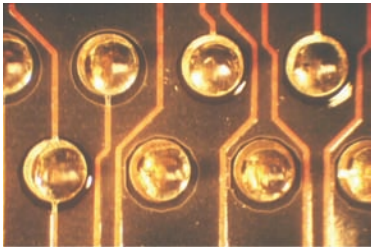


Figure 9-10: Evidence of poor wetting on the pad surface.

has been inserted. Calculations of the correct solder volume need to be reviewed.

SOLDER BALLING

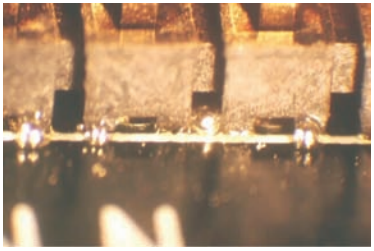


Figure 9-11: Solder balling caused by paste being trapped between the circuit board and the connector.

During reflow the paste cannot coalesce back to the through-hole joint area. A video section of the reflow of connectors showing solder balling is shown on the Pin In Hole/Intrusive video listed in Chapter 10 - Suggested PIHR Resources. The design of the stencil, and the resulting paste image, is produced to meet the demands of the connector pin layout. The stand-off height of the paste should be greater than 0.010” to avoid contact with the base of the component.

LIFTED COMPONENT

It is unlikely that a connector would lift from the board if correctly positioned prior to reflow. It is more probable that the connector was not pushed fully into the hole. It is possible that insertion may have been restricted by the hole-to-lead ratio, making full insertion impossible. Care should be taken to check the hole-to-lead tolerances.

Through-hole connectors have been known to lift if the tolerances vary from one end of the connector. As the board and the connector

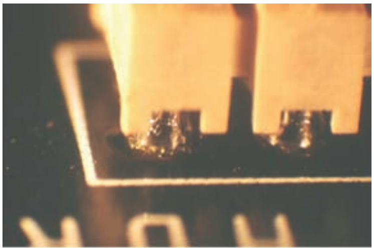


Figure 9-12: Connector that was not pushed fully into the hole, resulting in lifting from the board.

expand the pins can be forced against the hole wall distorting the body of the part and causing it to lift.

SATISFACTORY JOINT

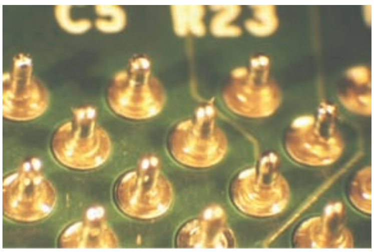


Figure 9-13: Satisfactory solder joints on through-hole Pin Grid Array (PGA) connectors.

The solder fillets have 100% fill and will provide reliable joints. The normal strength will be in excess of 20 pounds per joint. The inspection criteria in many companies may have to change for intrusive reflow as well as the existing international standards produced by the IPC and the IEC in Europe.

SATISFACTORY JOINTS

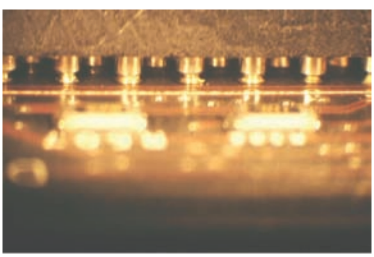


Figure 9-14: PGA socket.

The top-side solder fillets on this PGA socket are satisfactory and would meet most existing inspection criteria. It is relatively easy to achieve a 75% solder fill, which is the minimum criteria for most inspection standards. But often the 75% rule still requires the solder to have wetted both sides of the board, which is not possible on all pin in hole joints.

POOR WETTING

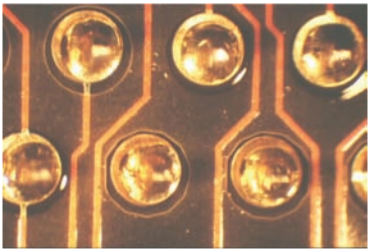


Figure 9-15: Close-up view of poor solder coverage on the surface of through-hole pads.

Poor coverage on the surface of through-hole pads can be a function of the solderability of the basic board. But on the base of the board, it is more likely to be insufficient paste pushed through or available in the hole. Tests have shown that the strength of the joints is not affected, but it would make visual inspection difficult. Examination of the paste volume will be required and modification of the stencil design may be necessary.

