A Practical Guide to Design & Assembly Using Double Sided Reflow

Produced by Bob Willis

Produced By:

Bob Willis
2 Fourth Avenue
Chelmsford
Essex CM1 4HA
England.

Tel: (44) 01245 351502
Fax: (44) 01245 496123
Email: bob@bobwillis.co.uk
Web: www.ASKbobwillis.com
Web: www.leadfreesoldering.com

Bob Willis General Profile & Lead-Free Specific

Involvement in Lead-Free Process Development

Bob Willis has been involved with the introduction and implementation of lead-free process technology for the last seven years. He recently received A ***SOLDERTEC/Tin Technology Global Lead-Free Award*** for his contribution to the industry, helping implementation of the technology. Bob has been a regular contributor to Global SMT magazine for the last five years. He was responsible for co-ordination and introduction of the first series of hands-on lead-free training workshops in Europe for ***Cookson Electronics during 1999-2001***. These events were run in France, Italy and the UK and involved lead-free theory, hands-on paste printing, reflow, wave and hand soldering exercises. Each non commercial event provided the first opportunity for engineers to get first hand experience in the use of lead-free production processes and money raised from the events was presented to local charity. More recently he co-ordinated the ***SMART Group Lead-Free Hands On Experience*** at Nepcon Electronics 2003. This gave the opportunity for over 150 engineers to process four different PCB solder finishes, with two different lead-free pastes through convection and vapour phase reflow. He also ran the ***Experience 2 & 3*** in 2004/2005. 2006 sees Nepcon back at Birmingham and Bob will again be organising the features.

He has also run training workshops with research groups like ***ITTF, SINTEF, NPL & IVF*** in Europe. Bob has organised and run three lead-free production lines at international exhibitions ***Productronica***, ***Hanover Fair*** and ***Nepcon Electronics*** in Germany and England to provide an insight to the practical use of lead-free soldering on BGA Ball Grid Array, CSP Chip Scale Package, 0210 chip and through hole intrusive reflow connectors. This has resulted in technical papers being published in Germany, USA and the United Kingdom. Bob also defined the process and assisted with the set-up and running of the first ***Simultaneous Double Sided Lead-Free Reflow*** process using tin/silver/copper for reflow of through hole and surface mount products. This year 2005, he will be running a Lead-Free Production and Seminar feature at Productonica in Munich Geramany with Global SMT magazine.

Bob also had the pleasure of contributing a small section to the first Lead-Free Soldering text book ***"Environment - Friendly Electronics: Lead-Free Technology"*** written by ***Jennie Hwang*** in 2001. The section provided examples of the type of lead-free defects companies may experience in production. Further illustrations of lead-free joints have been featured in here most recent publication "***Implementing Lead-Free Electronics"*** 2005.

Mr Willis led the *SMART Group Lead-Free Mission to Japan* and with this team produced a report and organised several conference presentations on their findings. The mission was supported by the DTI and visited many companies in Japan as well as presenting a seminar in Tokyo at the British Embassy to over 60 technologists and senior managers of many of Japans leading producers.

Bob was responsible for the *Lead-Free Assembly & Soldering "CookBook" CD-ROM* concept in 1999, the world's first interactive training resource. He implemented the concept and produced the interactive CD in partnership with the *National Physical Laboratory* (NPL), drawing on the many resources available in the industry including valuable work from NPL and the DTI. This incorporated many interviews with leading engineers involved with lead-free research and process introduction; the CD-ROM is now in its 3rd edition.

Bob has recently produced three new lead-free interactive CD-ROMs with Soldertec Global/Tin Technology covering *Hand, Wave and Reflow Soldering* each CD introduced by Kay Nimmo world leading expert on lead-free and the WEEE and RosH legislation.. These CDs complement the range of lead-free training CD-ROM offered by Bob who has just introduced a CD entitled *PCB Design, Layout, Assembly and Lead-Free Defect Guide*.

Recently Bob has produced one of the first set of *Lead-Free Inspection Wall Charts* covering reflow and wave solder joints using lead-free terminations and different alloys and PCB finishes. New sets recently introduced cover *BGA X-Ray Inspection* & *BGA Optical Inspection.*

Although the problems associated with fillet lifting of through hole joints have been well documented by many researchers, it was Bob Willis who highlighted the same problem could exist with pin in paste/intrusive reflow and selective soldering processes. He demonstrated that the problem could occur with each of the common lead-free alternative alloys, but despite its poor appearance provided reliable joints even after 2000 thermal cycles. He has recently produced video simulations of fillet lifting to help understand the way fillet lifting occurs, similar to the work done in the US by NIST. Bob has conducted workshops on lead-free production process for *IPC, APEX & Nepcon Exhibitions* in the USA as well as SMT Nuremberg and Productronica, Germany and *Nepcon Malaysia*. In addition Bob has coordinated the annual *SMART Group Lead-Free Update Seminars* with the SMART Group PR Director, Mike Judd for the last six years. He has also assisted with the launch of two *DTI Lead-Free Reports* written by representatives of Soldertec global and NPL at two Nepcon Exhibitions.

Currently Mr Willis is supporting the NPL *"Lead-Free Masterclasses*" workshops on design, manufacturing and rework which are being presented around the UK. These workshops are sponsored by EM&T magazine.

Bob Willis General Profile

Bob Willis currently operates a training and consultancy business based in England. Bob is the Technical Director of the SMART Group and a member of the technical committee. Although a specialist for companies implementing Surface Mount Technology Mr Willis provides training and consultancy in most areas of electronic manufacture. In the last 10 years focusing on lead-free manufacture which has eared him the SOLDERTEC/Tin Technology Global Lead-Free Award for his contribution to the industry He has worked with the GEC Technical Directorate as Surface Mount Co-Ordinator for both the Marconi and GEC group of companies and prior to that he was Senior Process Control Engineer with Marconi Communication Systems, where he had worked since his apprenticeship. Following his time with GEC he became Technical Director of an electronics contract manufacturing company where he formed a successful training and consultancy division.

As a process engineer, he was involved in all aspects of electronic production and assembly involved in setting up production processes and evaluating materials; this also involved obtaining company approval on a wide range of Marconi's processes and products including printed circuit board manufacture. During the period with Marconi, experience was gained in methods and equipment for environmental testing of components, printed boards and assemblies with an interest developed in many areas of defect analysis. Over the last 15 years he has been involved in all aspects of surface mounted assembly, both at production and quality level and during that time has been involved in training staff and other engineers in many aspects of modern production.

Over the past few years Mr. Willis has travelled in the United States, Japan, China, New Zealand, Australia and the Far East looking at areas of electronics and lecturing on electronic assembly. Mr. Willis was presented with the Paul Eisler award by the IMF (Institute of Metal Finishing) for the best technical paper during their technical programmes. He has conducted SMT Training programs for Texas Instruments and is currently course leader for Reflow and Wave Soldering Workshops in the United Kingdom. Mr Willis is an IEE Registered Trainer and has been responsible for training courses run by the PCIF originally one of Europe's largest printed circuit associations.

Bob has conducted workshops with all the major organisations and exhibition organisers World Wide and is known for being an entertaining presenter and the only presenter to use unique process video clips during his workshops to demonstrate each point made.

Mr. Willis was Chairman of the SMART Group, European Surface Mount Trade Association from 1990-94 and has been elected Honorary President and currently holds the position of SMART Group Technical Director, he also works on BSI Standards Working Parties. He is a Fellow of the Institute Circuit Technology, an NVQ Assessor, Member of the Institute of Quality Assurance and Society of Environmental Test Engineers. Bob Willis currently writes regular features for AMT Ireland, Asian Electronics Engineer and Circuits Assembly the US magazine. He also is responsible for writing each of the SMART Group Charity Technology reports, which are sold in Europe and America by the SMTA to raise money for worthy causes. Bob Willis most recently helped organise the SMART Group-Lead Free Mission to Japan to examine and report on the current state of lead-free research and implementation of lead-free processes. Bob ran the SMART Group PPM Monitoring Project in the United Kingdom supported by the Department of Trade and Industry. He now is coordinator of the LEADOUT Project for the SMART Group.

Bob Willis

Tel	+(44) 1245 351502	Fax	+(44) 1245 496123
E	bob@bobwillis.co.uk		
W	www.leadfreesoldering.com	W	www.ASKbobwillis.com

A Practical Guide to Design & Assembly Using Double Sided Reflow

Introduction

Surface mount is a well established production process used in all types of market sectors. Most of the processes have been well documented apart from (DSRS) Double Sided Reflow Soldering. The process of soldering both sides of an assembly has been practised for many years but few engineers have documented the techniques used.

This report hopes to bring together documented information and the results of process trials and potential problems faced by the process engineer tasked with introducing DSRS.

DSRS Process Sequence

Basically the assembly process typically used for DSRS is just a repeat of the process used for single side reflow soldering. The process typically only requires a screen printer, placement system and a reflow oven. Placement may be done manually if the volumes are low with less than 250-350 components placed per hour. The process flow is shown in the following diagram.

Normal Process Stages for DSRS

Solder paste application (63% tin/37% lead)

↓

Surface mount component placement

↓

Reflow solder first side
(Possibly with Nitrogen)

↓

Turn the board over

↓

Print solder paste (63% tin/37% lead)

↓

Place surface mount components

↓

(Possible insertion of through hole components)

↓

Reflow solder second side
(Possibly with Nitrogen)

↓

Possible assembly of conventional components

↓

Hand solder conventional components

The conventional components may be assembled and soldered using the paste in hole/intrusive reflow process provided the components are process compatible. The methods used for PIHR (Pin In Hole Reflow) are fully documented in another charity report on the subject available from the SMART Group office.

Two alternative methods to the traditional assembly process for double sided reflow are currently practised in the industry. As you will see in the survey at the end of this report they are still in the minority and the process flow charted on the previous page is the most common.

To eliminate the problems associated with secondary reflow on the bottom side, some engineers have employed different solder paste alloys. Solder paste which has a higher melting temperature than the traditional 63/37 tin/lead alloy has been used on the first side assembly. Either 63/37 or 62/2/36 alloys has been used on the second side; this reflows at 184^{o}C and 179^{o}C respectively. One example is where a company used 43Sn/43Pb/14Bi which has a melting range around 165^{o}C on the second side. This is also worth considering if the components required cannot stand the temperatures normally associated with reflow.

Some engineers have used adhesive on the first side; this is applied after paste printing between the printed pads before component placement. During the reflow on the first side the adhesive is also cured providing a strong bond as well as the surface tension effect during second side reflow. This prevents the parts being lost during reflow even if a mesh belt is being used to transport the boards during reflow.

It is understood that Phillips and AT&T and some companies in Japan have been conducting trials on simultaneous reflow of both sides of the board using adhesive and solder paste. In this case the solder paste is applied and then the adhesive dispensed prior to placement. It is not clear from the information available but a fast UV cure cycle for the adhesive could be conducted prior to repeating the cycle for second side assembly. The boards would then travel through reflow to simultaneously reflow both top and bottom parts in one operation. The process may be feasible but for most companies it would be a nightmare for process control and transportation.

In realistic terms this process may be feasible with small passive parts but designers require greater flexibility not less than they currently have with wave soldering. The design engineer wants to be able to place any component on both sides of the board.

Obtaining reflow on both sides of the board at the same time is not as easy as first it may seem. This is due to the oven's configuration and the greater differential that may be seen between bottom and topside designs.

Using this process can promote voiding and solder inclusion into adhesive deposits and the adhesive contaminating solder joints. Fast curing, as with reflow soldering, can also promote outgassing due to any moisture in the adhesive.

This adhesive process has also be used when wave soldering is to be applied in manufacture. After both reflow operations conventional components are inserted manually and the assembly may be wave soldered with a conventional single wave system. As the solder joints have already been formed during reflow it eliminates the problems of solder skips (unsoldered surface mount joints) and reduces dross generation associated with double wave systems. Both these approaches are ***Not Recommended*** by the author. They have been used to eliminate the need for two solder waves which is the ideal method when surface mount parts are to be used on the base of the board.

Single sided SMT assembly

Double sided SMT assembly

Double sided SMT assembly with reflow soldered through hole parts

PCB Design Rules

The use of double sided surface mount board designs provides the maximum design density possible. All traditional surface mount design rules applied to reflow designs should be followed. A video on Surface Mount and Conventional Design for Manufacture is available from the SMART Group. This covers all the traditional layout rules. It is often traditional practice to reduce the possibility of any problems with double sided assembly by ideally placing all large components above SOIC level on one side of the board, so that components over 20-30 grams are eliminated from the bottom side during second side reflow.

It is possible to increase surface tension forces by increasing the size of pads and with it the amount of paste. This may cause problems with solder shorting during reflow on closely spaced terminations below 0.025”. In the case of large parts on a 0.050” pitch this should not be an issue.

In the case of large boards or boards under 1.0 mm thick the ability to support the board should be considered. Reflow ovens can provide support pins, wire or tape which can support the centre of the board during first side reflow. This reduces sagging of the board and the associated problems of trying to print and place on the second pass. This support may also be used on second side assembly if access to the central base of the board is available. As a guide, a gap of 3-4mm is required along the approximate centre line of the board. Engineers should also consider the component height below the board as there may well be a limit to the centre board support drop clearance on some reflow ovens.

Some reflow suppliers have also introduced special clamping fingers to overcome warpage. Warpage of a board can be at the centre of the board or a twisting action around the centre point of the board. By clamping the edge of the board but still allowing the board to expand in its width may improve the overall flatness of the board after soldering.

Normally a 5mm no go area is placed around the board edge to allow conveyorisation, through hole automation, pin transfer during reflow and mounting boards in cassettes from a conveyorised process. When assembling double sided assemblies on a conveyorised process, the cassette container pitch may also be an issue. Ideally, for maximum capacity, a board should be in every cassette slot but if the height of the components is excessive, which can happen with through hole parts, the automatic cassette loader may require a clear slot between each board.

When processing thin boards <1.00mm there is a rigidising effect from the components that are soldered to the first side. On second side reflow the rigidising helps support the board during reflow. This is only true if the solder does not go into a liquid state. If the solder joints do enter the liquid phase sagging can cause component loss and is probably one of the most common reasons for component movement and missing parts.

Calculation of Surface Tension

One method which has been developed for DSRS process compatibility compares the weight of the component against the total pad mating area. When calculating this, provided the grams per square/inch are equal or greater than 30, the component may be placed upside down during second pass reflow. Even if the joints do go into a liquid state the surface tension should be sufficient to hold the parts if no other forces are applied.

$$\frac{\text{Weight of component (grams)}}{\text{Total pad mating area (square inch)}}$$

If we take a 84 pin PLCC as an example the component has a weight of 10 grams. The typical pad size for each of the terminals is 0.018” x 0.086” which equals 0.0015 square/inches. Multiply 0.0015 by 84 gives the total pad area in square inches. Dividing component weight 10 grams by 0.130 results in 76 grams/square inch.

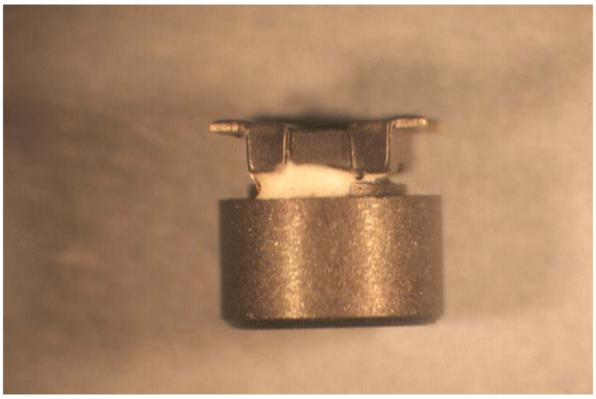
If we take a SOIC16 as an example the component has a weight of 4 grams. The typical pad size for each of the terminals is 0.094” x 0.024” which equals 0.0023 sq.in. Multiply 0.0023 x 16 gives the total pad area in sq.ins. Dividing component weight 4 grams by 0.036 results in 111 grams/sq.in.

