A Practical Guide to Design Assembly and Reflow of Through Hole Components

Produced by Bob Willis

SMART Group
86 Easton Street
High Wycombe
Buckinghamshire
HP11 1LT
England

Tel: (44) 01494 465217
Fax: (44) 01494 473975
Email: info@smartgroup.org
Web: www.smartgroup.org
This is the fourth special report produced by the SMART Group for the benefit of its members and for others in the industry. As in the past the profits from the sale of this report will be donated to charity. Over the last couple of years the Group has been able to raise over £2000 from these activities.

We would like to again take the opportunity of thanking Bob Willis for personally volunteering his time and effort in producing these technology reports for our industry. Also thanks to members of the Committee for reviewing the report.

The full results and an introduction to Pin In Hole Design and Assembly can be provided by The SMART Group as an in-house Workshop. A video tape will also shortly be available to SMART Group members. For further details on reports, workshops or videos contact your local SMART Group office.

The next SMART Group report will be available in March “A Practical Guide to Double Sided Reflow Soldering” which will provide members with a special guide to processing assemblies.

Tony Gordon
SMART Group Secretary

All text and illustrations are © Bob Willis
The spread sheet example is © Alan Hobby, DEK Printing Machines
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. Issues also occur where parts are to be removed, surface mount parts may be use but with through hole adapter sockets.

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 and may leave more flux residues on the surface of the joints. Many through hole parts are used for direct contact during test and residues can quickly clog test pins.

A range of automatic selective soldering equipment is available to either semi or fully automatically solder through hole leads. This does of course require capital expenditure on equipment and extensive engineering work on programming the systems. Solder preforms are another technique that can be employed with reflow soldering. The pre-formed alloy is available in standard shaped outlines from many suppliers in different alloys. In the early days of large Telecom back planes the use of pre-formed solder donuts was 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 back plane soldering process was not very elegant and some companies still employed wave soldering even with the long wire wrap spills.

One method of soldering all surface mount and through components in a single operation is Pin In Hole Reflow, or intrusive Reflow soldering, which has steadily gained importance over the last five to ten years. This report’s author recalls the first presentation on the subject by Racal at Nepcon West, USA nine years ago. Unfortunately the paper was never published.
Introduction to Pin In Hole Reflow (PIHR)/ Intrusive Reflow Soldering (IRS)

Basically all through hole locations have solder paste applied prior to mounting the components into the through holes. This is generally done as 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. When complete, the assembly is passed through a reflow oven.

The process stages used during production of this report can be seen in the attached photographs and on the PIHR video tape available from the SMART Group.
Standard Pin In Hole Assembly Process Flow

**Single sided assembly**

Print Solder Paste on to surface mount and through hole pads

Place surface mount components

Insert through hole components

Pass assembly through reflow oven

**Double sided assembly**

Print solder face for first side surface mount pads

Place surface mount components

Pass assembly through reflow oven

Turn board assembly over

Print solder paste on to surface mount and through hole pads

Place surface mount components

Insert through hole components

Pass assembly through reflow oven

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.
**PIHR Advantages**

* Eliminates manual/wave soldering
* Compatible with current assembly process
* Minimum cost of implementation
* Potential for increased automation
* Reduction in floor space
* Reduction in capital equipment
* Only one heating process
* Use no clean technology

**PIHR Disadvantages**

* Components must be temperature compatible
* Lead lengths need to be defined
* New stencils required
* Correct PCB specification
* Modification to inspection criteria
* Customer acceptance
* Possible manual insertion
* Possible increased assembly stages

There are three alternatives to adding solder paste to the through holes for intrusive reflow soldering. The method of paste application can directly affect the volume of solder available to form the solder joint. Each of the basic methods are described below.

**PIHR Design Guidelines**

The major factors affecting the design engineer are compatibility of components with reflow temperatures and solder volume. Reference should be made to the component specification contained in this report. Although it is based on a surface mount component compatibility standard it does provide a good reference.

The solder fillet formation is controlled by the following:

* Volume of paste printed
* Metal content of paste
* Plated through hole size
* Thickness of printed board
* Printed circuit pad size
* Component lead diameter
* Component lead pitch
A basic formula most often used was originally produced by AMP Incorporated some years ago to assist design and process engineering staff when using AMP products. It also tried to meet the IPC inspection standards for solder joints produced by hand or wave soldering on standard 1.6mm thick boards. As many applications may use thinner board the process of through hole fill is simplified.

As a starting point a design engineer should consider using standard design rules for the pin to hole ratio and the pad size and compare the results and paste requirements with the formula. This also includes aperture size for resist around pads which is normally the pad size plus 0.004/0.006". This may be changed to assist paste application and damming for reflow.