SOLDER BALLING

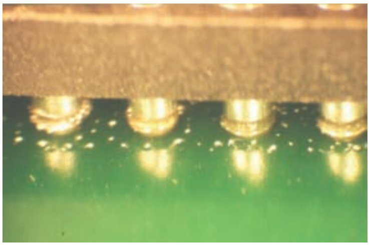


Figure 9-16: Solder balls on the top side of the board after reflow.

Solder balls on the top side of the board after reflow are often the result of incompatibility between the solder resist and solder paste. Different solder resists affect the mobility of paste in a liquid or semi-liquid state when reflowing on the mask coating.

As the paste reflows and coalesces, some of the solder particles do not flow back to the joint. This is seen more with no-clean/low-activity materials than with high activity flux. It could be argued that with high activity materials customers would be cleaning their boards. In this case, the solder balls would be removed in the cleaner process.

The solder mask should be tested with the paste by reflowing a volume of paste on the chosen mask and looking for any balling. The problem is less likely to be directly related to the paste. There is evidence that round paste deposits produce less solder balling than square printing. The corners of the print tend to separate during reflow.

PASTE SEPARATION

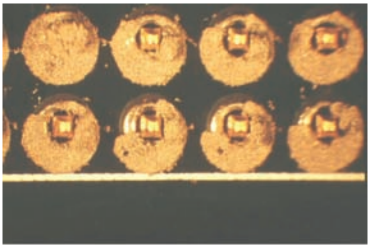


Figure 9-17: Pin in Hole/Intrusive Assembly conducted upside-down with components mounted in pallets and boards placed over the connectors from above.

Pin in Hole/Intrusive Assembly may be conducted upside-down with components mounted in pallets and boards placed over the connectors from above. In this case the paste deposit is on the top of the board when the pins enter from the bottom side. If the solder paste has dried out, the pins can break off paste causing it to be displaced from the joint area which may lead to solder balling. Any reduction of the amount of solder to fill the joint can also be a problem, possibly causing rejection of the joints. Today paste-drying is less of a problem than it used to be, with the major suppliers making products with many hours of tack life.

PASTE PUSHOUT

Paste being displaced from a through-hole can occur for a number of reasons (e.g., hole-to-lead ratio, pin length, paste drying out, shape of the pin, etc.) The most common cause is length of the through-hole pins which must be controlled for this assembly technique to work

Figure 9-18: Paste being displaced from a through-hole.

effectively. Pin length should be controlled between 1.0mm and 1.5mm. Any longer will displace paste which will not always reflow back to the joint.

PASTE SEPARATION

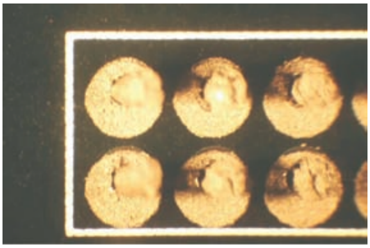


Figure 9-19: Pin in Hole/Intrusive Assembly conducted with components mounted in pallets and boards placed over the connectors from above.

Pin in Hole/Intrusive Assembly may be conducted with components mounted in pallets and boards placed over the connectors from above. In this case, the paste deposit is on the top of the board when the pins enter from the bottom side. If the solder paste has dried out, the pins can break off some of the paste deposit causing it to be displaced from the joint area. This can lead to solder balling and reduce the amount of solder available to completely fill the plated through-hole. Today most paste vendors supply products that have extended tack life.

HOLE FILL

This example shows the complete hole fill achieved during stencil printing of paste. It is possible to increase through-hole fill by changes to print speed, angle of the squeegee blade or by using the sealed printing processes available from DEK and MPM. Having 100% hole fill still may not provide the correct solder fillet since, when reflowed, the

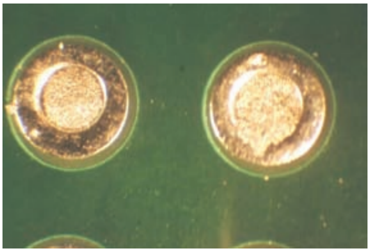


Figure 9-20: Complete hole fill achieved during stencil printing of paste.

paste will only be 50% of its original volume. It may also be necessary to print over the pad or even extend onto the resist surface.

PASTE PUSHOUT

Figure 9-21: Excessive paste push-out.