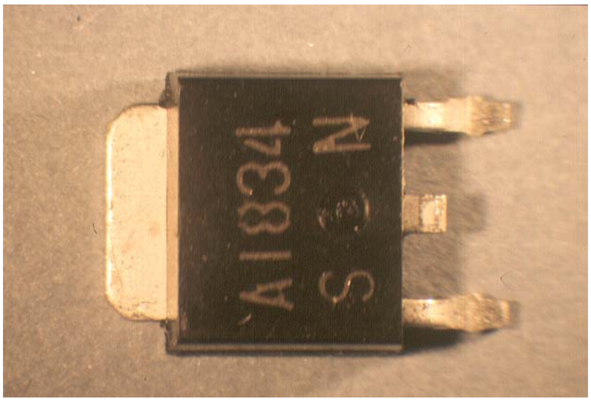
Guidelines provided by Soltec, a major supplier of reflow ovens, suggest a figure of 50mg/mm^2 (component weight/paste or pad area) for successful double sided reflow.

Component Selection

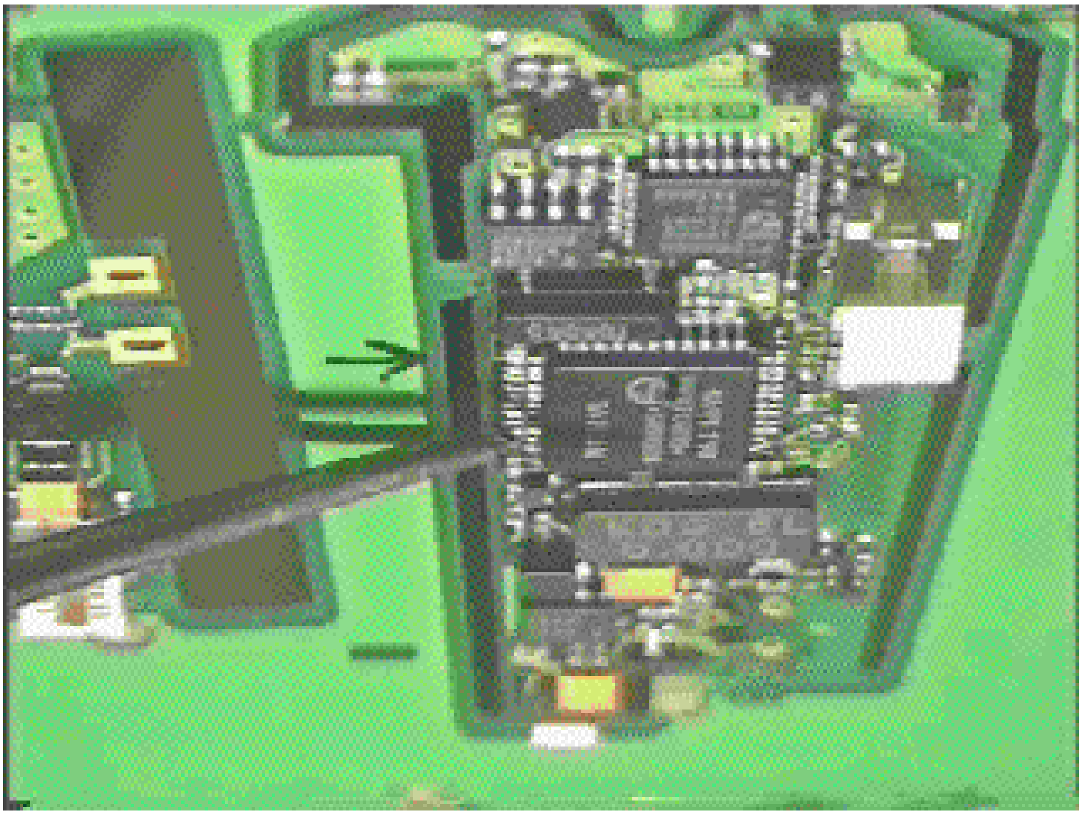
Probably the most significant issue relating to the component is individual weights and the surface area of the terminations soldered to the printed board surface. Often engineers focus on the weight but unbalanced parts can be just as much of an issue.

Normally components have terminations on two or four sides but some components have unevenly balance terminations like the examples shown below. During double sided reflow the weight of the component would not be evenly distributed causing movement on one side and then possible separation from the base of the board.





The two photographs above show examples of components which have unbalanced terminations. The ratio of the part to termination area is excessive which may lead to uneven weight distribution. Other ceramic and metallic devices are susceptible to separation from the board.



Example of component on the base of the board just prior to displacement using a non metallic probe. The solder joints were in a liquid state during probe contact and initially failed to displace the component indicating the surface tension forces that are present even in a liquid state. Again, this demonstrates the effectiveness of double sided reflow and the potential reliability of the process. The live video examples are available on the SMART Group Double Sided Reflow Assembly tape.

The following table provides a list of common surface mount components and selected component weight; it also includes the number of terminations.

Chip 0805	0.007g	2	PLCC28	0.688g	28
Chip 1206	0.009g	2	PLCC44	2.21g	44
Chip 1210	0.012g	2	PLCC68	4.67g	68
SOT23	0.008g	3	PLCC100	9.97g	100
SOT89	0.0976g	4	LCCC44	5.32g	44
V/Resistor	0.124g	3	PLCC84	5.21g	84
MELF	0.1325g	2	PLCC100	9.97g	100
Mini MELF	0.031g	2	QFP100	4.24g	100
SOIC8	0.102g	8	BGA225	2.65g	225
SOIC12	0.123g	12	BGA313	6.32g	313
SOIC16	0.142g	16	CBGA256	24.21g	256
TSOP20	0.212g	20	CCGA625	29.53g	625

CCGA1089	34.12g	1089
Tape BGA360	2.695g	360

Surface Tension Assessment

There are two distinct engineering methods of assessing double sided reflow and component compatibility:

Measure the surface tension effects
SISIIW (Suck it and See if it Works)

Today in the industry both methods are commonly practised and many people are doing double sided reflow and just because it works they do not have to know why it works. Other engineers like to know why it works so that when problems arise they can pinpoint what has changed in the process. Just ask yourself “which engineer would I like to be?”

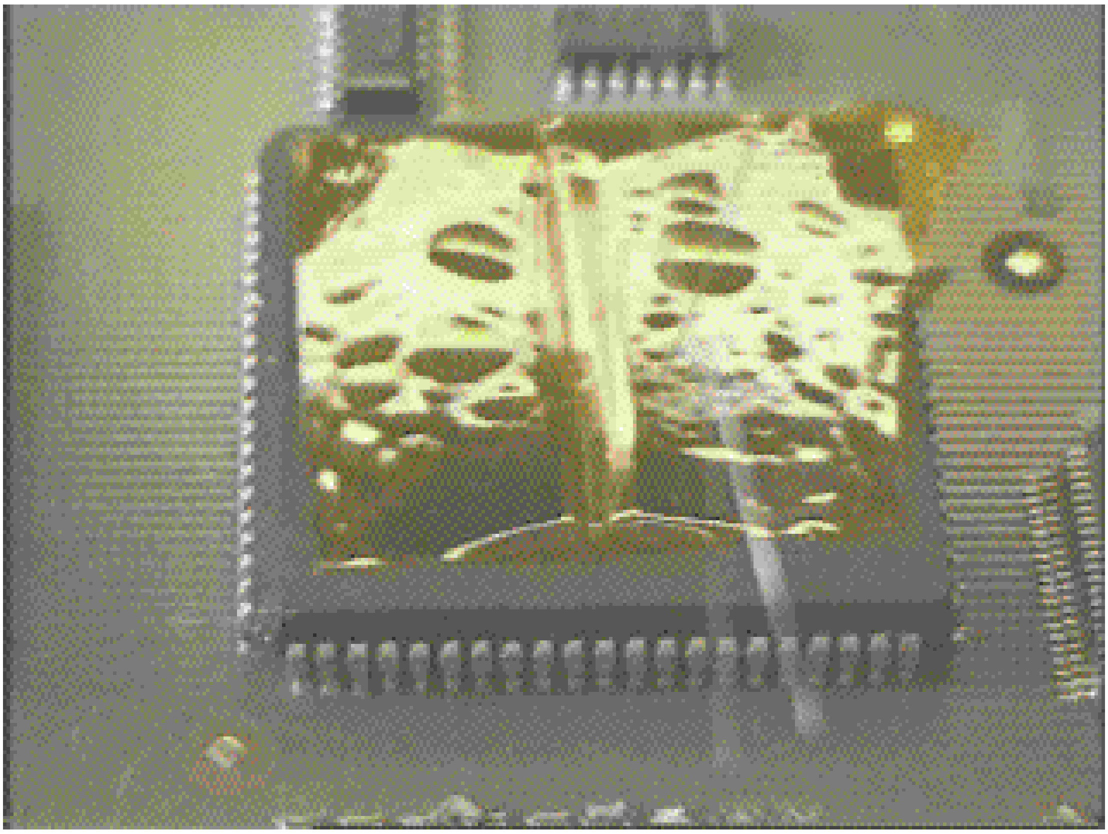
The following are examples of measured component weight compared with the pull force in a liquid state for the multiple solder joints.

Component	Component	Force Measurement Weight
QFP100	4.2g	31g
PLCC68	4.67g	31g
QFP64	3.8g	14g
PLCC44	2.2g	20g
PLCC28	0.68g	14g
SOIC16	0.142g	6g
BGA225	2.65g	46g

Measurements were produced by attaching wires to components and placing the boards on a hot plate. The force require to lift each part was measured when the solder joints where in a liquid state. The measurements are based on an average of three tests for each component type. Examples of this testing procedure are included on the SMART Group video on “Double Sided Reflow Assembly” to be released late 1998.

During double sided reflow solder of larger parts there may be some movement during reflow. When and if the solder does move into a liquid state the part will move away from the surface of the board. The actual movement is fairly minimal approximately 0.001-0.003" on a QFP. The weight of the part allows the solder to elongate. Phillips have suggested that this movement will not cause any reliability problems. There may be differences between soldering in air and nitrogen but no information has been documented and differences have not been seen by the author.

It has been suggested in the past by Multitone Electronics that this benefits cleaning as the final standoff height is increased.



The photograph above shows a PLCC being reflowed on a hot plate while the force required to lift it from the surface of the board is being measured. This is the simple technique to establish the difference between the weight of the components and the force of the solder surface tension. It would also be possible to perform wetting tests on different components to establish the surface tension forces. This could be conducted on systems supplied by either GEC-Marconi or Multicore Solders.

Printed Board Solder Finish

The following is a brief review of the solderable and protective finishes which are available and being used in the industry. The finishes are all currently used for printed boards which will contain both conventional and surface mount components. Although a mixed technology process exposes boards to heating during reflow and glue curing double sided reflow exposes the board to two high temperature cycles, care needs to be taken that the intended soldering process is compatible with the solder finish.

A more detailed introduction to evaluating solderable finishes and the introduction to a company is covered in the SMART Group/Shipley Europe reports or on the SMART Group video on Alternative Solder Finishes.

Tin/Lead Reflow

Tin/lead has been the standard finish in the industry for many years due to its use as an etch resist for the production of plated through hole boards when subtractive processes have been adopted. It has provided an ideal production solution to protecting the copper surfaces during the final copper etching process. It has also proved useful as it provides a solderable finish for the protection of the copper pads and tracking for subsequent soldering operations.

With the increase in modern manufacturing methods using wave soldering and reflow soldering, the finish has proved unacceptable due to the circuit's exposure to high temperatures during assembly which causes reflow of the tin/lead coating under any solder resist coatings. This has led to lifting and de-lamination of the resist if the tin/lead coating is too thick.

It has also been necessary for the tin/lead plate to be reflowed during PCB manufacture prior to solder resist coating. The reason for reflowing the tin/lead has been undertaken for two reasons:

- During the etching stage tin/lead slivers are left due to the undercutting which takes place during etching. If not removed the slivers are trapped under the solder resist coating and during the assembly process shorts between tracking.

- The second reason or benefit for reflowing the tin/lead plate has been the improvement in long term solderability of the circuit. The tin/lead plate has a short solderability life, but if reflowed the surface is no longer porous and provides a longer shelf life. A minimum of one year's shelf life should be obtained from a surface coating of five microns after it has been reflowed.

Unfortunately all tin/lead coatings which are reflowed or are applied to the circuit in a liquid form will tend to form a convex meniscus of solder on the circuitry. This is generally of no consequence to conventional assembly processes apart from affecting hole size, but has led to poor yields on screen print, glue dispense and component placement during surface mount assembly if the coating is inconsistent.

Brushed Tin/Lead Plate

The brushed tin/lead process has been offered over the last eight years by PCB manufacturers as a compromise for surface mount assembly as it provides a flat surface for component mounting and for screen printing. It probably became popular as it was a simple solution to the reflow process which resulted in uneven pad geometry for component placement. It prevented the distortion of the laminate due to exposure to the high temperatures associated with reflow. A further benefit was that printed board manufacturers avoided the investment in solder levelling equipment and could still offer a "Surface Mount Finish".

The process has the disadvantages that the tin/lead remains in a plated state and can become unsolderable within six months; it also still has the problems of tin/lead slivers. This is probably how the finish got its name of brushed tin/lead as all surfaces were mechanically brushed to remove any tin/lead slivers from the surface of the board prior to solder resist coating.

Solder Levelled

The solder levelling process became popular in the early 80's and is still the most commonly specified finish for surface mount boards. Results from the author’s last survey on solderable finishes trends is included in this report for reference. Eliminating the solder coating under the resist reduced the possibility of the resist lifting during the assembly soldering operation. It provided a guaranteed solderable surface from the PCB manufacture. It also provides a further benefit to the assembler of stressing the board. If the solder resist coating was poor or the lamination of a multilayer circuit was questionable then it would generally show up during exposure to the molten solder bath prior to shipment to the customer. Originally the coating was more expensive than traditional tin/lead plated finish but this is not now true.

The solder levelling process also eliminated significant mismatch between circuitry and the resist apertures. This was due to only limited temperature being applied to the laminate prior to resist application. With the tin/lead reflow process the laminate is exposed to soldering temperatures which exceed the laminate's glass transition temperature. This causes expansion and contraction due to the stress in the laminate which is no longer held by the copper foil.

Gold & Nickel

Gold is a traditional finish used in the industry due to excellent electrical finish, corrosion resistance and, when required, good solderability.

There has been some resistance to the use of gold, originally in Europe and still in the USA, due to concerns of reliability of the final solder fillet. In the past gold has been widely used for connectors; it was also used in the 1970's for a solderable coating on boards. In both cases the ill-informed use of thick gold > 1um coatings led to the formation of gold/tin intermettalics which in turn led to weak and fragile solder joints.

Ever since, soldering to gold has been avoided particularly in high reliability application like military and aerospace. Many existing standards relating to assembly and soldering require all gold coatings to be removed prior to the final soldering operations. It is a pity that standards are not re-examined every few years as many are just not relevant in today's technology.