Normal guidelines for pin hole assembly and soldering are the pin size plus 0.010" provides the finished hole size after drilling and plating and is the minimum hole size for round pins. The pad size is the maximum hole diameter plus twice the minimum annular ring plus any fabricators allowance. Care should be taken in discussing the capability of the hole size tolerance from the vendor. The final hole size after plating and any finishing should be specified on the PCB drawing or specification.

In the case of square or flat pins the pin size for calculation is taken across opposite corners. There are, of course, many cheaper connectors today which have punched and formed leads which may be V or U shaped but have an open section. This is difficult and will need the old engineering guesstimate!!

The thickness of the board is a another factor along with the pin length, which can be an issue in assembly. When specifying the component pin length and calculating the pin protrusion long pins can be an issue. If the pin protrusion is excessively long paste can be displaced so far away from the joint area it is difficult to reflow back to the joint. In some cases much of the solder volume may be taken coating the pin and deplete the joint area. Use as a guide the normal protrusion guidelines of between 1.00-2.00mm.

The location of via holes, test points and any other design feature needs to be borne in mind. As the solder paste will normally be printed and extend over the pad, 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.
It is difficult for the design engineer to determine what the process engineer is going to do in terms of stencil design so trials must be run to determine a no go area for printing. If this is not done then it is also possible for shorts to form between topside mounted parts and vias. In the case of through hole parts these would not be visible.

The following are the calculations which 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 and may be reviewed with reference to the articles listed in the bibliography. The spreadsheet produced by Alan Hobby of DEK may be useful and is illustrated in the following sections of this report.

One of the published formulas is shown below:

\[
\text{Volume of Paste} = (V_{\text{pth}} - V_{\text{pin}}) \times 2
\]

Where 2 is the 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.

\(V_{\text{pth}}\) is the volume of the through hole cylinder

\(V_{\text{pin}}\) is the volume of the pin cylinder

Volume of a cylinder = \(\pi R^2 h\)

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 used.

**PIHR Component Guidelines**

Suppliers have for some time realised the benefits of reflowing through hole parts. It has been process and design engineers who have been slow to see the benefits of this technique in manufacture. The strength of the surface mount joints has been long held as a reason for not using the surface mount parts. Often the best applications for surface mount parts do not require excessive strength.
Different base design on the post headers which may lead to paste displacement and solder balling.

Solder balling after reflow due to the solder/paste resist incompatibility. The surface of the mask will affect the paste’s mobility during reflow.
The use of large surface mount components do and will always suffer from coplanarity issues hence a conventional through hole part still has its place in modern designs. During reflow if the board is not supported the connector or socket may remain rigid but the board can sag leading to open joints. This is virtually eliminated with through hole leads.

The main component issue of PIHR is the higher temperature which the component may have to meet during production. Generally reflow soldering is conducted between 210°C-225°C. Parts may be exposed to peak temperature for up to 30 seconds. The IEC and IPC specifications require compatibility of all surface mount parts at 235°C for minimum of 10 seconds. It is sensible to test potential through hole parts for temperature compatibility.

Often the method used to specify component compatibility is by its material characteristics. Unfortunately this does not always work in practice. In a molded state components may be susceptible to stress cracks built up during molding. They may also change dimension due to the mould procedure, amount of filler and many other factors. The pin terminations may move in the housings leading to poor location with adjoining parts, something the material classification would not highlight. The most common areas examined after testing are dimensional changes, visual damage, blistering, warping and mating compatibility. Each of these is done in comparison with some parts not exposed to the testing procedure.

Clearance under the component should be available at the base with some form of stand-off pip or foot. A minimum of 0.015" prevents the part contacting the paste. If stand-off feet are present on the base, care should be taken to prevent them smudging the paste deposit as this will lead to solder balling. The design of the base and the paste deposit should be considered when specifying the stencil aperture. Stand off is also preferred as it may allow some degree of visibility of the joint area beneath the component to improve confidence in the overall process. It will also improve cleaning if activated fluxes are employed during production operation. When the components are selected and the paste stencil aperture is designed don’t let Purchasing change the supplier without your approval. The stand off pins on the base of the parts can be in different positions!

Ideally all components will have sealed body pins. It is possible that during reflow flux vapours could condense into open apertures adjacent to the soldering areas which may occur on top or bottom side access. These parameters should be used as an ideal benchmark when selecting suppliers.
The component packaging options should also be evaluated for automation. 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 parts. The automatic assembly of through hole parts can reduce the likelihood of displacing solder paste from the holes.

Often the most difficult problem with automatically placing parts is the incorrect packaging used by the suppliers. Next is the ability to pick the part up just like 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 socket pins or a spring clip is positioned in the centre of the part to allow vacuum pick-up.