Excessive paste push-out will prevent any satisfactory joints from being formed. The pin length should be controlled between 1.0mm and 1.5mm from the base of the board. In figure 9-21, all of the paste has been displaced from the through-hole. When reflowed, the solder will wet the entire length of the pin and leave no solder to fill the hole.

SOLDER BALLING

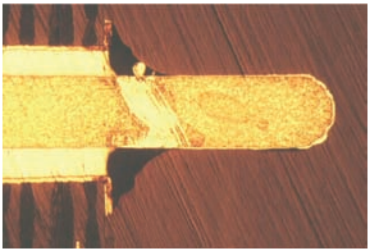


Figure 9-22: Solder balls on the surface of pins.

Solder balls are seen on the surface of pins if the reflow profile is incorrect or there has been a delay in soldering the assembly. Small solder balls form on the pins, which may or may not be an issue. Figure 9-22 shows a satisfactory solder joint. After microsectioning, one solder ball is visible on the side of the pin.

SATISFACTORY JOINT

Figure 9-23: Microsection of a pin grid array lead showing a satisfactory solder joint at the base of the pin.

In a pin grid array lead with a satisfactory solder joint at the base of the pin, the hole is filled with 100% solder. A small void is visible on the side of the through-hole, which will have no effect on reliability of the joint.

It is worth pointing out that voiding may not be due to solder paste. It can be due to outgassing from the board. Remember, the copper thickness in the through-hole should be a minimum of 25µm to eliminate the moisture gassing from the board during soldering. Even old wave soldering problems can have an impact on modern through-hole reflow assembly processes.

SOLDER VOID

Figure 9-24: Solder voids in through-hole joints during intrusive pin in paste assembly.

Solder voids may be present in through-hole joints during intrusive pin in paste assembly. The paste is trapped in the through-hole during this process and either the volatile material or the gelling agents may have difficulty escaping during reflow. Experiments with different profiles and different paste parameters before setting up your final process should reduce or eliminate the problem. Voids, however, have little effect on reliability. They have often been demonstrated to improve the long term reliability of joint.

POOR PASTE REFLOW

Figure 9-25: Incomplete reflow of solder paste.

During reflow of through-hole connectors, the lowest temperature on the board is often seen under the body of these parts. They have a far more significant effect than large surface mount parts. If the paste does not completely reflow, or is not in a liquid state for a long enough period, a true intermetallic bond will not be achieved. In figure 9-25, the paste has not fully reflowed or wet the hole and pin. The assembly would require a more detailed investigation into the profile being used and modifications would need to be made to the peak temperature or the soak section of the oven.

INCOMPLETE SOLDER FILLETS

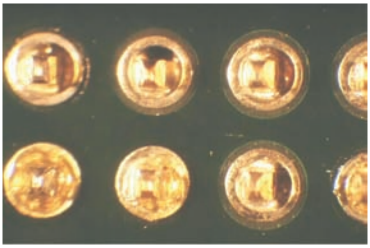


Figure 9-26: Inconsistent solder fill of the plated through-holes during pin in hole reflow.

Inconsistent solder fill of the plated through-holes during pin in hole reflow is probably due to a lack of consistency in the printing process. Some of the holes show complete fill and others show poor hole filling. Like surface mount, the printing process needs to be controlled to ensure that consistent solder paste deposits are maintained.

SOLDER BALLING

Figure 9-27: Solder ball visible on the surface of the board after microsectioning.

Solder balls may be visible on the surface of the board after microsectioning. In figure 9-27, the solder ball is positioned just away from the copper pad in the solder resist aperture. During reflow, all of the paste has not coalesced back to the joint, which may be due to the immobility of the paste on the surface of the solder mask. The solder ball may also have occurred as a result of a washing exercise during a previous poor print cycle.

SOLDER BALLING

Figure 9-28: Solder balls on the surface of pins.

Solder balls are seen on the surface of pins if the reflow profile is incorrect or there has been a delay in soldering the assembly. Small solder balls form on the pins which may or may not be an issue. Figure 9-28 shows a satisfactory solder joint after microsectioning the solder fillet. One solder ball is visible on the side of the pin.

SATISFACTORY JOINT

Figure 9-29: Microsection showing a satisfactory solder joint after reflow of the through-hole pin.