Over the last six years gold over nickel have become popular finishes for surface mount boards. They have provided an ideal assembly surface, highly solderable and an aid to inspection due to the contrasting colour between component leads, solder and the solder paste. When wire bonding is required for chip on board applications gold over nickel has been the finish of choice when bonding and soldering is required. The cost is generally the same as solder levelled boards in medium to high volume.

Immersion Silver

This is a relatively new finish which was developed to provide a solderable and wire bondable coating providing all the benefits of traditional tin/lead coatings. Basically the coating is an immersion silver coating of between 0.08-0.1um which also incorporates an organic layer as part of the process. The silver "Alpha Level" coating is maintained in a highly solderable state by the organic coating.

The surface coating has all the benefits of any alternative finish and also resembles the tin coating when soldered. In the case of unsoldered holes or test pads there is no visible gold or copper, which to some engineers is an emotive subject. The coating cost in medium to high volume is equal to nickel/gold, but may become more cost effective as the material is further established in the market place.

Like any alternative coating, provided it is processed correctly by the circuit board manufacturer, the surface will remain solderable even after multiple heating cycles. Hence it is compatible with double sided reflow.

Flux Lacquer

The protection of the copper pads during storage and assembly prior to soldering are of prime importance However, the cost of the printed circuit is also an important issue particularly in consumer electronics. Surface mount technology is being used in all sectors of the electronics industry, inevitably it is being used in the consumer industry.

The use of flux lacquer as a protective coating which is applied to a copper pad is particularly widely used in the high volume TV/VCR industry dominated by the Japanese and Korean companies. Its use is generally confined to single sided boards.

The flux lacquer materials are supplied by a wide range of suppliers, particularly those companies who existingly supply soldering fluxes to the PCB industry.

The coating is generally applied by dip, spray or roller coating. Unfortunately all coating methods provide an inconsistent coating to the board surface, with spray coating method probably being preferred.

The coating provides a limited life expectancy due to the porosity of the coating and to its inconsistent coating thickness. The material is now being used by selected companies as part of a two part process. The lacquer is used as a secondary coating after the copper surface has been chemically treated and protected by a proprietary treatment.

Limitations of the coating have been its short shelf life, inconsistent coating thickness and incompatibility with Low Residue/No Clean fluxes. Significant residues are still left on the surface of the board after soldering. The coating is not really compatible with solder paste and reflow soldering so it would not be a coating of choice for double sided reflow.

A further problem has been seen when using the coating on boards which are to be flux soldered. In the case of flow soldering the components are held in place on the underside of the board with adhesive. In cases where the coating is thick the bond between the adhesive and the component is with the lacquer coating and not the printed board. During the fluxing and soldering operation the bond strength between the adhesive and lacquer can drop, causing components to be lost during contact with the solder bath.

It is common for the soldering process to be blamed for this loss of adhesion, but only limited force is applied to components during wave contact. For example, measurements of as little as 10-20 grams have been recorded acting on SOICs during contact with the wave.

The use of flux lacquers is undoubtedly a cheap option for providing a limited shelf life protection to the bare board. It does, however, suffer from the same problems as other protective coatings. Multiple high temperature exposures affects the solderability of the remaining pads thus causing soldering problems.

Protective Coatings

The protective coatings are generally defined as organic coatings referred to as OSP, (Organic Solderable Protector). The most common coatings are benzotriazole and imidazole; both are organic nitrogen compounds. Benzotriazole has long been recognised as an anti-tarnish coating used in the general metal finishing industry. Inhibitor coatings are extremely thin and essentially invisible on the copper surface.

The coatings protect the copper by chemically bonding to the surface and prevent the reaction between the copper and oxygen. The coating may be applied by dip or spray coating and followed by a rinse operation to remove any residues remaining on the solder mask surface. If required, the coating may be removed and re-applied to rejuvenate a surface which has become solderable. If required the surfaces would need to be re-cleaned with an acid etch and rinse prior to re-treatment.

The protective coatings have been used for many years by large volume manufacturers for surface mount products, an example of which is IBM. The limitations of the coating was its general inability to stand up to multiple soldering operations. The coatings are degraded by exposure to high temperature and become unsolderable with mildly activated soldering fluxes. The use of high activity water soluble fluxes have often been used on second side wave soldering processes requiring thorough cleaning.

The coatings have in the past also been susceptible to damage by high humidity storage which can degrade the solderability. Incorrect handling by assembly staff has also been seen to affect the coating due to the introduction of handling soils. A training video covering each of the different solderable finishes is available from the SMART Group to provide guidance on the correct use of these finishes.

The new generation of alternative copper protective finishes have been demonstrated to protect the surface during multiple reflow and high temperature storage. They have also been shown to withstand the handling issues during assembly and storage and are destined to provide the best vehicle for the future due to their competitive cost which is much less than any other finish.

Recent trials have indicated that cooling rates after reflow should be improved to reduce the effects on copper coatings. Cooling the board surface directly after reflow below 80^{o}C can prolong the solderability life of the OSP coating. The use of nitrogen during first side reflow with a oxygen level of 100ppm has also provided improved performance during second sided yields. Generally reflow engineers strive to reduce peak board temperature as it exits the reflow oven to reduce the chance of component misplacement, reduce intermetallic formation and, of course, the board needs to be cool for second pass printing in a high volume operation.

OSP is currently used by divisions of IBM, Siemens, Motorola, AT&T, Olivetti, Compaq and Dell. It is also a common process offered by many printed board suppliers. OSP coated boards were recently shown by Motorola to provide better joint reliability than gold or tin/lead.

New Technology Finishes

There have been a number of new processes introduced into the industry specifically designed for Fine Pitch SMT. The three new process finishes still use tin/lead coatings but it is their method of application which has caused them to be considered new techniques.

SIPAD - developed by Siemens, SIPAD is used on the finished board after solder mask coating. The solder mask coating must be relatively thick as it is required to define the position and height of the solder coating. The process requires solder paste to be stencil printed into the solder mask openings and onto the copper pads. As cleaning is easy on an unassembled board activated pastes may be used without difficulty. After the solder paste is reflowed the domed molten meniscus is then flattened using a hot metal plate. As the metal plate cools to below the melting point of the solder a flat surface is achieved with the solder filling the solder mask opening.

OPTIPAD - in the OPTIPAD process a temporary solder mask is used to create a mould around the land areas on the circuit board with the molten solder forced under pressure into the apertures. A membrane then seals the board surface during the solder solidification. After removal from the process equipment the temporary solder mask is stripped. The result is a well defined solder deposit of a controlled height and surface flatness.

OPTIPAD is a registered trademark of SMW Elecktronik, Germany

PPT (Precision Pad Technology) - the PPT process is a variation on the previous two processes. As in the SIPAD process, solder paste is used to provide the tin/lead coating. A dry film resist is used to create a mould around the individual pads on the circuit board. The solder paste is applied by stencil printing and reflowed in a standard reflow oven.

In the case of PPT a fixture is used to help control the position of the solder paste and to create a dimple effect on the surface of the pad surface. By using different fixtures the surface topography of the pad may be changed to aid coverage and adhesion of the sticky flux which is required for component positioning and to aid soldering.

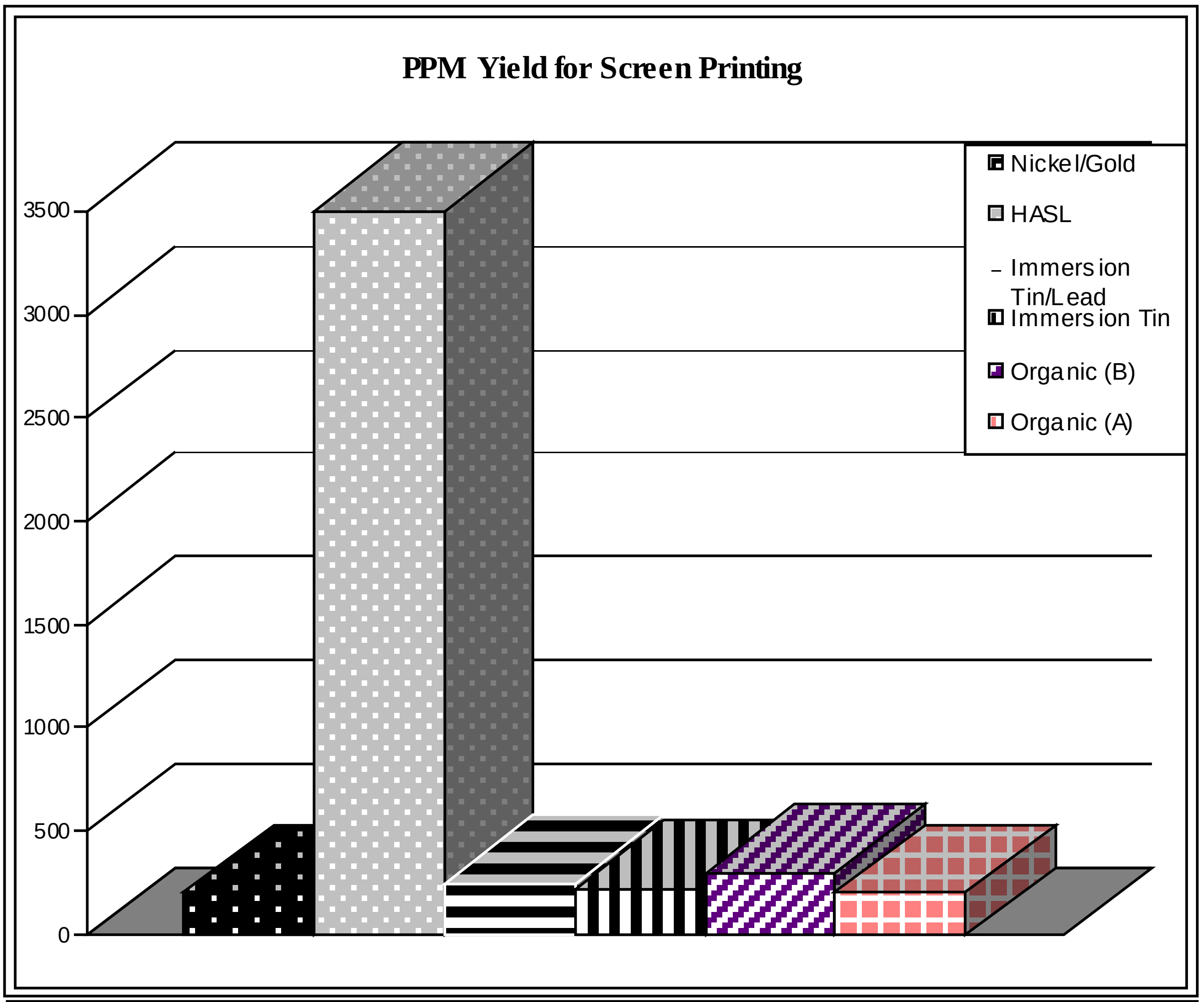
PPT (Precision Pad Technology) is a registered trademark of Mask Technology

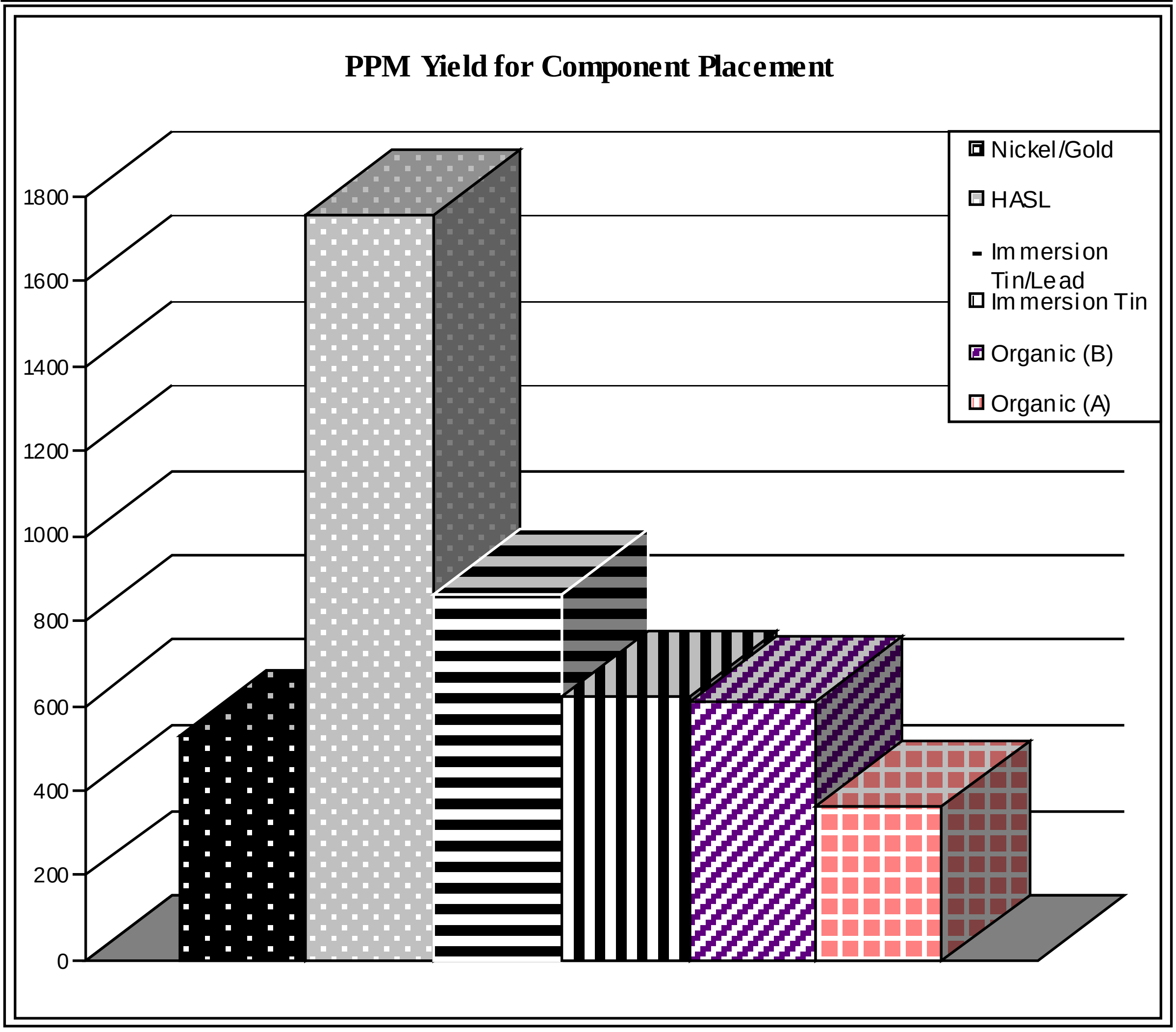
The four graphs that follow are taken from the SMART Group/Shipley Europe Solder Finishes Report produced by the author in 1997 which show the effect on assembly yield when using alternative solderable finishes.