Today most components are available in automation compatible packaging like waffle trays, tape and reel etc. but never assume, always ask. Some types of components like sockets may have guide pegs which help locate or retain the parts in position prior to soldering. If these are an interference fit this may cause problems during surface mount placement.

The solderable lead terminations need some attention. Ideally a tin/lead coating is preferred for soldering. A gold flash over 2-5um of nickel is acceptable for soldering provided the gold is below 0.5um. If brass pins are used as the base material the pin must be first plated with copper to a minimum of 2.00um before the tin/lead is applied. This is normally standard practice due to zinc migration into tin lead coatings, but some plating companies have been known to skimp on this operation.

Generally the variation on 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. If not it could change and, where design engineers have defined a smaller than usual hole size, problems could occur.

Lead length has been mentioned as it can affect solder paste push out. This problem is mostly overcome with the paste wicking back to the joint area during reflow. If the hole to lead ratio is small, or the paste has dried out, long pins can displace significant amounts of paste. As a guide the pin length protruding from the board should be 1-1.5mm 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 0.010” inch.
Stencil Printing

Solder paste is printed into through holes and over the pad surface during normal surface mount paste printing. The size of the aperture in the stencil may be adjusted to allow paste to fill the hole, cover the pad and fill the resist aperture on the board.

Provided the stencil and board are correctly positioned and all the apertures are gasketed to the pads a double print stroke may be undertaken. Printing twice on to a board without separating the PCB and stencil is an ideal way of increasing the paste volume in the hole. During the second print no further paste is applied to the surface pads but paste will flow further into the through hole barrel. Component insertion is then conducted either manually or automatically prior to the whole assembly passing through the reflow process.

Double Stencil Print

As it may be difficult to obtain the volume of solder for a through hole component if lead to hole ratio is excessive, a double print stroke operation is possible. The lowest pressures will need to be considered for this type of operation or change the pressure on the second stroke as paste will be displaced under the stencil. It is important to remember that the metal content of a paste is only half the volume of the deposit during printing printed.

Alternatively an initial printing operation may be conducted to force paste into the holes only. A second printing operation is then conducted to add additional 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.

Some companies have reversed the operation by doing the 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. The etched recesses are on the bottom of the stencil to prevent the first print being smudged. This seems unnecessarily complicated.
Through hole filled using a double print stroke during paste application. This has been viewed from the base of the board.

Round paste deposits over printing the through hole pads
Square staggered paste deposit which increases the area and the volume of solder after reflow

Round split paste deposits to aid manual assembly of components. This is not recommended due to the solder balling which may occur during reflow. It may be possible to use this technique on corner pads only to aid alignment.
In terms of process the amount of solder paste required for through hole soldering is much greater than surface mount. This will mean far more regular checks on paste on the stencil surface as it will be depleted during printing. Any automatic printer with programmed paste dispense will need to 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. This would not, however, be a problem with solder paste dispensing operations.

During any of the above methods a further problem may exist, that of under stencil contamination. If the stencil aperture for through hole parts is larger than the through hole pad it will contact the resist surface with no positive stop. This will lead to paste on the base of the stencil and require extra under stencil wipes.

**Solder Paste Dispensing**

Dispensing of the solder paste allows a greater volume of paste to be applied than with stencil printing. The only limitation is the need for a dispensing system and the possible speed of dispensing compared with printing. You are of course still going to print the board anyway unless dispensing is used as a standard process. Depending on the number of terminations the throughput may be effected in an in-line process.

Dispensing paste has always been a benefit where small volumes of any one design is produced but where there are many hundreds of different designs which will inevitably require the equivalent number of stencils. Dispensing is also very flexible as a simple program change is all that is required to change the volume of the paste rather than changing the stencil thickness or aperture.

Dispensing of paste may be used when the component has already been inserted into the board. Camelot Systems have demonstrated a number of applications where paste dispense after insertion is possible. This is illustrated on the PIHR video tape.

PIHR can be used for single sided boards where the density of surface mount parts is high or the yields from a wave soldering process have proved to be less cost effective. In this case the board or panel is first printed with paste and surface mount components are placed on to the board and refloved.
Solder paste dispensed on to a single sided board after initial reflow of the surface mount parts. This operation allows automatic insertion of through hole parts prior to the second reflow operation.

Single sided through hole joints formed after paste dispensing and reflow
The boards are then inverted and paste is applied to the surface of the board between the components previously soldered. The board is then inverted and the through hole components inserted into the through holes before passing through a reflow oven to form the joints.