Figure 9-29 is a microsection showing a satisfactory solder joint after reflow of the through-hole pin. A near 100% solder fill has been achieved on the through-hole joint after paste reflow. It is relatively easy to achieve 75% to 100% solder fill. Making positive solder fillets is far more difficult and is not necessary for long-term reliability.

SATISFACTORY SOLDER JOINT

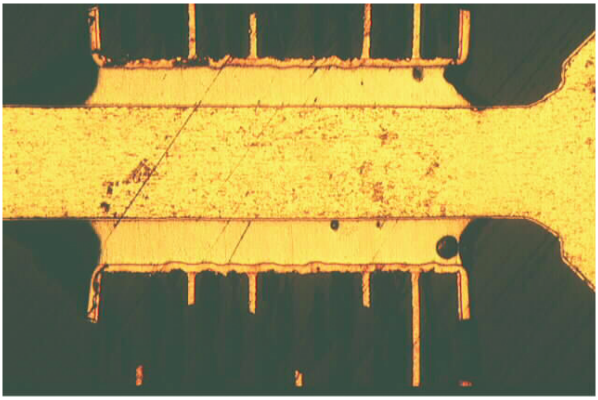


Figure 9-30: Microsection showing 100% solder fill after through-hole reflow.

Figure 9-30 is a microsection showing 100% solder fill after through-hole reflow. The solder has filled the whole of the joint and formed a fillet with the pin and through-hole. Control of the solder paste volume is the correct way to achieve a stable process for pin in hole/intrusive assembly.

SATISFACTORY JOINT

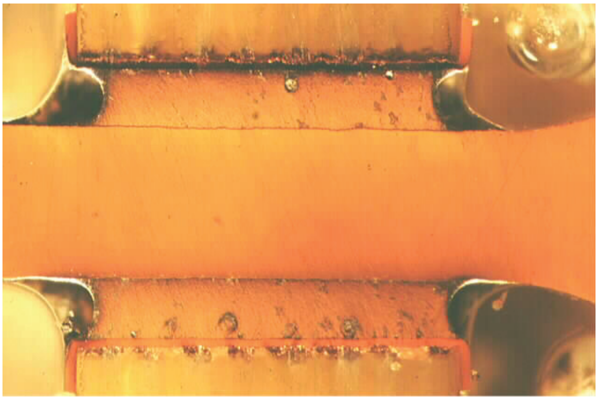


Figure 9-31: Satisfactory joint with less than a 100% solder fill.

Figure 9-31 is a satisfactory joint with less than a 100% solder fill. Solder wetting is still visible on the plated through-hole and the pin. It is common in this process not to always have a 100% solder fill.

INCONSISTENT JOINTS

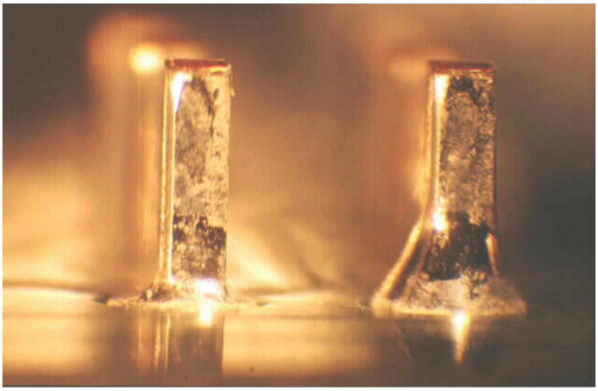


Figure 9-32: Two solder joints that have been reflowed.

The two solder joints that have been reflowed, shown in figure 9-32, were produced using a tin/lead solder preform. One of the problems with solder preforms is that they do not always reflow consistently, resulting in more solder on adjacent joints. Solder preforms are produced from standard solder alloys and formed into a pattern to suit the connector. The solder pattern for each joint is linked together with thin solder sections that do not break evenly during reflow. Solder preforms do, however, provide a consistent solder volume to each joint when correctly engineered.