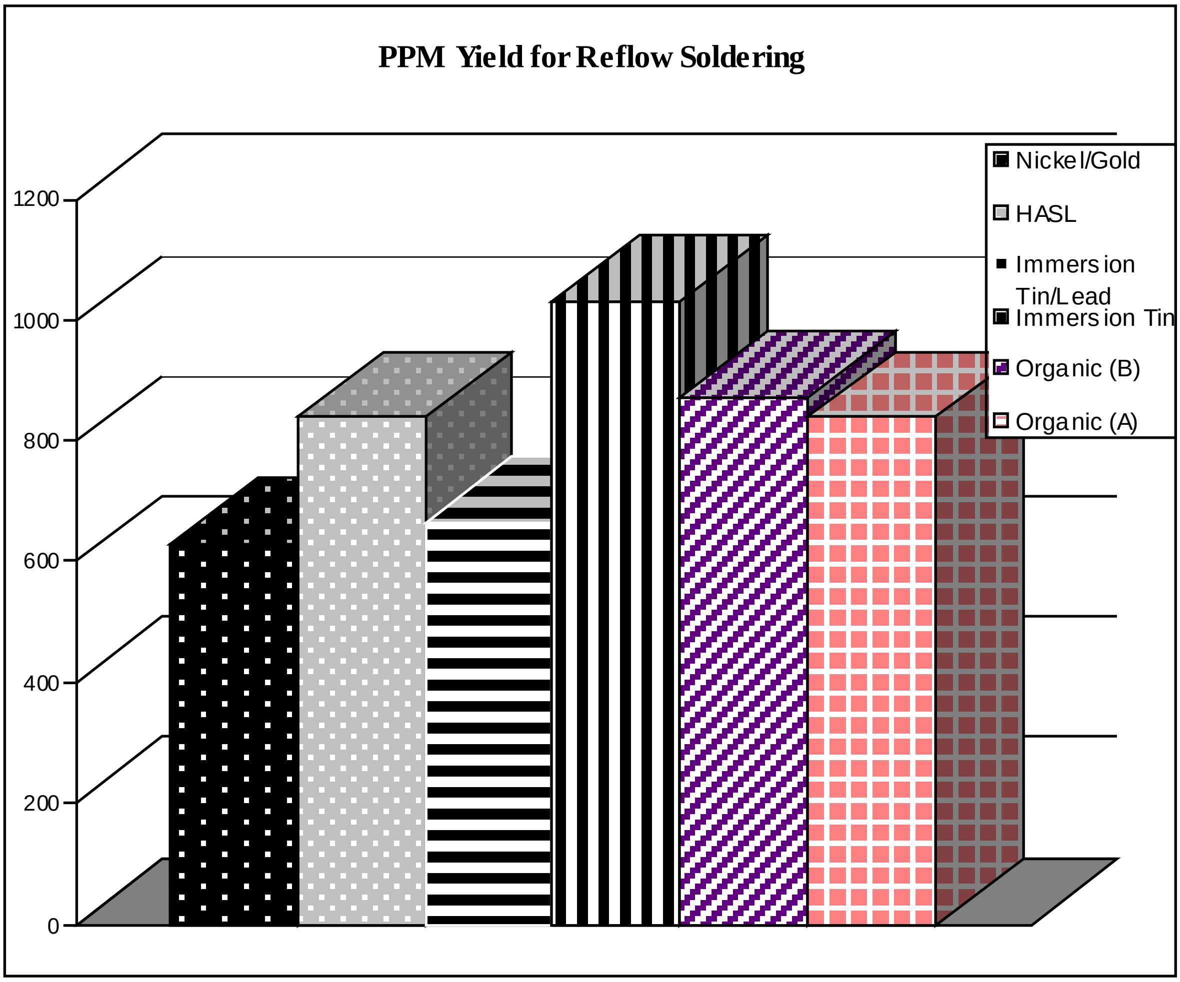
The first graph shows the yield from screen printing which shows the highest defect rate on solder levelled boards. The most common defect was halo prints where the solder paste was only around the outside of the pad due to the solder coating coming into the stencil aperture.

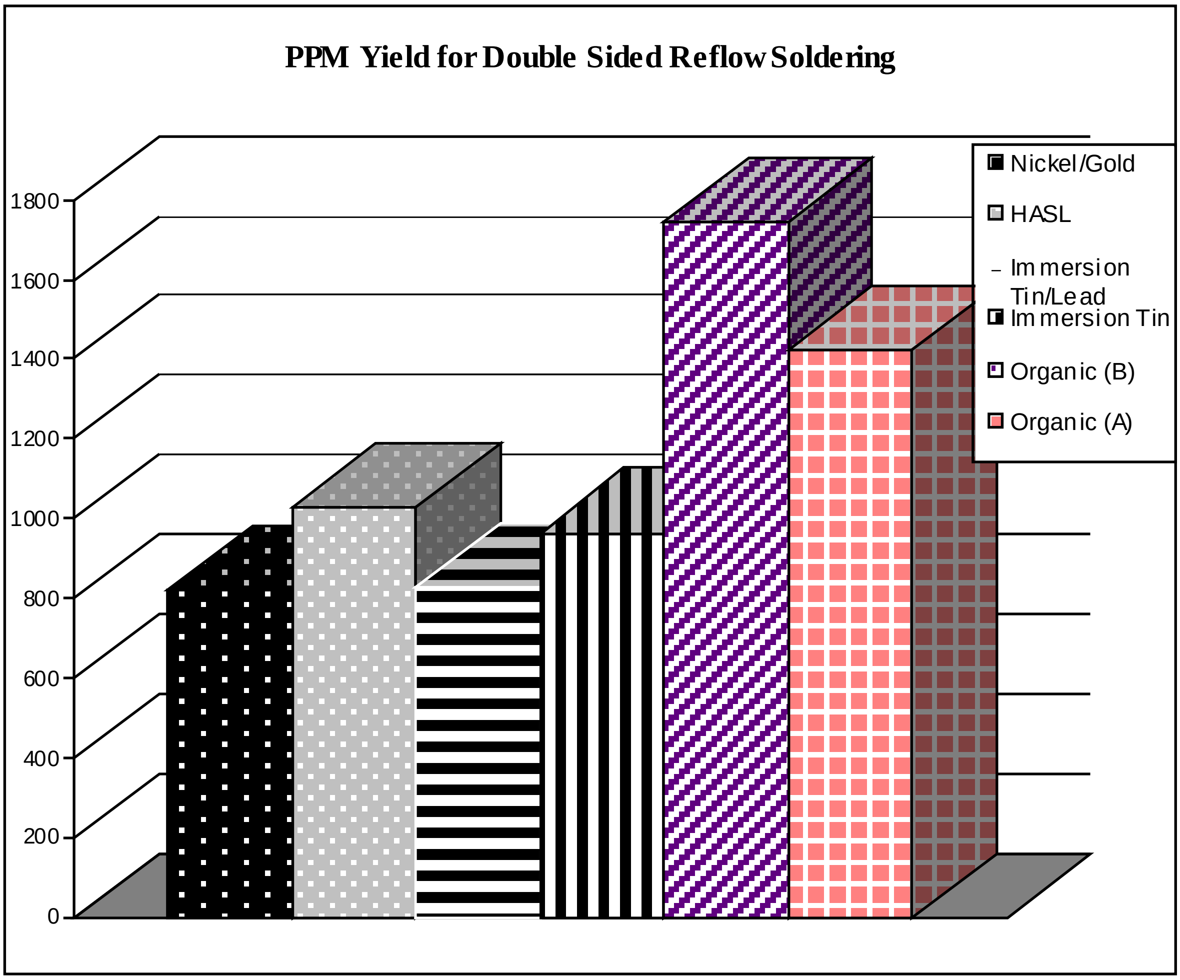
The second graph shows the defect levels at placement which are partly due to poor placement and component slippage. The tin/lead surface also affects adhesive dispense which in turn prevents satisfactory green strength forces on the component.

The third and fourth graphs show defect levels after first and second side reflow. Most of the differences in the results is due to the changes in solderability due to the first reflow process. Modern coatings, correct temperature control and the possible use of nitrogen during reflow eliminates the problems associated with the old copper (OSP) coatings.


PPM Yield for Screen Printing
3500
3000
2500
2000
1500
1000
500
0
Nickel/Gold
HASL
Immersion Tin/Lead
Immersion Tin
Organic (B)
Organic (A)


PPM Yield for Component Placement
1800
1600
1400
1200
1000
800
600
400
200
0
Nickel/Gold
HASL
Immersion Tin/Lead
Immersion Tin
Organic (B)
Organic (A)


PPM Yield for Reflow Soldering
1200
1000
800
600
400
200
0
Nickel/Gold
HASL
Immersion Tin/Lead
Immersion Tin
Organic (B)
Organic (A)


PPM Yield for Double Sided Reflow Soldering
1800
1600
1400
1200
1000
800
600
400
200
0
Nickel/Gold
HASL
Immersion Tin/Lead
Immersion Tin
Organic (B)
Organic (A)

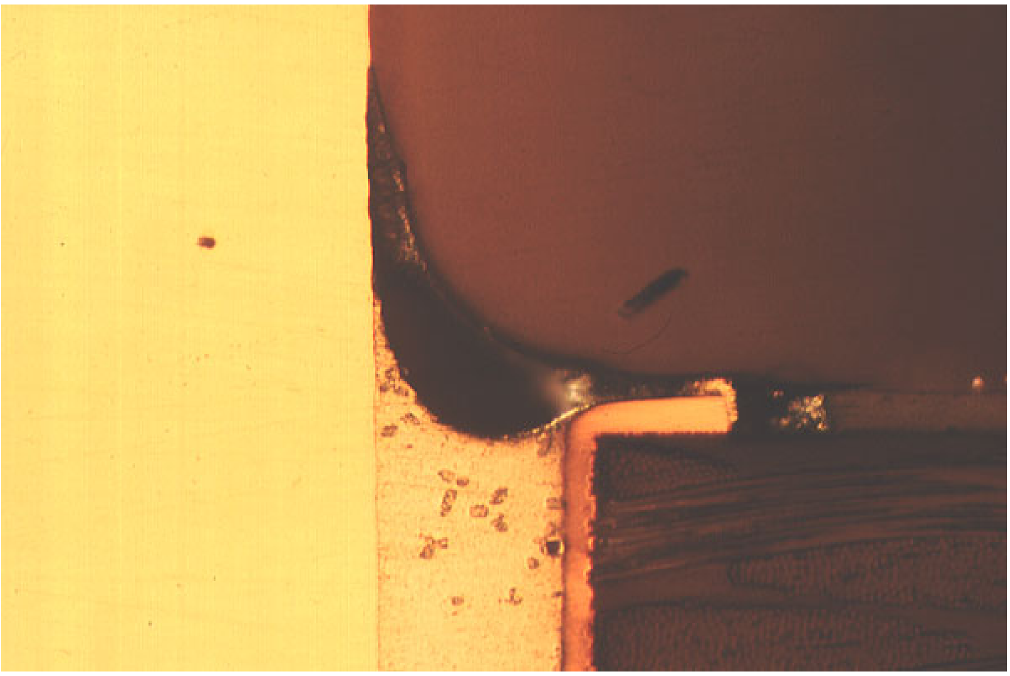
Conventional Through Hole Reflow

A complete report and process guide on Pin In Hole Reflow is available from the SMART Group office and should be obtained for a fuller reference. A video tape on the use of the technology is also available from the SMART Group and SMTA. A short section on the technology is included here. PIHR is the final stage in eliminating all other soldering processes and, with double sided assembly, provides the true surface mount process assembly. This is despite the fact that leads still go through the board.

One problem that has always proved a problem to design and process engineers is existing through hole parts where no direct equivalent SMT part is available.

It is possible to hand solder conventional through hole components after surface mount assembly and soldering operations are complete. This, however, is time consuming and may leave more flux residues on the surface of the joints causing a problem to in circuit test. Many through hole parts are used for direct contact during test and residues can quickly clog test pins in a no clean process.

A range of selective soldering equipment is available to either semi or fully automatically solder through hole leads. This does, of course, require capital expenditure on equipment. One method of soldering all surface mount and through components in a single operation is Pin In Hole Reflow (PIHR) or Intrusive Reflow soldering.



The microsection above of a Pin In Hole joint shows just less than 100% solder fill after reflow soldering. The pullout strength of joints like this example are no different than hand or wave solder joint at 20-25lbs.

There are three alternatives to adding solder paste for through hole components prior to inserting parts:

* Stencil Printing
* Double Stencil Print
* Paste Dispensing

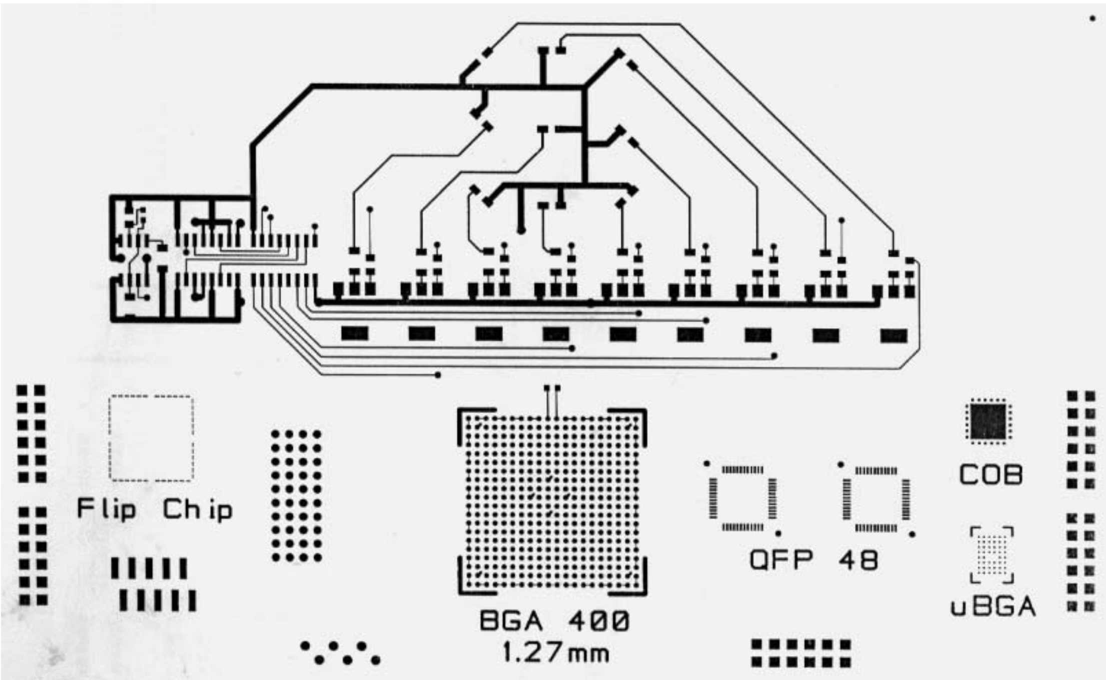
Stencil Printing

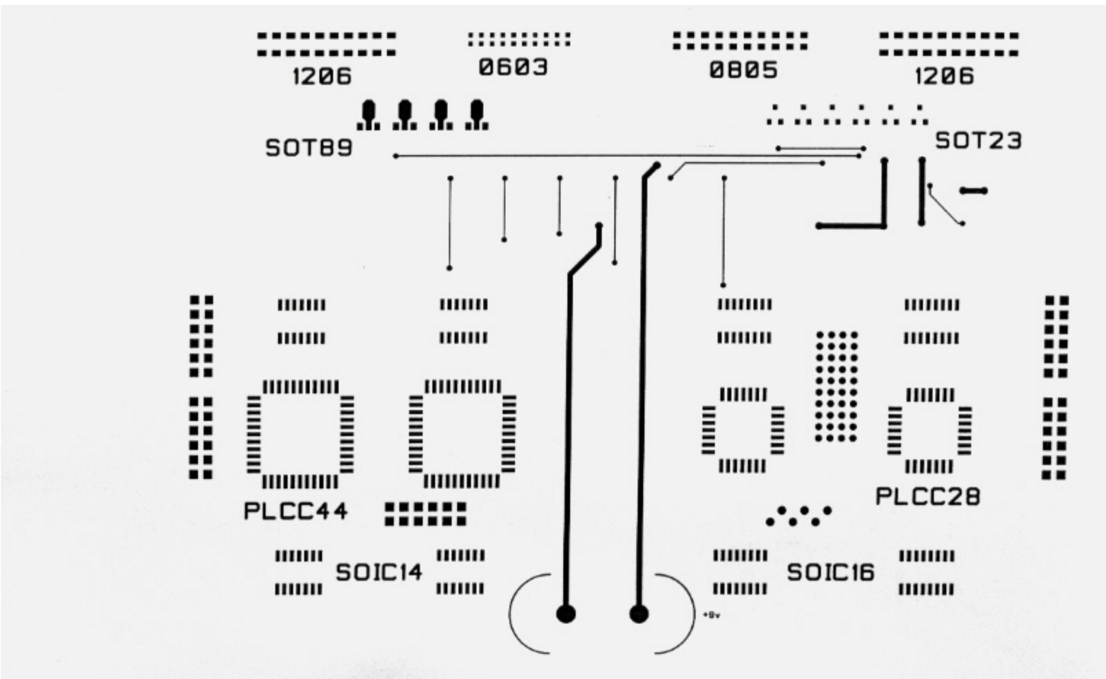
Solder paste is printed into through holes and over the pad surface during normal surface mount paste printing. The size of the aperture in the stencil is adjusted to allow paste to fill the hole, cover the pad and fill the resist aperture on the board. Component insertion is then conducted either manually or automatically by robot prior to the whole assembly passing through the reflow soldering process. Rather than allowing one squeegee stroke a double stroke can be used if the stencil is perfectly in contact with all the pad surfaces. If the stencil is not sealing the pad surface as it should the solder paste print will bridge particularly on fine pitch locations.