Paste can be applied using multiple needles or dispensed through a single needle on to the surface of the joint. Alternatively a piston dispenser system may be used with multiple nozzles on to all terminations in one operation. A piston system is used instead of pressure. If one nozzle were to block, the air pressure applied to all the remaining nozzles would increase providing excess paste. The same type of multiple nozzle system has been employed with a paste roller. In this case a roller resembling a paint roller forces the paste through a series of nozzles on to the board surface. This is a simple process and appears to work quite well.

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

\[
\begin{align*}
LD & = \text{ Diameter of pad} \\
CH & = \text{ Height of cone} \\
L & = \text{ Lead diameter}
\end{align*}
\]

As the normal lead length is between 1-2mm the estimated height of the cone should be 0.5-1.0mm. Ideally the lead length should not be greater than 1.5mm due to concerns of the pin displacing the paste too far away from the joint area.

**Solder Preforms**

A final method of soldering through hole leaded components is by the use of specially designed solder alloy preforms. Basically this is the same pattern as the stencil would apply to the board. The pre-formed alloy is of course solder of the appropriate alloy formed into "Doughnuts". This method provides 100% metal transfer as opposed to paste which may only be 50% by volume.

During assembly either the pre-formed is positioned on the component pins or on the board surface and held in place with a flux. The pre-form may be placed on to 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.
As an alternative to the use of liquid flux with solder preforms solder paste has been used. The trials run by one company were attempting to improve solder joint quality by providing extra solder via a preform with the paste provided 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 those of the author where solder joints continually scavenged solder from adjacent joints. This resulted in both excess and insufficient joints mainly due to the linked preform not breaking evenly during reflow.

The trials did not result in a stable process and are fully detailed in an article by of Hewlett Packard which is included in the bibliography listing at the back of this report.

Soldering with preforms is hardly new; it was used by many telecommunication companies in the 1970's for soldering back plane assemblies before the introduction of press fit connectors. The use of pre-forms 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 lot of fluid hence the process can be expensive.

**Solder Paste**

Any solder paste selection is based on the smallest lead pitch on the design rather than the a PIHR product. The higher metals content between 90-92% metal by weight would be preferred for both fine pitch and through hole parts. A low residue paste would also be beneficial if cleaning is to be avoided. This reduces the possibility of test probe failure if through hole pins are to be probed during in circuit test. A low temperature metal is again preferred for conventional component compatibility. The temperature variation between the solder types is relatively small, the difference being 179-184°C.
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. This is done to increase the volume available to form the joints. Some pastes and solder mask combinations may leave solder particles behind during reflow. This may be 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 the paste coalesces during reflow. The effects can be seen in the attached photographs and on the PIHR video tape.

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.

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 processes.

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 pin insertion. This is particularly true when boards are lowered on to connectors mounted 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.

**Solder Paste Stencil**

The stencil type and specification is defined by the surface mount component 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 then a 0.006" stencil will normally be specified or a stepped foil used. A stepped stencil may be appropriate for a combination of PIH and surface mount where all surface mount parts are at 0.006" stencil thickness and the through hole are at 0.008". It is suggested that 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 surface. Printing into the resist aperture may also be required. In this case the component lead pitch may become an issue as this will limit the degree of oversized printing as it will tend to bridge between adjacent deposits. When paste bridging occurs prior to reflow it can cause scavenging from adjacent pin deposits during reflow. As the paste coalesce 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 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 most common but triangles have also been suggested to meet 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 activity. As an example, a no clean paste is better during reflow with a round than a square aperture. Solder at the corners of the deposit tend to separate and may form solder balls. In the case of water soluble paste with greater activity this is not the case. Using oblongs with rounded corners fairs well with no clean and does increase volume over round deposits. Wherever possible use the stencil thickness to increase the paste volume not the paste area.

**Manual Component Placement**

Normal surface mount placement will be accomplished using a pick and place system. The through hole parts are normally manually inserted prior to or after surface mount part placement. If the connectors are to be inserted on an in-feed or out-feed conveyor, it is useful to have a board support system in place. Often engineers have calculated the pin to hole ratio as tight as possible to aid solder formation. They have not considered variation in hole size, pin size or errors in pin location; each can affect insertion force of the components.
Minor displacement of the paste during pin insertion which will be increased as the pin length projection increases.

Excessive lead length causes paste displacement which will affect the solder joint size after reflow.
Post header showing displacement of solder paste which will lead to solder balling during reflow of the solder paste

Satisfactory assembly of a Pin Grid Array socket into solder paste prior to reflow soldering
A guide plate positioned over the board during loading can be beneficial. An operator can rest their hand on the plate and it eliminates other components being dislodged during insertion. The guide plate has cut-outs for each of the through hole parts which aids 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 is placed under the board position and as the board is positioned against the conveyor stops, light will shine through the hole if paste is not present. This technique has been used for a number of years on single sided 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.