SOLDER BALLS

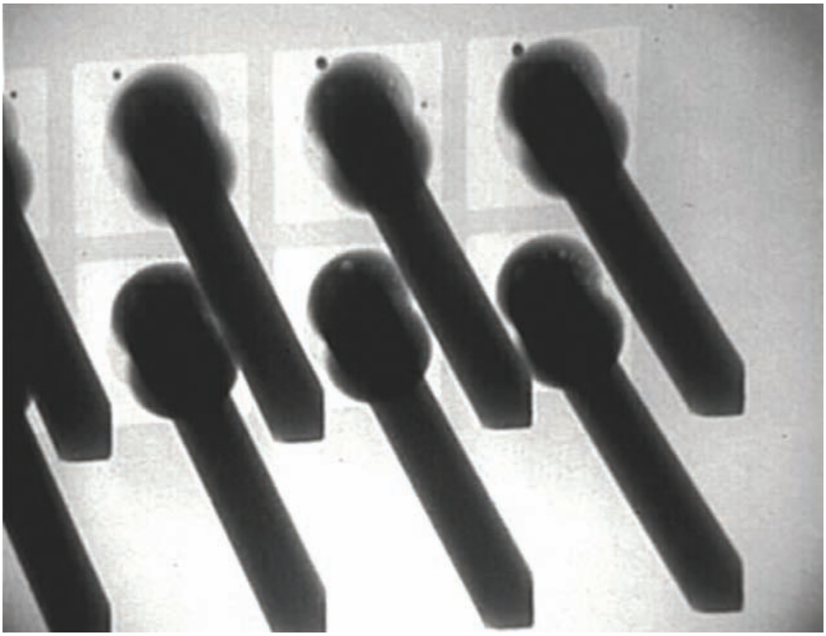


Figure 9-33: Solder balls visible during a routine x-ray examination of a joint.

Solder balls in figure 9-33 are seen around the square pads at the top of the picture and would be easily seen on the top of the board if the joints were not masked by the connector body. The solder joints are sound with some minor evidence of voiding in the fillets. X-ray examination is a useful option for analysis of joints, but not a necessity with pin in hole assembly.

SOLDER VOIDING

Solder voiding is either due to volatiles from the flux not escaping prior to solidification of the solder or due to gelling agents in the

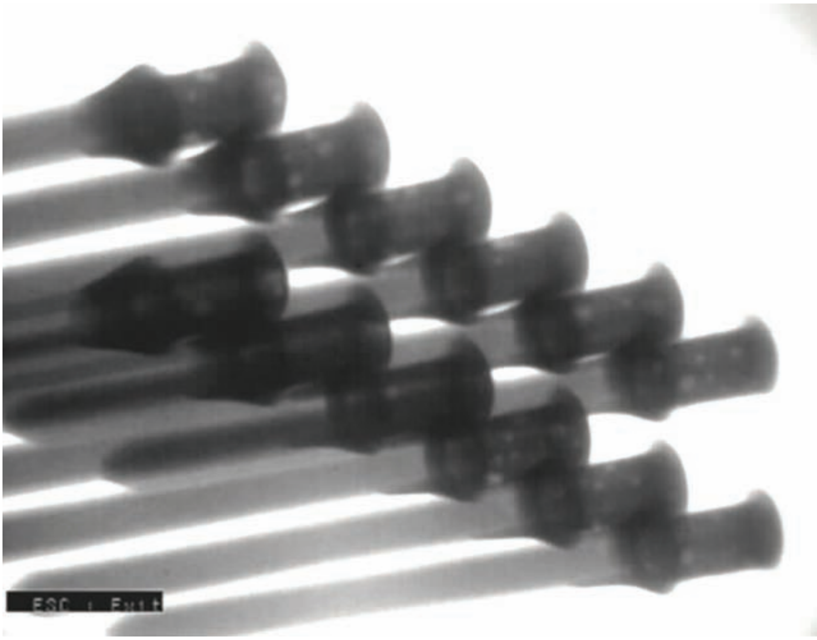


Figure 9-34: Solder voiding in joints detected with x-ray examination.

paste not escaping from the body of the paste when in a liquid state. The voiding does not necessarily affect the reliability of the joints, but is a process indicator that the reflow has not been optimised. Voiding can be seen if the body of the component is flush with the surface of the board or if the top of the pin seals the top of the plated through-hole.

PASTE NON-COALESCENCE (GRAPING)

Graping is a common fault on small paste deposits when people return to a traditional soak profile. Some solder pastes are far more tolerant than others, the resulting solder joint is sound but the balls have not fully reflowed into the bulk of the solder joint. This is what engineers refer to as graping or the author calls warts.