Board support during printing is crucial especially with thin boards. Care must be taken with double sided boards to position the support pins or custom jigs so that no damage can occur on bottom side components and that the board remains flat. The same is true for placement systems.

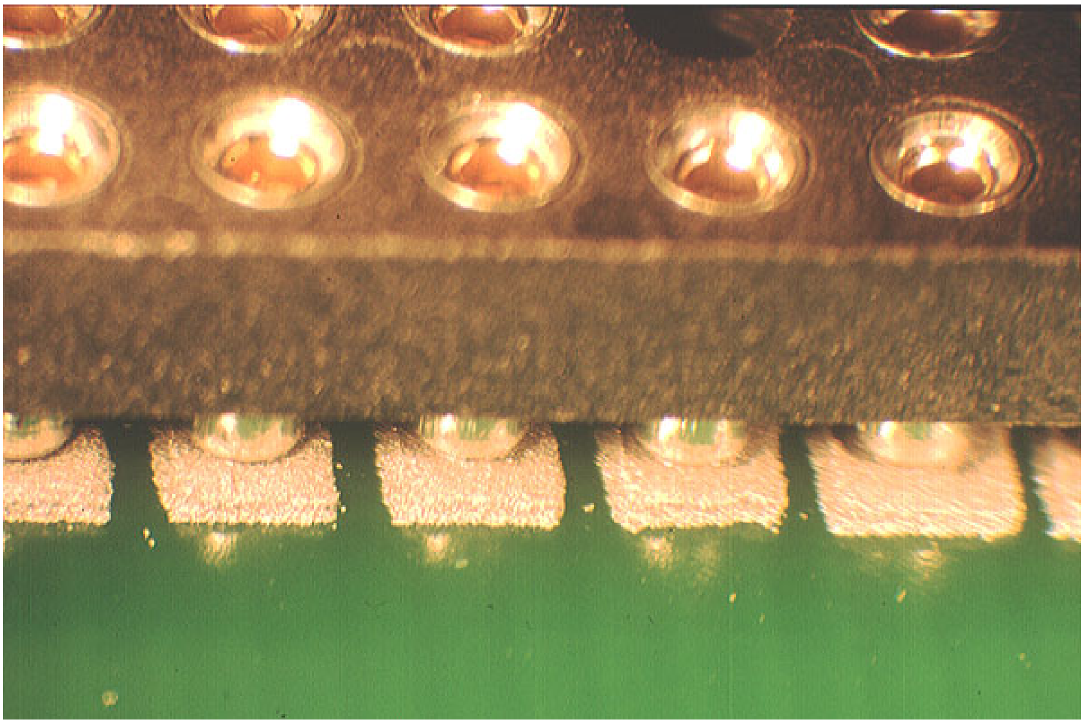
Double Stencil Print

As it may be difficult to obtain the volume of solder for a through hole components if lead to hole ratio is excessive, a double print operation is possible. In this case an initial printing operation is conducted to force paste into the hole only. A second printing operation is then conducted to add additional paste to the hole, pad and resist window. This increases the solder volume in the joint. This operation does require two printers and two stencils unless the production operation is in batch production when a set of boards may be printed and the stencil changed for the second print.





The double sided test board above was produced for Speedline Technologies and Siemens and was used during the double sided profiling exercises. The board features surface mount components on each side of the board. The top side was also used to demonstrate double sided reflow by simply turning the board up side down and repeating the reflow process.



This example of a Pin Grid Array socket is shown inserted into a through hole after paste printing. This process is referred to as Intrusive or Pin In Hole Reflow Assembly and is becoming very popular when conducted with double sided assembly.

Paste Dispensing

An alternative to printing is dispensing the paste into the through hole which can overcome all the problems of metal volume. Today, due to the increased speed of dispensing, the throughput would not be a problem for an in line application.

In each case the goal to through hole reflow is to achieve the required solder fill and guarantee the reliability of conventional components. The desired solder joint is a fillet visible on each side of the board after reflow. This would be the same criteria applied to wave or hand soldering operations to either IPC, IEC or British Telecom standards. However, it may not be possible to obtain a positive fillet above the surface of the board. Often the solder fillet remains flush with both sides of the circuit board but it is as strong and reliable as a full connection.

The following process parameters affect the solder joint formation, each needs to be known to calculate the volume of solder required for each joint. In some cases design modifications can be made to improve manufacture and the final joint quality.

* Diameter of component contact
* Diameter of plated through hole
* Thickness of circuit board
* Diameter of pads

Component compatibility needs to be confirmed with the suppliers. Today many connectors, sockets and some active parts may be assembled by reflow. The main problem that may occur is distortion of the parts causing them not to sit flat on the board surface. Stress cracking may also be seen on the corner of parts due to stresses building up during their manufacture.

Care needs to be taken over the reflow process to eliminate voiding in the joints. As so much paste is confined in a hole rather than on the surface of the board the evaporation of volatiles from the paste can cause a problem. This is particularly true when connectors are mounted on the topside of the board during reflow. In this case volatile gasses find it difficult to escape from the through hole. The profiles of most reflow systems may need to be re-examined as most of the through hole parts like connectors, pin grid arrays and sockets can have a marked effect on local board temperatures.

Component Compatibility

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.

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.

Clearance under the component should be available at the base with some form of standoff pip or foot. A minimum of 0.015" prevents the part contacting the paste. If standoff 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. Standoff 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 standoff pins on the base of the parts can be in different positions!

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.

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 ration 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.

Component Placement

There is very little to complicate the placement process when DSRS is introduced. The main difference is the components on the base of the board. The height of the parts can be an issue along with their position. Special mounting support is required to support the board or panel to stop flexure.

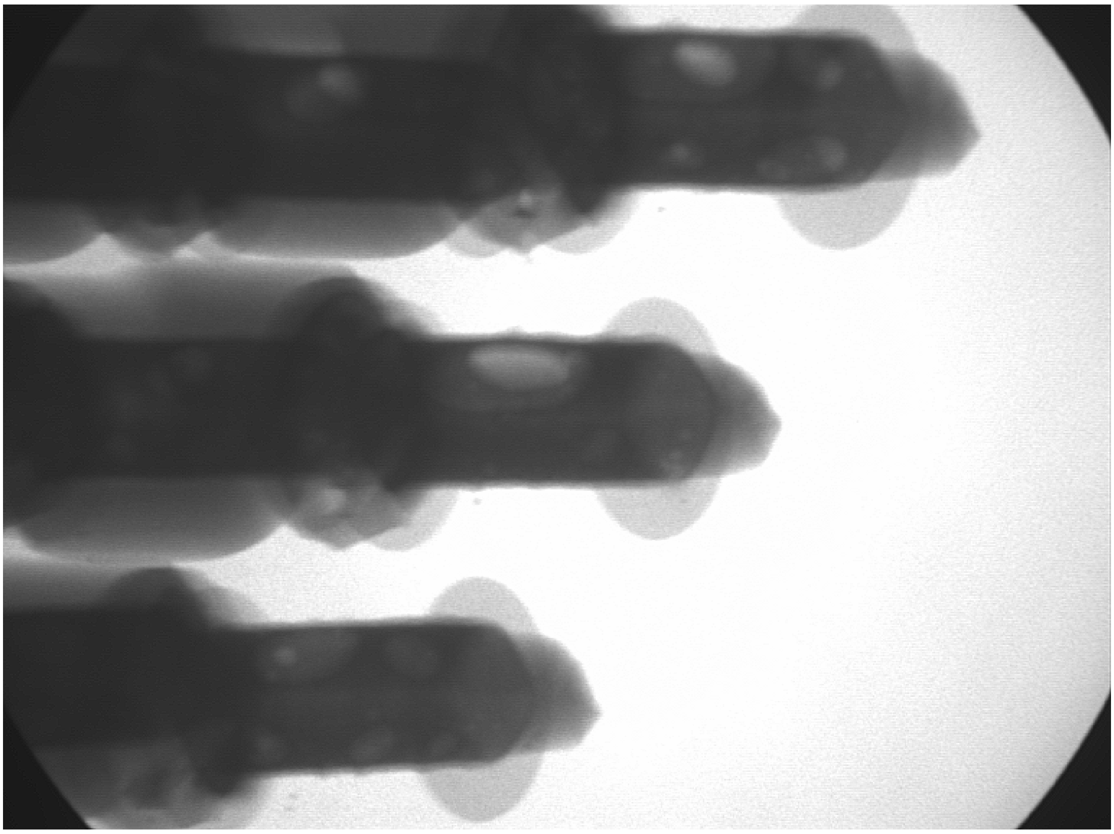
Support is generally provided by a series of pins which are positioned on a steel table to contact the board in specific positions. The pins have magnetic bases which allow fast positioning. It is important to select the correct positioning ideally with clearance around the pins. If clearance is not considered components may be damaged. The pins must be set at a height to support the board during placement not with a +/- position to the base of the board. This would cause flexure of the board and again may cause damage to the joints or components already assembled.

Ideally the loading of the support pins should be as even as possible to distribute the force on the board, a similar principle to in circuit pin contact and of course they must all be of the same height. An easy method of positioning of pins and re positioning during setup is to use the reference board used for placement inspection. This board is normally dedicated as a reference for placement and comparison of known component parts. This circuit may be used for support pin checks. Holes are drilled in the board in the correct position for support pins. It provides a simple quality control check and a guide for machine setter adjustment.

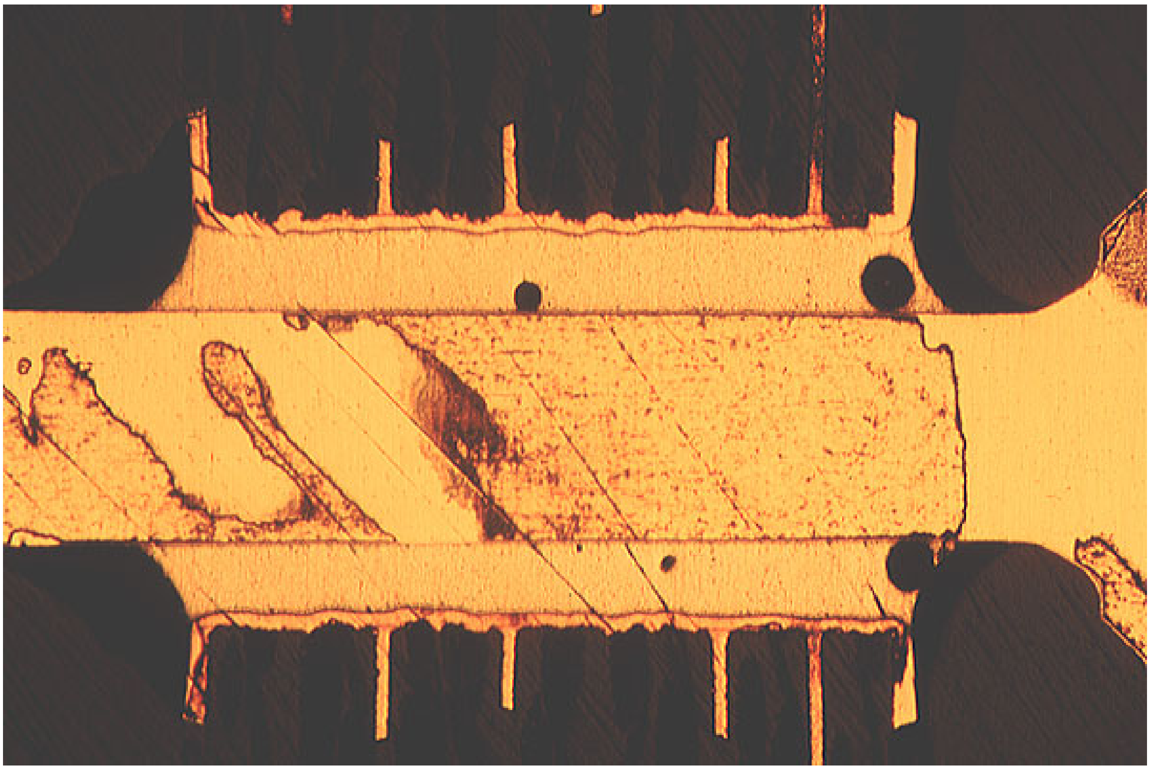
Board support pins have been successfully used for a number of years on placement systems and screen printers. As the industry moves away from the traditional 1.6mm laminate to 1mm and below for PC Card technology pins may not be ideal. Simply the space for support pin positioning may not be available or the pins can damage the board. Now we see the introduction of vacuum plates specially moulded to support the board and components in a nest.

Reflow Soldering

Both convection and vapour phase soldering systems can be employed for double sided reflow assembly, with convection being the dominant force in reflow today. There are two approaches being taken during reflow. One is to accept that the solder joints will reflow again and not compromise other factors. The second approach is to reduce the input into the base of the board in some way or specifically try to reduce the temperature on the base of the board.



The X-ray image shows a series of pin in hole solder joints with voids present after reflow soldering. Comparing the joint strength with joints which were found to be completely solid made little difference to the final joint strength. The degree of voiding shown is not typical of a well defined process.



Microsection taken from a pin in hole joint taken from a Pin Grid Array socket. The joint is shown with 100% solder fill, positive fillets and some minor voids which have no effect on joint strength.

It is feasible to cool the underside of the board in non convection based reflow units but in the case of the convection systems the forced cooling must adversely affect the oven's ease of setup or operation. The difference in temperature on the top and the bottom of the boards will also be affected by the throughput speeds.