**Automatic Insertion**

Many engineers have turned to automation for through hole insertion in the PIHR process as it eliminates manual assembly issues. It is fair to say that most companies who adopt the assembly process will not be able to justify automatic assembly and will only use manual techniques.

Although vacuum pickup tooling may be used on small parts most automatic insertion systems for PIHR have in the past been robotics insertion systems specifically designed for the purpose with clamp tooling. Examples of machine suppliers involved in odd form insertion are Universal and PMJ.

The following is an example specification for through hole insertion equipment for odd form component insertion provided by Universal Instruments.

<table>
<thead>
<tr>
<th>Specification</th>
<th>Minimum</th>
<th>Maximum</th>
</tr>
</thead>
<tbody>
<tr>
<td>Width/Diameter</td>
<td>1.016mm</td>
<td>38mm</td>
</tr>
<tr>
<td>Length</td>
<td>2.54mm</td>
<td>127mm</td>
</tr>
<tr>
<td>Height</td>
<td>1.53mm</td>
<td>50.8mm</td>
</tr>
<tr>
<td>Weight</td>
<td>0.454kg</td>
<td></td>
</tr>
<tr>
<td>Hole diameter</td>
<td>0.76mm</td>
<td>2.03mm</td>
</tr>
<tr>
<td>Minimum lead hole</td>
<td>0.43mm</td>
<td>plus lead</td>
</tr>
</tbody>
</table>

The Universal system requires a topside clearance of 50.8mm and bottom side 25.4mm.
Fairly recent work has resulted in a tape and reel system for large through hole components to be introduced. The system has been adopted by companies like Universal Instruments, Quad, and Panasonic. Prior to that time various of 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 1500 components per hour can be achieved. Again the component design may make pickup difficult, hence the need for design for manufacture. If little thought is put into selection of the components then problems will occur.

Due to the size of many of the through hole parts there is a limit to the number of parts held on an oddform insertion system. In reality there are very few through hole parts to insert so machine capacity is not normally an issue.

Most machines can develop insertion forces of 10kg which should be considered in detail as excessive force will displace other components. The tack of the paste must hold the parts in place if the parts are placed prior to insertion.

If the through hole parts are inserted prior to insertion of the surface mount parts the height of the conventional parts needs to be considered. Placement of the surface mount parts may have restricted access. The placement speed can also be affected as 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.

**Reflow Soldering**

The following section includes a guide to setting up a reflow oven for production. In the case of through hole components there is little change in the philosophy of profiling a board. In the case of connectors, sockets or transformers, then it is the temperature under the parts which may be the significant factor. Like Ball Grid Array (“BGA “) the component body will mask the joint areas to some degree, that is why it is necessary to monitor temperature under the parts during profiling as the conventional parts will most commonly be reflowed when on the top of the board.

Peak temperature and time of peak temperature should be investigated closely due to the differing requirements of conventional components. To aid setting process parameters a DataPaq, MOLE, KICK or similar multiple point profiler should be used. Four probes may be used for normal surface mount parts and two specifically used for through hole parts.
Solder balling caused by poor component and paste design resulting in the paste being displaced prior to the reflow operation and not being able to coalesce back to the joint area.

Uneven solder reflow and wetting on gold plated through hole joints which may be due to excessive paste displacement during component insertion.
A further modification to the process which may benefit component selection is the use of a lower temperature solder alloy employed in the solder paste. One example is included in the bibliography where a company used 43Sn/43Pb/14Bi which has a melting range around 165°C. This is worth considering if the components required cannot stand the temperatures normally associated with reflow.

One issue to bear in mind is when paste is printed and reflowed on a non-metallic surface it can solder ball. A couple of examples are on flexible circuits where the copper has been etched from the polyamide surface. Some of the bonding resin may be left behind causing balls to stick and not coalesce. The same may also be true with some other laminates as the surface of the substrate is seldom smooth after the copper has been etched from the surface.

**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 or 184°C. When the solder is in the liquid state a solder joint will form between the two surfaces.

The speed of wetting will depend 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. Further details on reflow and profiling are contained on the EPS video tapes on profiling reflow ovens.

**Board Support**

The printed circuit board should remain as flat as possible throughout the first or 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.

With modern convection ovens the convected air or nitrogen should not disturb components or cause the printed board to flex. This can occur with thin boards of less than 0.5mm like PCMCIA panels. Ideally all reflow ovens should be able to adjust the level of convection rates to minimise component movement.
The 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 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 will be based on an existing profile for a similar board design. The speed of the conveyor will be adjusted to the desired assembly throughput of the oven. This may be limited by the type and length of the unit.

**Profiles and Thermocouples**

Thermocouples will be 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 terminations it may affect the readings. 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 is a mark of a good reflow oven. This should be part of the initial oven evaluation and understood by production staff.

Next the first board with thermocouple leads attached may be passed through the oven and the temperature profile analysed. Adjustment may 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 manufacture, the component suppliers guidelines and the printed board 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.

To conduct the reflow operation correctly it is important to know what temperatures are being seen by the whole board assembly. This requires the use of thermocouples to monitor selected solder terminations. In the case of surface mount parts the thermocouple beads are soldered directly to the joint surface using high temperature solder.
Solder joint formed with solder paste and reflow soldering

Microsection of through hole joint showing minor voiding and 95% solder fill after reflow of the paste
With Ball Grid Array (BGA) the lead must be positioned under the centre of the device. In most cases these are the last terminations to reflow during soldering. Either thin wire is used or more commonly, a profile board is produced with a thermocouple wire mounted through the board into a ball termination to improve the repeatability of the temperature measurement.

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 then they should be used during profiling. The pallets will contribute to the mass and hence affect temperature rise on selected areas in contact with the board. It can easily affect the temperature rise by as much as 20°C.

When a profile has been established then the board should be run through the oven again monitoring the profile but 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 profile has been established and been run in production with satisfactory soldering results the following information should be retained. The solder temperature in each zone, the speed of the conveyor, the extraction rates and the board loading. A temperature profile should be run on the oven initially each day to build up a picture of the process stability. The frequency may then be adjusted depending on the repeatability of the results.

Further trials should also be run on the desired profile with the production paste to determine the degree of slumping of the paste as it will affect solder shorting. Lower 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 chip components. This test is very useful to understand many of the causes of solder beading on chip devices.

Even with the best convection oven there is a difference in peak temperature or duration between different board assemblies. So don't be a Lazy Engineer with a single profile be a Great Engineer and learn more about your process.
Process Trials Procedures.

Standard trials are often conducted on reflow ovens by production engineers during product assessment, machine approval or in process set-up. The following trials are also used by machine suppliers during equipment development.

Temperature Uniformity

Measure the surface temperature on an assembly or ideally on a blank laminate test board to obtain any variations across the complete conveyor belt width. This will show any peaks or low points between centre or near the edge of the conveyor.

Test results ideally achieved to be between 5-10°C

Thermal Loading

First a temperature profile is produced as a reference using six thermocouple probes soldered to the assembly, three on the top and three 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 a further 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.

Test results achieved ideally less than 10-15°C

Temperature Stability

Measuring surface temperature on an assembly or test board 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.

Test results achieved ideally less than 10-15°C
Solder joints formed using solder pre-forms showing scavenging of the solder between two pins. This occurs due to inconsistent reflow of the connecting links on the pre-form and is a common problem.

Solder joints showing flush joints but 100% solder fill on the base of the board.
**Throughput Speed**

Adjustments are made to the conveyor speed for the maximum envisaged circuit board throughput requirements. The preferred temperature profile for the most complex product is then the goal. Reference must be made to the paste or adhesive requirements when considering these tests.

Evaluating nitrogen usage must be conducted with discussions on existing users of machines for consumption and maintenance. The use of nitrogen has benefits but needs to be justified. Solderability assessment of surface pads is a good measure of the benefits of nitrogen comparing samples before and after reflow in nitrogen.

**Solder Pallet Guidelines**

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.

Using a pallet the leaded connectors are pre-located in the pallet and the board is lowered and located on to the pallet. The weight of the board and subsequent components guarantees the connectors are flush with the base of the board. Also the connector base design is less critical as 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 prior to the pallet passing through the reflow oven for soldering. This technique provides some protection to the connector in terms of peak temperature. It is also possible to have the large parts pre-loaded to the pallet with the board passing the pallet through the placement system. This may avoid problems of manual location of parts and misplacing surface mount components prior to reflow.

To aid alignment of the board to the pallet it is advisable to have some tapered tooling pins inserted in the pallet. These should align the board prior to the connector pins contacting the base of the board. This will greatly assist printed board loading and may be left in place or removed before proceeding. In this case the connector pin length is quite critical for PIHR. Connectors with a large amount of pin float can be a real problem for lead location and should be avoided.
Care must be taken during profiling the reflow oven as a pallet will always have some effect on the circuit board temperature, possibly delaying reflow. When profiling a check should be made on the peak temperature at the base of the connector to allow complete reflow of the pin and solder pad in this area.

A spin off benefit to this type of design and method of build is to the design engineer. Normally this type of board with surface mount and a large number of connector 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 a lot more restrictions on design in terms of parts which can be used and component orientation. Further advice on pallet design rules may be obtained from a guide written by Peter Swanson and Bob Willis which is available from Intertronics.

**Solder Joint Inspection**

Many companies have been put off the use of PIHR process due to the variation in solder volumes seen after reflow soldering. 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 hole. Most specifications today are practical in their approach and have a minimum solder fill of 75% of the board thickness. Two companies have a specification for intrusive reflow of 50% of the board thickness.

In practical terms, if reliability testing were conducted on example joints, none of the above would fail if correctly soldered. Conducting pull strength measurement is impractical for most pin in hole applications. Most joints will develop pull strength figures in excess of 10-15kg in line with a wave solder fillets. By way of a comparison a surface mount gull wing lead will be between 800-2.2kg based on pitches of 0.020-0.050" components.

The best and most practical approach is to conduct some microsections of selected joints. This will provide confidence and evidence of complete reflow with full wetting. Example of joints produced during this study are shown in the attached photographs.
Incomplete reflow and wetting of the solder paste shown during microsectioning of a joint. The top of the joint area is often the coolest part of the board during reflow. The through hole parts often hold down the temperature of the board locally and should be considered during normal profiling.

Dried solder paste displacement caused during component pin insertion which can lead to solder balling or shorts. In this case the connectors were pre-loaded to a pallet and the board assembly lowered on to the pins. The board had been left for a couple of hours prior to assembly.
X-ray examples of connector pins examined after reflow. Minor voids are visible in the solder joints.
X-ray examples of connector pins examined after reflow. Minor voids are visible in the joints.
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 the solder joint reliability and has very little effect on pull strength. Voiding is a process issue which 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 process of reflow is not considered properly. Correct profiling can virtually eliminate voiding in PIHR process. X-ray inspection is a method which may be used to examine through hole joint as is microsectioning. X-ray is, of course, non-destructive and a number of examples of joints are included here for reference.

"If voids are an issue in PIHR consider if your company has done anything to eliminate traditional through hole outgassing, if not why worry?" - Bob Willis

It is important during the initial PIHR trials to involve quality engineering so they can establish inspection criteria for production staff. This should be part of a training exercise undertaken and implemented prior to production. This eliminates the 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. The photographs included on the CD ROM may be useful in setting inspection criteria, alternatively the inspection standards available from the SMART Group can be applied.

Currently the IPC are looking at the inspection criteria for pin in hole reflow assembly. The existing standard, if applied sensibly, would be suitable but if the document is taken word for word PIHR joints would not be acceptable. Fortunately engineers have probably been indicating these issues to IPC and changes will no doubt be made to the existing criteria.
Satisfactory solder joints on a Pin Grid Array Socket visible after reflow soldering

Pin header lifted after reflow due to poor initial insertion of the part
In Circuit Test

In the case of test access joints which have been soldered by PIHR these can be treated like any other through hole soldered joint. They may be used as test access points during in circuit test. They may be probed with suitable test pins to aid test access. The only possible consideration is the amount of flux residues on the surface of the joint. There should be no more flux than on a hand soldered joint but there may be more than with a wave soldered termination using a no clean flux. Increasing test pressure or using different test pins will improve test probe penetration. Test fixtures and probes should always 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.

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

*We hope this process guide will be useful for your company’s implementation of PIH or Intrusive reflow. The video tape may be useful to support the introduction of this assembly technique. The tape illustrates live many issues raised. If there are any further points you find during production which would be useful to future readers please let the SMART Group office aware. The SMART Group, like any other non-profit voluntary body, are only as good as its members.*
Universal Instruments Oddform Insertion system

Close up view of component insertion
Close up view of component insertion on a Oddform Insertion system supplied by PMJ
PIHR Bibliography

The following are the known published articles, videos and reports on Pin in Hole Reflow or Intrusive Reflow Soldering. Copies of text should be available from the paper’s author, the technical publication where it appeared or though a technical library. The text is NOT available from the SMART Group office or this report’s author.

The video tape and CD ROM on PIHR and how to adopt the process is available from the SMART Group or from Electronic Presentation Services.

Technical Papers

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

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

"Advanced Surface Mount Manufacturing Methods" by Joe Belmonte, MPM Corp

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

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

“Reflow of Solder Pre-form Arrays in Nitrogen Reflow” by Dan Uno, Hewlett Packard, Palo Alto, California

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

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

“Paste Printing for Through Holes Components" by Ray P Prasad, SMT magazine
"Design Characteristics of a Surface Mount Compatible Through Hole Connectors" by Martha L Rupert, AMP Inc.

Training Video

"Introducing Pin In Hole/Intrusive Reflow Soldering" by Bob Willis, Electronic Presentation Services available from the SMART Group office

CD ROM Photo Album

“Pin In Hole and Intrusive Reflow Photo Album” by Bob Willis, Electronic Presentation Services available from the SMART Group office

If the reader is aware of any further articles on PIH or Intrusive Reflow the SMART Group would appreciate a copy of the article or further information to include in this report’s bibliography.
**Example of Excel Spreadsheet for Predicting Solder Joint Quality**

The following spreadsheet was produced by Alan Hobby of DEK Printing Machines where a copy of the spreadsheet may be obtained. The instructions for the use of the spreadsheet were written by the report’s author so he could easily describe the use of the spreadsheet in this report.

The spreadsheet provides an indication of the 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 reports author’s to assist the completion of the form and to better understand the resulting information.

**Column**

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Enter the board thickness. All dimensions should be entered in metric.</td>
</tr>
<tr>
<td>2</td>
<td>Enter the stencil thickness you intend to use in production.</td>
</tr>
<tr>
<td>3</td>
<td>Enter the component reference number. This may be used for later reference to avoid reassessing components in the future.</td>
</tr>
<tr>
<td>4</td>
<td>Enter a description of the component.</td>
</tr>
<tr>
<td>5</td>
<td>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 level 2.</td>
</tr>
<tr>
<td>6</td>
<td>Enter the final required size for the plated through hole.</td>
</tr>
<tr>
<td>7/8/9</td>
<td>Enter details of the stencil aperture size - <strong>either</strong> the diameter for a round aperture <strong>or</strong> the length and width if square/oblong.</td>
</tr>
</tbody>
</table>

10/11/12 Enter **either** the diameter of round pins **or** width and material thickness in the case of square/oblong terminations.

After entering all these dimensions the spreadsheet displays the pin volume and hole volume. The calculated solder volume is provided for the hole, the print and total is calculated from the stencil thickness and aperture dimensions. The spreadsheet also displays the difference between the solder and the annulus.
<table>
<thead>
<tr>
<th>Board thickness, mm</th>
<th>Stencil thickness, mm</th>
<th>Component number</th>
<th>Component description</th>
<th>Predicted result (1=good)</th>
<th>Hole diameter, mm</th>
<th>Print (length, mm)</th>
<th>Hole diameter, mm (thickness, mm width, mm)</th>
<th>Pin Volume</th>
<th>Pin Volume</th>
<th>Annular Volume</th>
<th>Solder metal Volume</th>
<th>Difference between good and actual</th>
<th>Good = 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.6</td>
<td>0.153</td>
<td>1</td>
<td>Pin Header</td>
<td>1</td>
<td>1</td>
<td>2</td>
<td>0.8</td>
<td>1.2566</td>
<td>0.804</td>
<td>0.4524</td>
<td>0.628</td>
<td>0.24</td>
<td>0.9</td>
</tr>
<tr>
<td>1.6</td>
<td>0.153</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1.6</td>
<td>0.153</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1.6</td>
<td>0.153</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1.6</td>
<td>0.153</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1.6</td>
<td>0.153</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1.6</td>
<td>0.153</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1.6</td>
<td>0.153</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1.6</td>
<td>0.153</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1.6</td>
<td>0.153</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1.6</td>
<td>0.153</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1.6</td>
<td>0.153</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1.6</td>
<td>0.153</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1.6</td>
<td>0.153</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1.6</td>
<td>0.153</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
PIHR Frequently Asked Questions

The following are a list of typically asked questions on PIHR and the answers often provided. These may be beneficial as a quick guide to the process and any questions you may have. They have all appeared in the AMT, EM&T and Asian Electronics Engineer magazines.

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 soldering is to be eliminated then 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. This is conducted 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 though hole printing one for through hole and one for traditional SMT parts?

Depends if you are feeling charitable to your stencil supplier. Some people have used the technique to increase paste volume.

(Bob's Note) Make sure you tell your stencil supplier that the round apertures are required in your stencil for through hole printing. They are so often removing them for customers who have not supplied a solder paste file for the stencil, those wonderful 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 needs to be considered for this process and needs to be 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 specifically produced for reflow applications.
What is the best use of this technique?

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.

What will the solder joint reliability be like for 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. The microsection I have done looks good.

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. Care on setting process parameters will keep this to a minimum. Voids have been seen to improve reliability not reduce it!!! Think of honeycomb structures.

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

Yes, some paste is forced on to the tips of the pins. The amount is dependent on the care during component loading.

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

As reflow takes place the solder does remain 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 which way up the pin is during reflow.

What about flux residues 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 will be yes. 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 eliminate the problems before they hit the shop floor.