Graping has been seen during the last couple of years when implementing 0201 and 01005 chip terminations in convection reflow, but

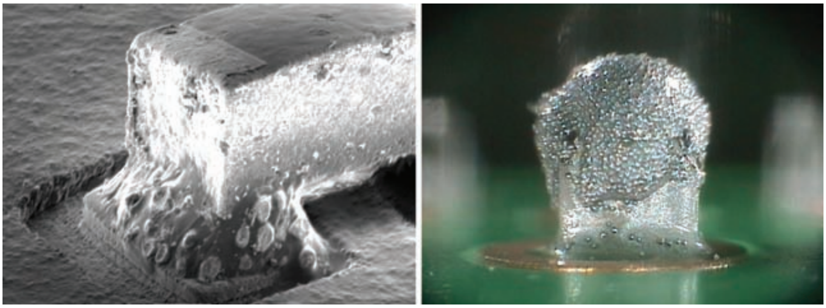


Figure 9-35: 0201 chip component with lead-free solder paste that has not fully reflowed in air (left). Similar effect on PIHR (right), caused by having paste on the end of the pin for a long period during preheat and soak.

is never seen in vapour phase soldering due to the inert atmosphere. Basically, the ultra small solder paste deposit is exposed to elevated temperatures for a long period during a lead-free profile, which reduces the performance of the flux in the paste. It is seen more with profiles that have a long pre-heat like a traditional soak profile. A profile that does not have a long dwell prior to reflow is less affected. **PIHR**

10

"Many of the papers may be available through an Internet web search."

CHAPTER 10

10

Suggested PIHR Resources

The following are published articles, reports and videos on Pin in Hole Intrusive Reflow that may provide additional information beyond the scope of this book. Copies of texts should be available from the author, the technical publication where it appeared or through a technical library. Many of the papers may be available through an Internet web search.

If you are aware of any further articles on PIHR, please contact the author and provide him with the reference information to update this list of resources.

Bob Willis
Bobwillisonline.com
bob@bobwillis.co.uk
Tel 00 44 1245 351502

TECHNICAL PAPERS AND REPORTS

Evaluation of High-Density Surface Mount Process Compatible Through-Hole Connectors
Martha L Rupert, AMP Inc.

Reliability of Mixed Lead and Lead-Free Alloys: Simulating the Effect of Reworking Electronic Assemblies
Alan Brewin, Chris Hunt, M Dusek, J Nottay – NPL

Measurement Variation in the Fill of Intrusive Reflowed Lead-Free Solder Joints with Different Board Finishes by X-Ray Inspection
David Bernard, DAGE and Bob Willis

X-Ray Inspection Criteria and Common Defect Analysis Book
David Bernard, DAGE and Bob Willis

Process Temperature Testing and Classification of Surface Mount Connectors
Carman LaRosa, AMP Inc

Advanced Surface Mount Manufacturing Methods
Joe Belmonte, MPM Corp

Step Soldering to Aid Intrusive Reflow
Joe Peek Futronicx & Karl Seelig, AIM Inc

Through-Hole Assembly Options for Mixed Technology Boards
by Ross B. Berntson, Dr. Ronald C. Lasky,
Karl Pfluke Indium Corporation
http://www.indium.com

From Wave Soldering to STH Technology
Sauro Electronic Products
http://www.sauro.net/pdf/cat/SAURO_STH_1.pdf

SMART Group Lead-Free Experience Report 2006
Bob Willis SMART Group

Pin In Hole Reflow Introduction University Course Content
Martin Tarr
http://www.ami.ac.uk/courses/topics/0226_pip/index.html

Determining Solder Volume for High-Density Surface Mount Process Compatible Through-Hole Connectors
Martha L Rupert, AMP Inc.

Reflow of Solder Pre-Form Arrays in Nitrogen Reflow
Dan Uno, Hewlett Packard, Palo Alto, California

Developing the Paste-In-Hole Process
Tom Gervascio, Group Technologies,
Tampa, Fa, Nepcon West proceedings

Through-Hole Reflow Drives Production Costs Down
Gerald Rutter, BTU Europe,
Electronics Manufacturing Products magazine

Paste Printing for Through-Hole Components
Ray P Prasad, SMT magazine

Design Characteristics of Surface Mount Compatible Through-Hole Connectors
Martha L Rupert, AMP Inc.