Profiling Double Sided Boards

Standard techniques are used for profiling printed boards during the reflow process. Care needs to be taken to understand the temperature difference between top and bottom side of the assembly which can be easily over 20°C. It is possible to obtain a different temperature on one side of the board to the other when using top and bottom heating. Ultimately heat is absorbed by the board and is transmitted through it by conduction which allows component joints on the base of the board to be in a liquid state. The rate of heat transfer will be affected by the inner core on a multi-layer board, the through holes connected to it and the via locations which may also have a heat sinking effect. Large components on one side of the board will also affect the surface temperature on the opposite side.

During experiments of temperature measurement on a double sided product a profile should first be established for the assembly with only topside components fitted. The profile should also be checked for the same board with the board turned upside down. This may be conducted without the second side components fitted if only topside heaters are used or if the bottom side heaters are set at a lower temperature for normally processing. A clear understanding may then be obtained on the effect of the additional components and the oven's capability.

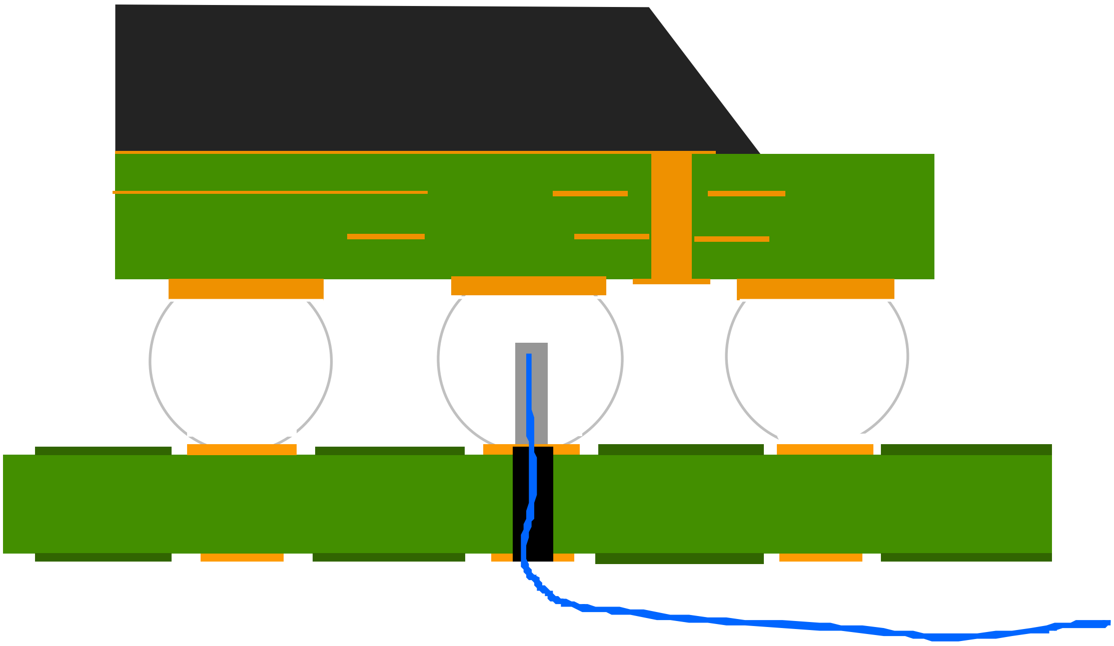
Profiles and Thermocouples

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.

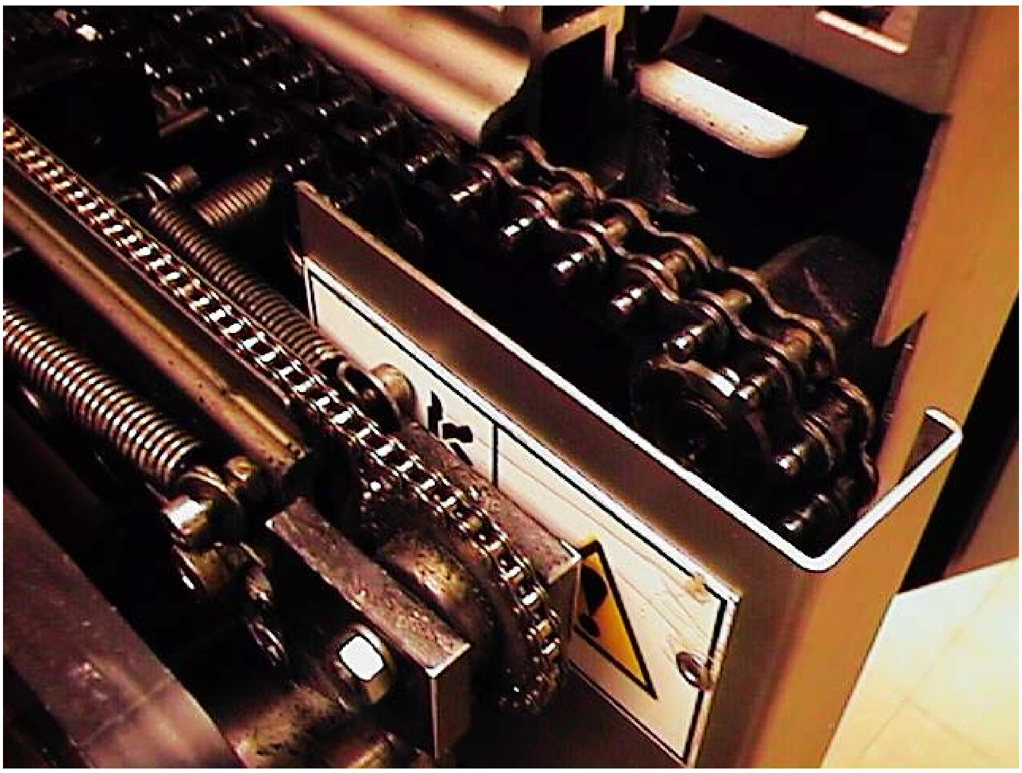
Next the first board with thermocouple leads attached may be passed through the oven and the temperature profile analysed. Adjustment may then be made to the zone temperatures and conveyor speed to obtain the correct profile. The desired profile is a combination of recommendations from the solder paste 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 visible 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 a number 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.

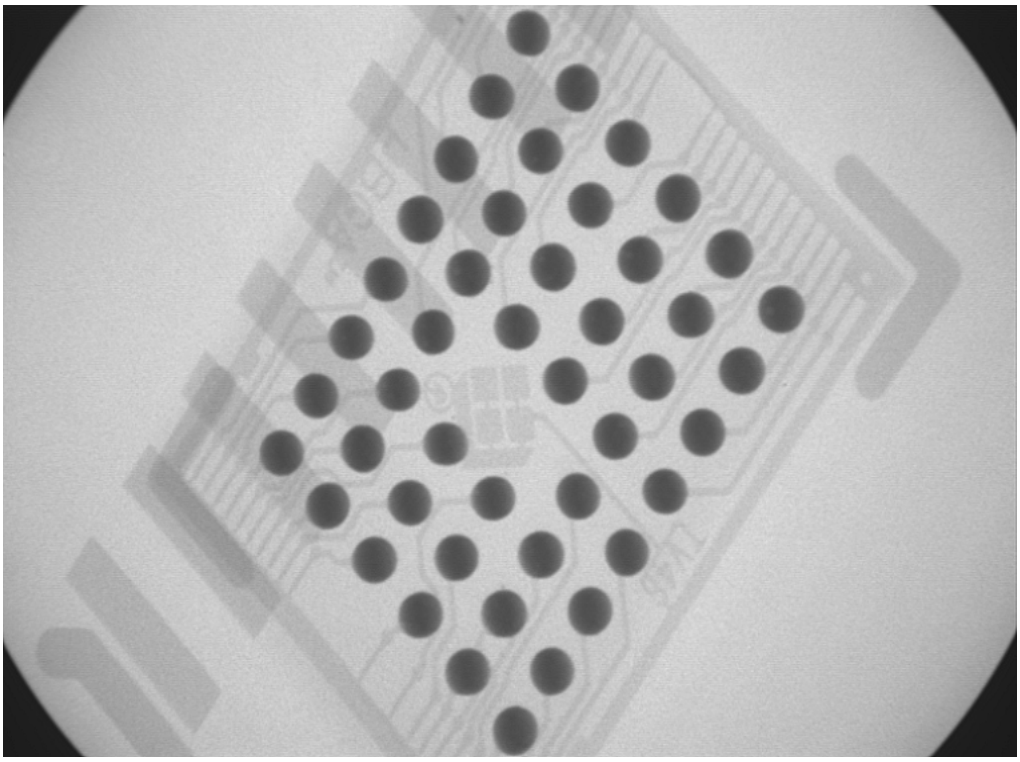
With Ball Grid Array (BGA) the lead must be positioned under the centre of the device. In most cases these are the last terminations to reflow during soldering. Either thin wire is used, or more commonly, a profile board is produced with a thermocouple wire mounted through the board into a ball termination to improve the repeatability of the temperature measurement. An example of which is shown in the following diagram:

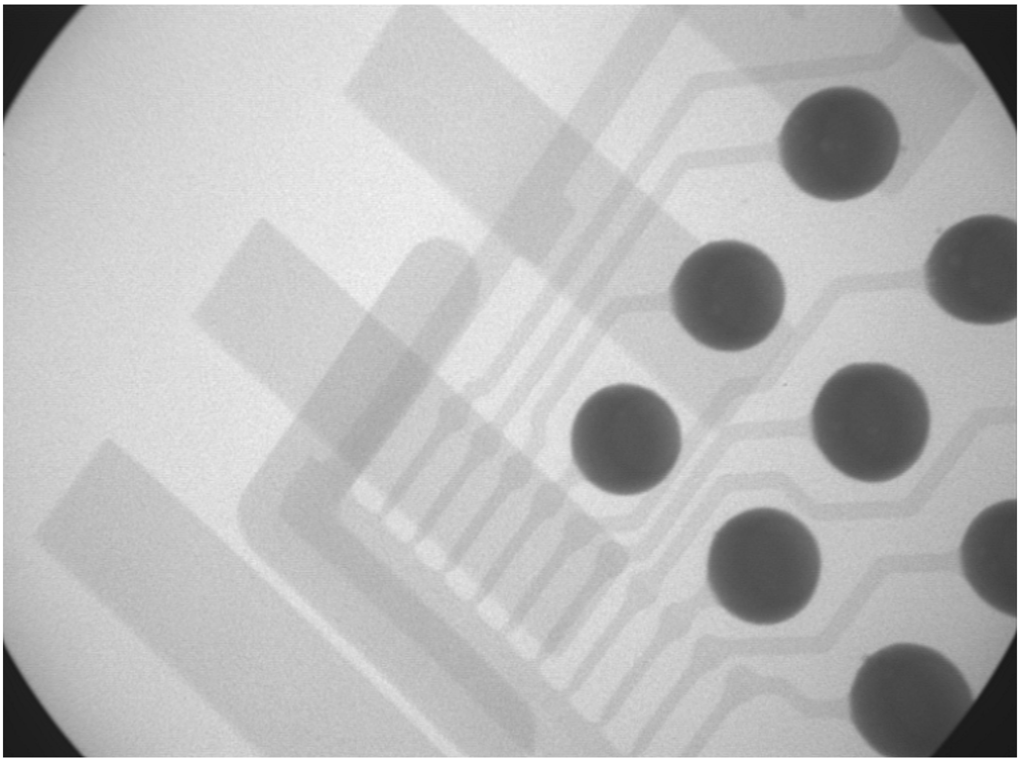


Example of the thermocouple positioned through the printed board and into the ball termination after reflow.



Example of reflow conveyor system with a centre board support chain to reduce the possibility of boards sagging during reflow. Newer systems require even less area on the base of the board for support.





Two x-rays of a uBGA after double sided reflow soldering. The devices were exposed to two reflow cycles in a liquid state. Due to the limited component weight there was no evidence of joint elongation after the initial reflow cycle.

All profiles should be developed on a fully populated board to guarantee that the correct conditions are achieved. If the boards are to 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. A procedure for reflow oven evaluation trials is available from the author’s Internet webpage: http://www.bobwillis.co.uk

Final Trials

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

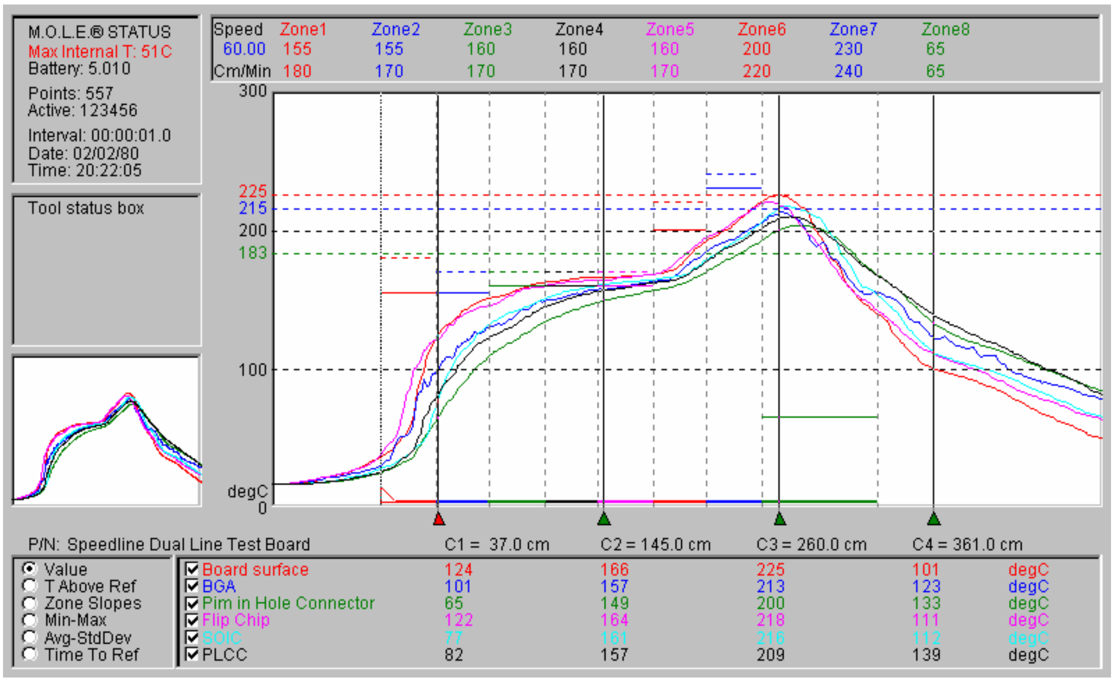
- the solder temperature in each zone
- the speed of the conveyor
- the extraction rates
- the board loading

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 the 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.

Below is a typical thermal profile for a double sided board with six thermocouple probes mounted on the board using high temperature solder. The probes were mounted in the following areas on the board:

1. On the surface of the board
2. Under the centre of the BGA
3. Under the edge of the flip chip
4. Under the base of the through hole connector
5. On a PLCC on the base of the board
6. On a SOIC on the base of the board



The temperature profile was recorded on a ECD Mole profiler with the leads connected to a double sided test board. The design of the board is shown at the start of the report.

A simple joint test on a trial board after reflow may consist of pulling individual leads off the board. As a guide a pull force on a good 0.020" gull wing joint will be between 750-900 grams. After removal the solder fillet should not exhibit any voiding. A small amount of voiding is acceptable as it has very little effect on the reliability of the joints. There are reports that indicate that voids increase the reliability as they stop crack propagation in certain plains. It is, however, very difficult to produce voids to order.

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 convection profile, be a Great Engineer and learn more about your process.

PRACTICAL DSRS PROCESS ISSUES

Pin Support or Linked Conveyor

The preferred conveyor for double sided reflow is a pin chain conveyor as this prevents contact with components on side one. During passage through the reflow oven the components hang free without physical contact. Using a linked mesh system allows the components on the base of the board to contact the mesh. If the solder fillets do reflow, components can be misplaced or lost. Using a standoff jig does make it possible to run boards on a mesh.

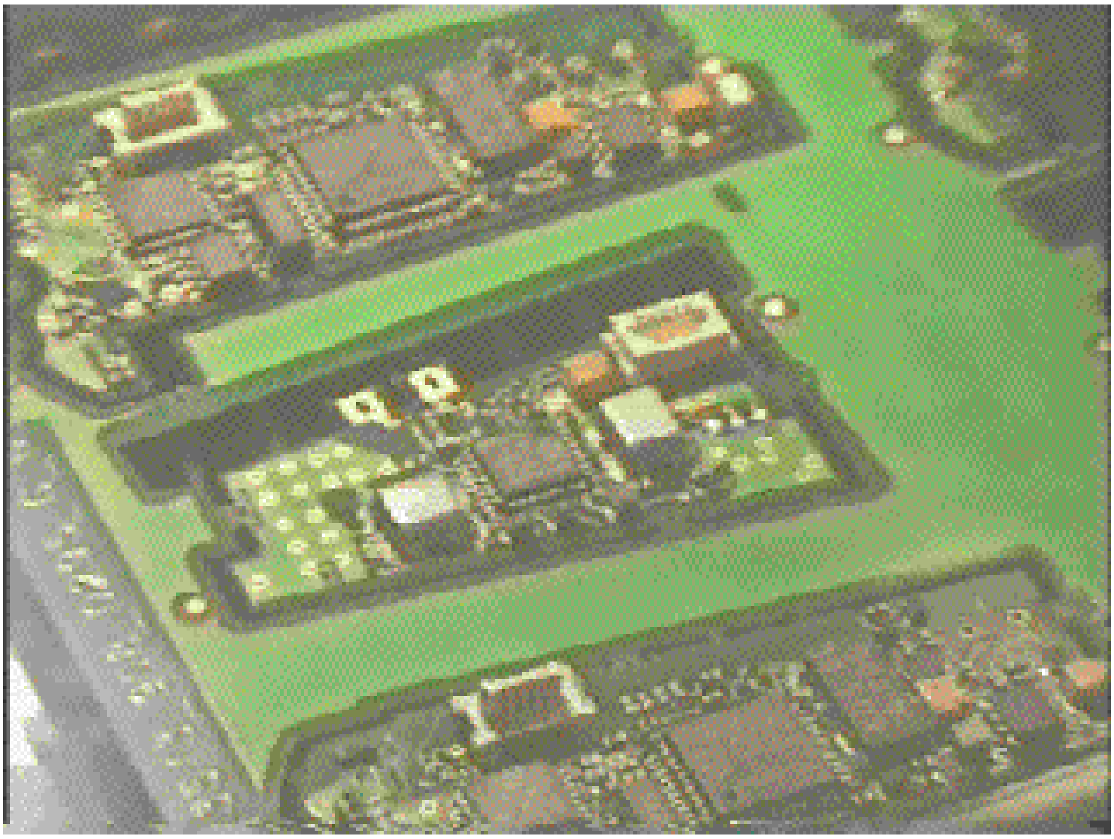
Many engineers do run double sided boards on mesh conveyors and say that there are no problems. Just ask your rework department how many components are reworked due to misplacement and loss?

Reflow Conveyor Vibration

As previously highlighted, when solder is in a liquid state there is a degree of strength in the liquid solder. This will prevent components moving on the base of the board during reflow soldering. If vibration is present in the conveyor system this could cause a resonance between the board and component and cause it to drop. Vibration in the conveyor is not desirable for reflow soldering but must be eliminated for DSRS.

Vibration can be caused by poor maintenance of the system’s conveyor, vibration of a convection oven’s fans or by fluctuation in the air flow patterns. The higher the air flow in a convection oven the more it is possible for movement of the board to occur. This is becoming more of an issue as we move away from traditional 1.6mm thick circuits to below 1.0mm.

A simple test to examine for component movement due to vibration has been used by Intel. A flat laminate sheet is placed on the conveyor and passed through the machine using the required process settings. The laminate has been loaded with surface mount components and an outline marked on the sheet for each component. There is no attempt to hold the parts in place with paste or flux while the samples pass through the oven under test. Provided the test panel exits the oven with no component movement the product has passed the test. Originally this test was used for flip chip assessment but it could be applied to may board designs.



Care needs to be taken on the amount of board flexure seen during the reflow process. Flexure due to vibration, or simply the board sagging under its own weight, can displace components on the base of the board. The components are simply pushed from the surface of the board if the solder joints are in a liquid state.

Solderability of Boards and Components

As was highlighted in the section on solderable finishes for printed boards, care needs to be taken on the effects on solderability when passing the board through two reflow operations. The first reflow operation may cause problems for the second pass.

Personal experience would indicate that it is often the original solderability of the board rather than the multiple heating operations.

Component Weight

Weight is the first issue normally considered by engineers evaluating DSRS. If you are simply second pass soldering a board with chip components on the base of the board it is not an issue. Chip components all weigh less than 2g.

In the case of QFP and PLCC devices they weigh considerably more. If the solder on the base of the board goes into a liquid phase a PLCC is more likely to fall from a board than a chip component. The results from the report indicate that this is not the case and many large parts can be processed on the base of the board which will be great news for design engineers. It also shows that the calculations for component compatibility are conservative!

Many of the cases of lost or rotated components are crystals, oscillators and RF parts which have ceramic and metal cases and probably not considered when the original calculations were made for the formulae used in industry.

Board Flexure

We all know that when a circuit board passes over a wave soldering system the board may sag. When a board passes through the reflow soldering process a board is also likely to sag or flex if not supported. The degree to the amount of flexure depends on the board material, reflow temperature, width of the board and the degree of support.

When a component is in contact with the board both surfaces are parallel. During reflow the board may sag or flex but the component will remain rigid due to the materials used in its construction. There is a tendency for large components to be forced away from the board surface causing them to drop. In this case board flexure is the root cause of the problem.

Secondary Reflow

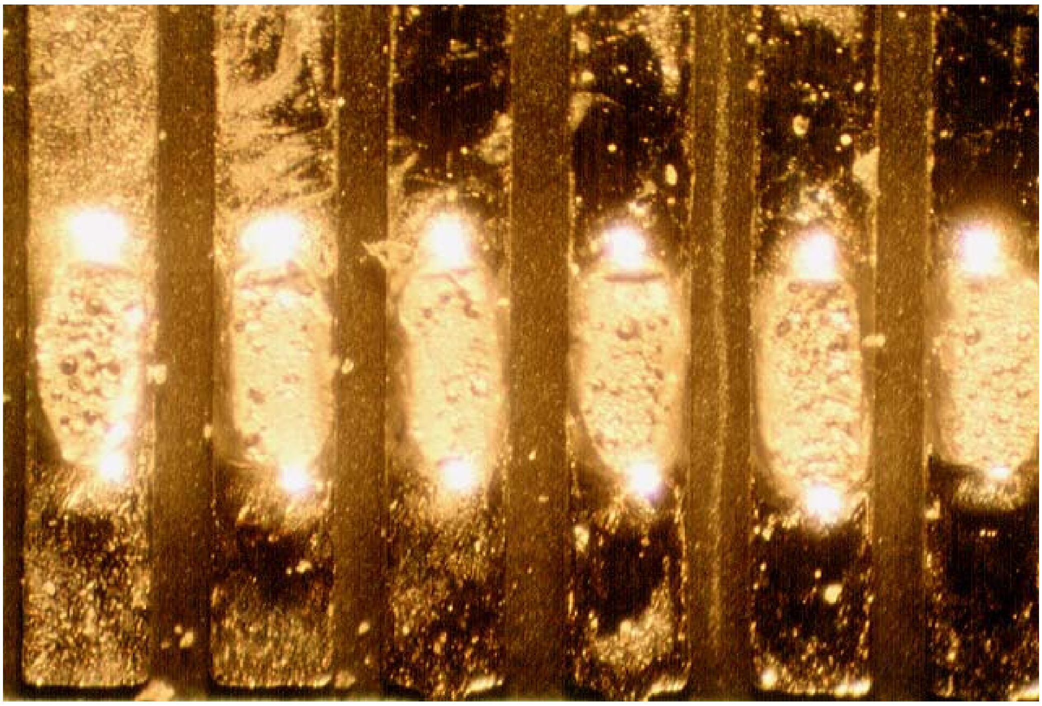
Secondary reflow is where the solder joint on the base of the board becomes liquid during second pass soldering. Most engineers try to avoid this occurring because of the concerns of lost components. The results of trials conducted as part of this report indicate that this is less of an issue than engineers have thought in the past.

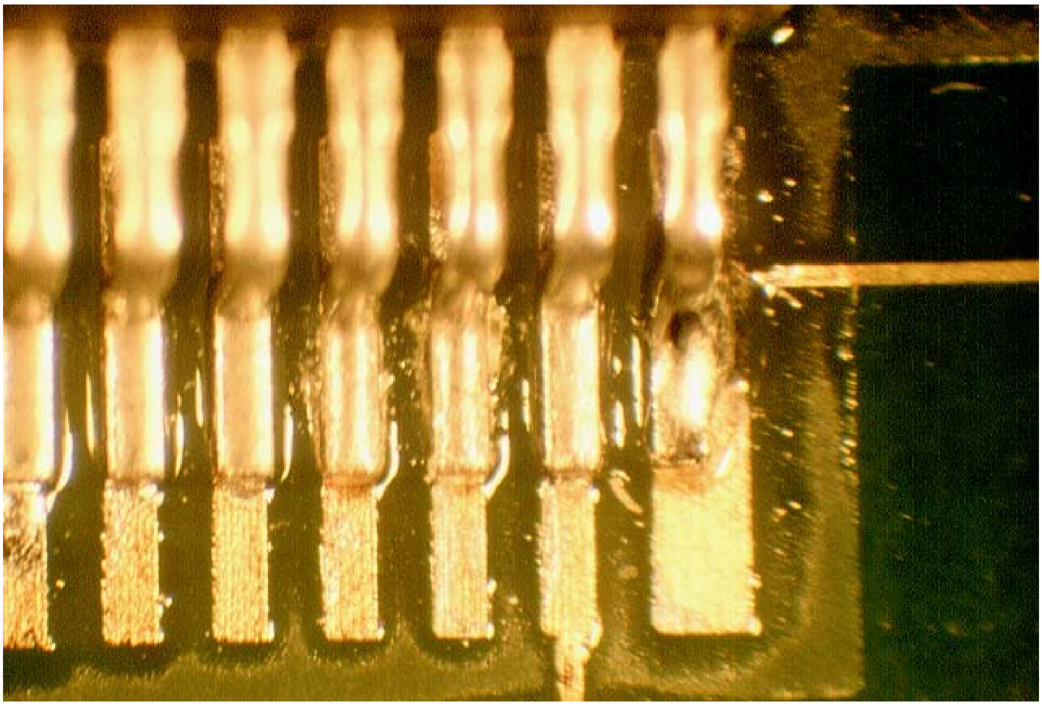
In reality not all solder joints reflow again and in many cases only selected terminations on each part become liquid, hence the importance of boards being supported and held flat. Secondary reflow of selected terminations can lead to intermittent joints if the board is not supported flat. The same thing happens on wave soldering if the topside of the board gets too hot and fine pitch terminations reflow. As most boards sag during wave soldering selected leads can lift.

Many companies have suggested that reflowing joints may adversely affect the reliability of the joints due to the increase in intermetallic thickness of the joints. It is fair to say that the intermetallic thickness in joints may increase but that would also happen during rework. Studies conducted by companies in the USA have concluded that although intermetallic thickness does increase it is not likely to decrease the reliability of the product for its intended working life.

Solder Voids

Voiding of solder joints during initial reflow should be avoided as it is an indicator that the process has not been optimised. This is due to the volatile vapours or the no metallic part of the paste not escaping prior to solidification of the alloy. Voiding is however often seen in boards produced by DSRS. The voids, as shown in the example below, are commonly seen on the joints produced on the second pass. This may be caused by the added component mass causing a delay in reflow and a limited time for the paste in a liquid state. This is why temperature profiling is so important.





Example of solder wicking away from the surface of the pad. This is caused by slow wetting of the pad surface during reflow with all the solder being wicked to the lead. This effect can occur if the solderability of the lands is poor or if it is affected by the initial reflow operation prior to second side reflow. In this particular case it is shown on a nickel/gold board.

A further example of solder wicking, in this case on an OSP coated board. The solder paste has all wicked up the termination due to slow or poor wetting of the copper surface. Again this may be caused by incorrect temperature control during initial reflow but is more likely to be a problem with the coated board prior to assembly.

Double Sided Reflow Survey Results

The following survey results were produced by Victor Li and provides the responses from a survey form placed on a number of E-mail forums. It provides an insight into current thinking on the use of double sided reflow soldering in modern manufacture.

Survey of double sided reflow soldering practices in the PCB/SMT industries

A survey was circulated via the Internet to professionals in the PCB/SMT industry. There are three Internet forums which serve as an idea/problem exchange medium to help insiders to share their professional knowledge globally. They are SMTnet, Webmaster and Technet. The purpose of the survey was to access the general practices used by the industries performing double sided reflow operations. In brief, the survey is designed to show the different techniques used in industry.

General Information:

I) Type of reflow oven used : Infrared conventional, Force Air, Force Air with Nitrogen, etc.

II) How frequently double sided reflow soldering is performed

III) The size of the printed circuit board being used

Process used for double sided reflow soldering:

IV) The connecting medium people choose to perform the double sided reflow soldering:

1) conductive adhesive + solder paste
2) same solder paste on both sides
3) high melting point solder paste + standard solder paste
4) low melting point solder paste + standard solder paste

(note: unless specified, it is general practice that solder paste refers to the most commonly used material which has the melting point around 183^{o}C)

V) The difficulties of performing double sided reflow soldering

VI) The frequency components fall off during the second reflow

VII) Reasons that contribute to the defects in VI)

A sample questionnaire is included in this report.

Results and Analysis

The survey was conducted for a period of two months. As a result, 37 questionnaires were returned, with other responses asking for the analysis of the compiled results. This shows the validity and general interest in the questions being asked.

Exhibit 2 shows the data sheet listing the respondent opinions regarding categories I, II, III, IV and VI. Figures 1 to 5 show these results graphically.

Figure 1 shows the percentage distribution among the respondents using various types of reflow oven. More than 60% are using forced air reflow ovens in the reflow soldering process. Infrared conventional ovens and forced air with nitrogen atmosphere ovens basically contribute to the rest of the sectors.

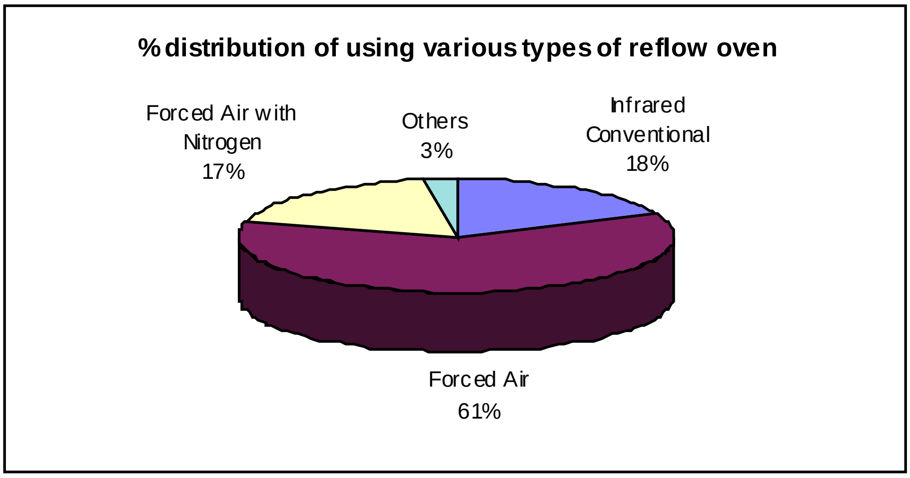


Figure 2 shows how frequently the respondents perform double sided reflow soldering in their production lines. Of the 37 respondents, 13 of them perform DSRS more than 75% of the time, while 11 of them perform DSRS less than 25% of the time. 12 of the respondents perform the DSRS more than 25% but less than 75% of the time. Thus, this should be a good mix to represent the overall picture in the industries.

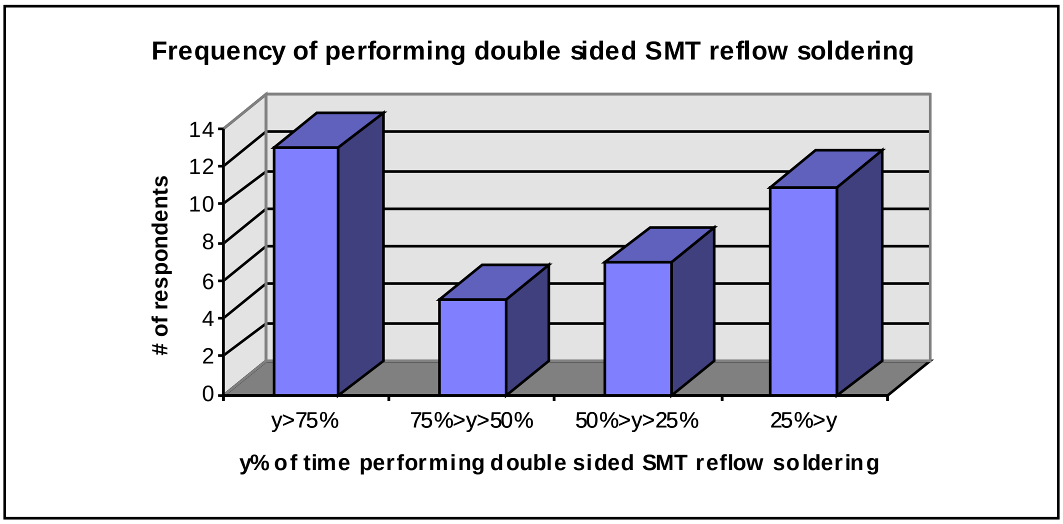


Figure 3 shows the length of printed circuit boards commonly used in the DSRS. Results show that majority of the printed circuit board are larger than 100mm.

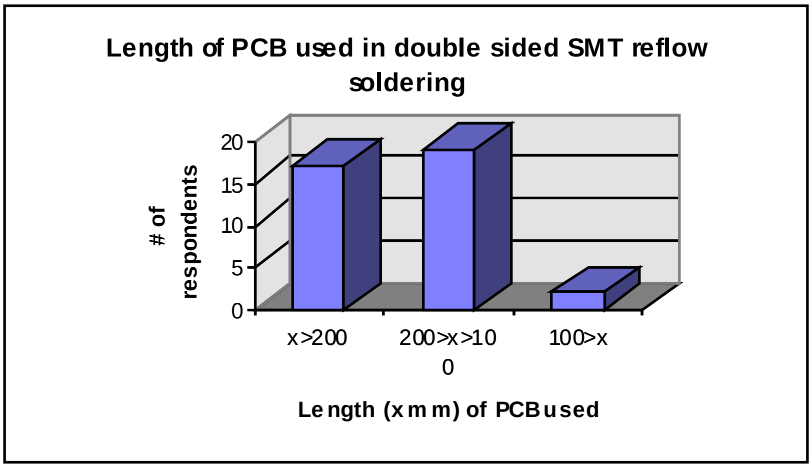


Figure 4 shows the percentage distribution of companies using various types of connecting media which bonds the components on both sides of the printed circuit board. 83% of the respondents use the same solder paste alloy during DSRS. 11% of the respondents use the conductive adhesives on one side and solder paste on the other side. Only 6% of the respondents use high melting point solder paste along with normal solder paste alloy. However, the survey indicates that none of the respondents use low melting point solder paste in the DSRS.

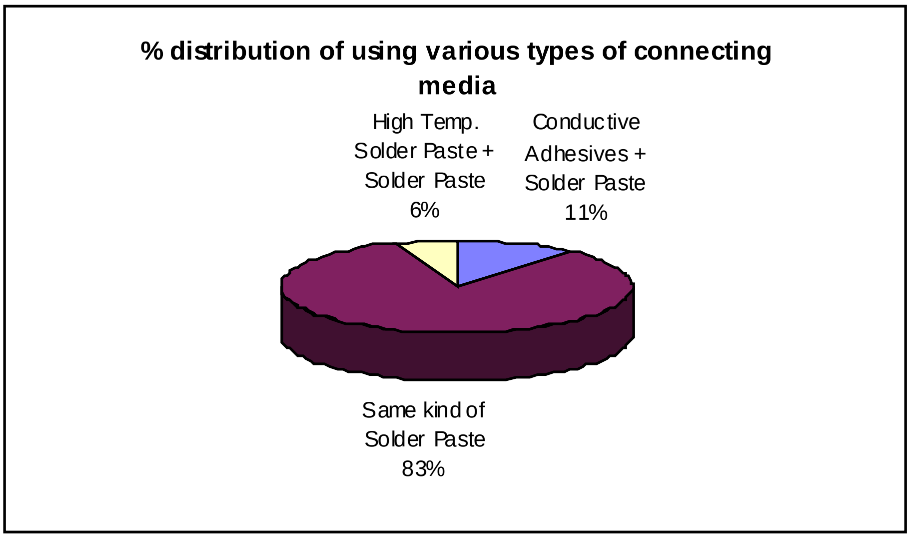
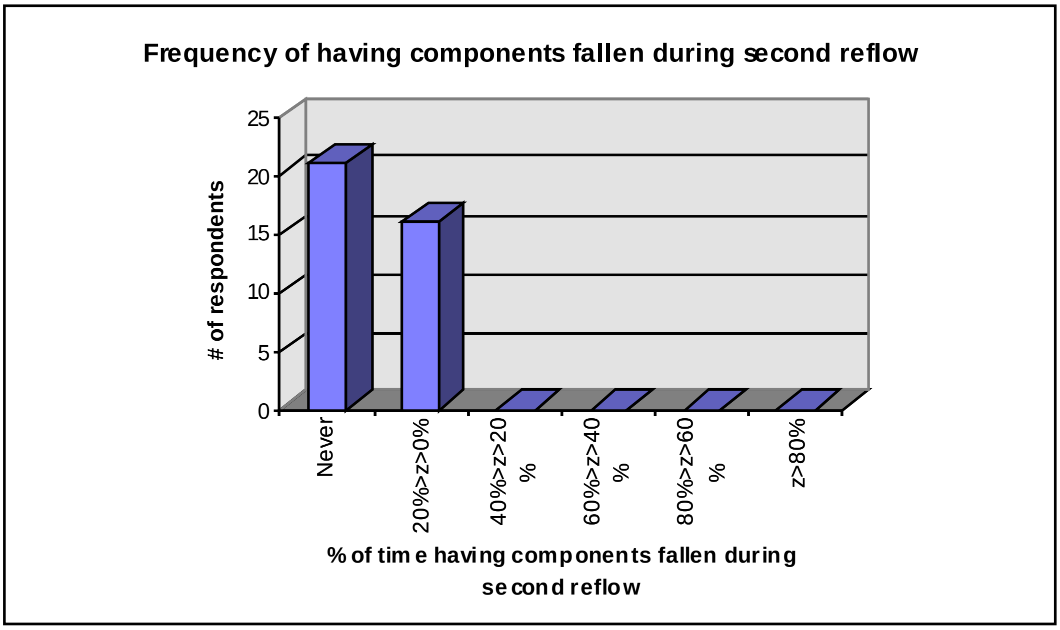


Figure 5 shows how frequently the respondents experience components falling during the second reflow. 21 respondents claim that this has never happened. 16 respondents claim that they have experienced such problem less than 20% of the time. Yet, none of the defect rates exceed 20% of the time, which is quite reasonable, as assembly engineers would not tolerate any defect rate higher than this percentage. Otherwise, the production would be stopped to examine if the equipment, raw materials, etc. contribute to the problem.



To explore the problems behind the double sided reflow soldering, questions 5 & 7 try to uncover the industrial experience. From figure 2, we can see that majority of the respondents are performing double sided reflow soldering either more than 75% of the time or less than 25% of the time. Thus, we can use these two extreme categories to determine the different problems encountered by the frequent and infrequent users.

Frequent Users

Of the 13 respondents who perform DSRS more than 75% of the time, six of them claimed that they had experienced components dropping from the bottom during the second reflow. They said that the reasons for the lost components were mainly due to the excessive vibration through the conveying belt, inadvertently touching the printed circuit board by the operators. Moreover, excessive weight of the large components or additional heat sinks connected to the components. When the PCB is washed after first reflow and before the second reflow, or when there is bending occurring to the component leads, the components are also likely to be lost.

Infrequent Users

Of the 11 respondents who perform DSRS less than 25% of the time, four of them claimed that they had experienced components dropping from the bottom during the second reflow. The reasons for component loss included vibration of the conveyor belt, air flow within the oven which may loosen the components, the temperature at the bottom side of the PCB becoming too hot, etc.

General comment

In terms of the robustness of the DSRS, the respondents were asked if they found the assembly process difficult to control. Most of the respondents to the survey indicated that it would be an easy process if the profile could be properly set. Also, suitable fixture to mask off the heat from the underside of the PCB is needed to keep the temperature at around 170^{o}C. In general, if the components underneath the PCB are small or light, the surface tension of the solder paste should be able to hold them in place.

However, when we step back and think about the comments, we will find some areas we can improve. Setting a suitable temperature profile is already time consuming for a single sided PCB, when process engineers have to balance the temperature distribution across the complicated component distribution. When the process engineers need to consider a good temperature profile across the top side of PCB, and at the same time try to keep the temperature on the bottom side below 170^{o}C, it would be a time consuming process, if not a painful one.

Moreover, if it is difficult to maintain the temperature underneath the PCB well below 170°C, but instead the temperature at that side be around 183°C, which is the melting point of the solder paste most commonly used in the industries, the solder paste on the bottom side will reflow again. Thus, it would be possible for some components to be misaligned. In some circumstances, some components may be too heavy for the surface tension of the melting solder paste to hold in place; plus the jerky motion of the conveyor belt, these components may just fall off in the reflow chamber.

In order to reduce the problem, process engineers will only suggest to the PCB designers to reduce the number of components and only use lighter components on the bottom side so that they will be less likely to experience defects during the second reflow. Thus, the limited area on PCB may not be fully utilised.

Today this would not be acceptable to the design engineer as often designs are now heavily populated on both sides of the board with an equal balance of heavy components. There are some occasions where products are returning to single sided surface mount. This tends to be in the high end consumer and telecommunications markets where design engineers are forced to increase the complexity of the interconnection in the printed board and decreasing the number of parts being used. This is often due to the need to make a thinner end product.

Questionnaire for Double Sided Reflow Soldering

Type of oven used :

( ) IR conventional
( ) Forced Air
( ) Forced Air with Nitrogen
( ) Others, pls. specify ________________

Brand Name of the oven : ________________________

How often does your shop floor perform double sided reflow process?

( ) 75 % of the time, or above
( ) 50 < x < 75 % of the time
( ) 25 < x < 50 % of the time
( ) less than 25 % of the time

What is the length of the PCB commonly used in double sided reflow process?

( ) 200 mm or above
( ) 100 < x < 200 mm
( ) 100 mm or less

Which connecting medium do you use in the double sided reflow process?

( ) conductive adhesive + solder paste (melting pt. 183 C)
( ) the same kind of solder paste
( ) high temp. solder paste + solder paste (melting pt. 183 C)
( ) low temp. solder paste + solder paste (melting pt. 183 C)
( ) Others :

____

____

Do you find this process difficult to control? Why?

____

When doing the second pass, how often did the components on the bottom side fall off?

( ) 1. Never happen

( ) 2. 0 < x < 20% of the time

( ) 3. 20< x < 40% of the time

( ) 4. 40< x < 60% of the time

( ) 5. 60< x < 80% of the time

( ) 6. 80% or above

If you answer 2. to 6. please list the components which caused problems :

( ) BGA, # of pins : _____ Size: _____ mm x _____ mm

( ) PLCC # of pins : _____ Size: _____ mm x _____ mm

( ) QFP # of pins : _____ Size: _____ mm x _____ mm

( ) Others(specify) : Type ____ # of pins ____ Size: ____mm x ____mm

: Type ____ # of pins ____ Size: ____mm x ____mm

What might be the suspected reasons for the fallen components?

____

____

SOLDER FINISH MARKET SURVEY RESULTS

A survey was conducted during the third quarter 1996 on the trend in solderable finishes used on printed board assemblies.

The latest survey results were based on 22 PCB manufacturing respondents and showed the most widely offered finish was still solder levelling (65%) with traditional tin/lead reflow (15%). Gold over nickel was offered by (20%) of the companies.

The printed board manufacturers who responded indicated that in terms of volume of boards or square inches solder levelling was the most often specified finish by customers, (72%) followed by traditional tin/lead finish (10%) Gold (10%) and organic coatings (8%).

The solder levelling process was indicated by 90% of the companies as offering the best compromise between the requirements of the assembler and the ease of fabrication.

In terms of popularity with customers the following finishes are now being most often requested for surface mount products. The finishes are listed in order of popularity.

1. Solder Levelling
2. Nickel Gold
3. Copper OSP
4. Tin/Lead Reflow
5. Silver

The following are the average costs quoted based on a specific four layer PCB cost model with specific delivery quantity. The tooling and test fixture cost is also shown for the cost model.

OSP finish	500 items	£7.60 each	1000 items	£6.99 each
Hot air levelled	500 items	£9.77 each	1000 items	£8.85 each
Nickel/gold	500 items	£9.82 each	1000 items	£9.42 each
Silver	500 items	£8.75 each	1000 items	£8.55 each
Tooling cost	£127			
Electrical test fixturing	£379			

Note Used in the Main Report

In any double sided reflow application care needs to be taken with the exit temperature of the reflow oven. In the case of OSP coatings the lower the exit temperature the less effect on the final solderability is seen. Trials on OSP boards have shown that dropping the surface temperature below 40C as quickly as possible is beneficial.

On a practical note the board surface must be at ambient temperature before entering the printing process for second side as a hot board will effect paste slump during or after printing.

Although it is possible to measure the weight of all devices it is extremely difficult to measure solder forces on small parts.

Stuff to do

Video and process material
Video reflow of boards up side down with hot air and flex boards.

Obtain example pallets used for masking for wave solder.
BGA Components reflowed upside down and tweak parts on the base of the board

Issues
flexure
Vibration
Air Turbulence
Poor soldering on one or two sides of a four sided device
Uneven lead loading

Close up of joints reflowing