Pin-In-Paste Solder Process Development
Raiyo Aspandiar, Mark Litkie, and George Arrigotti
Intel Corporation Hillsboro, Oregon

Pin In Hole Printing Application Note
MPM Speedline, Franklin, USA

TRAINING VIDEO

Introducing Pin In Hole/Intrusive Reflow Soldering
Bob Willis

CD ROM PHOTO ALBUM

Pin In Hole and Intrusive Reflow Photo Album
Bob Willis

INTERACTIVE CD ROM

Introducing Pin In Hole/Intrusive Reflow Soldering
Bob Willis

INSPECTION AND QUALITY CONTROL POSTERS

A set of inspection and process defect posters for pin in hole reflow assembly process are available from IPC, SMTA and the SMART Group.

11

“If wave or selective soldering is to be eliminated, either hand soldering, single point automatic soldering or reflow must be used.”

CHAPTER 11

11

Frequently Asked Questions

The following is a list of questions that are typically asked about PIHR and the answers that the author, Bob Willis, provides. They may be beneficial as a quick guide to the process and may answer some questions that may arise.

Why reflow solder through-hole components?

In many electronic assemblies, there are large multi-leaded components still being used in combination with surface mount. If wave or selective soldering is to be eliminated, either hand soldering, single point automatic soldering or reflow must be used. The major driving force is manual cost reduction and a simplified process.

How is through-hole reflow conducted?

Solder paste is applied by stencil printing to the through-holes and to the surface of the pads at the same time as the surface mount printing process. The through-hole components are then carefully inserted, just prior to reflow or before surface mount assembly, to avoid the possibility of jarring parts if any snap fixings are included on connectors.

Do I need two stencils for through-hole printing? One for through-hole and one for traditional SMT parts?

It depends on if you are feeling charitable to your stencil supplier. Some people have used the technique to increase paste volume or if there was a problem changing hole size or a thick PCB substrate is being used.

[Author's Note: Make sure you tell your stencil supplier that the round apertures are required in your stencil for through-hole printing. Suppliers often remove them for customers who have not supplied a solder paste file for the stencil. Those wonderful stencil people may get carried away. I forgot to tell my stencil manufacturer last week. Do as I say, not as I do!!!!!!!]

How many components can be soldered in this way?

I don't know the answer to the question. Each component must be considered for this process and its needs assessed and discussed with the component manufacturer, just like immersion cleaning, wave soldering, etc. Generally speaking, it is the high pin count devices, like connectors, pin grid arrays, post headers, sockets and dual in-line parts, that have been produced specifically for reflow applications.

What is the best use of this technique?

The technique is best used with back plane or junction boards, where you have loads of connectors and functionality with lots of surface mount components. It can also replace some press fit designs. Its a godsend to manual assembly lines and to situations where design engineers put through-hole connectors on both sides of the board.

What will the solder joint reliability be like for intrusive reflow joints?

There should be no difference in the solder joint. There may be a difference in the solder volume due to the limitations of the printing process. Just try ripping a through-hole lead out of an existing soldered plated through-hole if you are strong enough. If the microsection looks good, there is no difference in the pull strength characteristics.

Are any voids left in the joints?

Yes. You can find voiding due to the reduced escape of gas and other non-metallic materials during paste reflow. Being careful setting process parameters will keep this to a minimum. Voids can also be generated by outgassing from the PCB, just like wave soldering, and are not specifically related to the reflow process. Voids have been seen to improve reliability, not reduce it! Think of any honeycomb structures.

When the component is inserted, does the paste get forced out of the holes?

Yes. Some paste is forced onto the tips of the pins. The amount is dependent on the care taken during component loading, the shape of the pin and the paste type used.

What happens to the paste on the pin tips during reflow?

As reflow takes place, the solder remains on the pin, evening out the thickness on the pin. However, there will always be some slight build-up of solder on the pin tip. There is also a difference depending on which the pin is positioned during reflow (up or down). Paste drop-off during pre-heating can occur, but it can be monitored so corrective action may be taken.

What about flux residue on the pin tips? Will it cause problems during in-circuit test?

If you use a high solids paste or you don't tell your test engineers your preferred process, the answer is yes. Ideally, conventional joints that are to be hand soldered or reflowed should not be used for test access. With a little planning at the start of a project, you can eliminate the problems before they hit the shop floor. **PIHR**

Notes: