Introduction to Simultaneous Double Sided Reflow Soldering

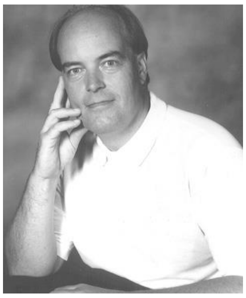
Produced by Bob Willis

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) 01245 351502	Fax	(44) 01245 496123
E	bob@bobwillis.co.uk		
W	www.leadfreesoldering.com	W	www.ASKbobwillis.com

INTRODUCTION

In any process there are always opportunities to improve, reduce the number of process stages or reduce the cost of manufacture. Each may not improve yield initially, may even increase manufacturing defects, but it must in time benefit some companies' processes or products. Engineering is always looking at alternative processes and Simultaneous Double Sided Reflow (SDSRS) is a possible new candidate for investigation. It is currently being considered by two Japanese companies, one large computer manufacturer in the USA and reportedly being used by Philips and one other company in Europe.

PROCESS SEQUENCE

The process of SDSRS includes the following stages:

Stencil Print Paste	Side One
Dispense Adhesive	Side One
Alternatively Screen Print Adhesive	Side One
(Using Recessed Stencils)	
Alternatively Dispense Paste & Adhesive	
Place Components	Side One
(Possible UV Curing Stage for Adhesive)	
Invert Board	
Screen Print Paste	Side Two
Place Components	Side Two
Possibly Insert Through Hole Components	
Reflow Solder Both Sides Simultaneously	

Screen printing side one is conducted in the normal way for the SDSRS process unless solder paste dispensing is going to be used for both process stages

After the application of paste, adhesive is placed on the surface of the board by dispensing prior to component placement and curing

An alternative to dispensing adhesives is stencil printing which may be conducted using a recessed stencil

After the second side of the assembly has been printed and components placed, the whole assembly may be reflowed in one operation

The question has to be asked why consider SDSRS as it will undoubtedly cause problems with loss of components in the early stages of process development. If we examine the time taken for the reflow cycle of a product, typically this is 3-4 mins. By reflowing both sides of the assembly then the time equates to a total of 8 mins. By conducting simultaneous reflow then the process cycle time is reduced with time saved on the total process cycle.

It used to be the case that placement was the slowest part of the process but with increasing speeds this is not the case. Printing was the last part of the process to be attacked by the speed merchants focusing on the solder paste specification which was the limiting factor in high speed printing. The formulations of paste have now provided the speed to make other parts of the process the weak links in the chain with cycle speeds for printing being less than 10 seconds per board.

SDSRS Advantages

Eliminates one reflow process stage
Only one heating process for double sided boards
Reduction in capital equipment
Reduced floor space requirements
Reduced handling stages
Increased PCB throughput speed
Potential elimination of component weight issues
Improved solderability of OSP circuit boards

Disadvantages

New process introduction
Possible loss of components
Gluing stage required
New glue materials required
Glue curing process
Possible limits on component positioning

ADVANTAGES

Eliminates one reflow process stage

With simultaneous reflow soldering both sides of a surface mount board can be conducted in one operation. If through hole parts are also designed to be reflowed this can be done in a single operation.

Only one heating process for double sided boards

With only one heating process there is a potential benefit to the reliability of the joints with a thinner intermetalic formation being possible. The issues related to reduction in solderability of surface mount pads are also eliminated.

With only one heating process during reflow soldering, in which the board may be supported with a central wire, it is possible to reduce warpage of boards susceptible to this problem.

Reduction in capital equipment

Eliminating a reflow process means one less piece of equipment in a production line is required. The cost of power consumption and extraction systems for reflow equipment is higher than any other piece of equipment on the line and they are effectively halved.

Reduced floor space requirements

Eliminating a reflow oven reduces the floor space required in a manufacturing area. Even with the need for an additional printer or the use of a single dispenser for both adhesive and paste, floor space is still being saved.

Reduced handling stages

The number of handling stages is reduced from a conventional surface mount process, albeit only slightly reduced from a fully double sided reflow process.

Increased PCB throughput speed

The total process cycle time for each product is reduced by the elimination of one reflow soldering stage. The increased cycle time for one printing or dispensing stage is still far less than the period taken to reflow one side of a board.

Potential elimination of component weight issues

In a liquid state there is a limit to component weight during traditional double sided surface mount assembly. Large heavy components can fall off the board. If adhesive is used and cured it will increase the range of components that can be used on both sides of the board.

Improved solderability of OSP circuit boards

There has been great debate on the benefits of using OSP: improved printing and placement, cost reduction and improved yields. OSP coatings have been shown to deteriorate during two reflow cycles; with one reflow process stage the problem is eliminated.

Note: It is the opinion of the author that most of the problems seen on OSP coated boards stems back to the quality or preparation of the original coating. Potential users of OSP should understand some of the simple process evaluation techniques or goods receipt tests.

DISADVANTAGES

New process introduction

With any new process there is a learning curve, particularly with new materials and processes. There will also always be a degree of reluctance on the part of some company departments to invest time on a new process. Always ensure that a justification has been well thought out prior to starting tests.

Possible loss of components

Component loss may occur if the adhesive is not cured or if products are incorrectly supported during the different process stages. Trials have shown that if a curing process is used them loss of parts on a normal process are eliminated.

Gluing stage required

To allow this process to be conducted successfully a gluing process is required, which requires either a printer or a dispensing system. It is one additional stage at the start of the process.

New glue materials required

The use of UV curing materials has declined for surface mount application in favour of thermally cured products. Most development of materials has focused on either their high speed dispensing or printing characteristics. Examination of the material properties of UV curing products will require further engineering time.

Glue curing process

If the process is to be successful the curing cycle or part curing will need to be achieved in <10 seconds without heating the board surface. The UV lamp would need to be placed over a conveyor suitably screened to meet all health and safety requirements. If the time to fully cure was insufficient then it would also be possible to place a UV source under a conveyor section after board turnover to supplement the cure. This may also allow a lower intensity source to be used.

Possible limits on component positioning

When using glue to hold parts on the base of the board there will be some limitations just like with the use of double sided reflow. BGA, CSP and most four sided devices may not be useable. The access for positioning glue spots may be restricted. This is dependent on the capability of the adhesive application process.

SDSRS PROCESS ISSUES

The most obvious problem with this process is the potential loss of components during flipping of the board, transportation or during the reflow cycle. Side one components may only be held by the paste tack characteristics and the green strength of the adhesive. If the process of adhesive dispensing is not controlled it can lead to contamination of the solder paste print. Adhesive may be mixed with the paste contaminating the joints either during dispense or during the reflow stages.

Process trials have shown that chip components, SOT23, SOT89 and SOIC16 parts will be held in place with the paste and adhesive holding the part in place without any curing stage. The limit has been found to be PLCC devices of 40 pins and above. Further trials are being conducted on small QFPs, TSOP and fine pitch SOIC devices. Although the adhesive volume can be achieved to secure the parts initially the glue will not hold heavy components on the board during reflow unless it is cured first. If a curing stage were possible without causing the paste to slump it would reduce the speed of throughput making the use of SDSRS of little value.

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

Component	Weight	Terminations
Chip 0805	0.007g	2
Chip 1206	0.009g	2
Chip 1210	0.012g	2
SOT23	0.008g	3
SOT89	0.0976g	4
V/Resistor	0.124g	3
MELF	0.1325g	2
Mini MELF	0.031g	2
SOIC8	0.102g	8
SOIC12	0.123g	12
SOIC16	0.142g	16
TSOP20	0.212g	20
PLCC28	0.688g	28
PLCC44	2.21g	44
PLCC68	4.67g	68
PLCC100	9.97g	100
LCCC44	5.32g	44
PLCC84	5.21g	84
PLCC100	9.97g	100
QFP100	4.24g	100
BGA225	2.65g	225
BGA313	6.32g	313
CBGA256	24.21g	256
CCGA625	29.53g	625
CCGA1089	34.12g	1089
Tape BGA360	2.695g	360

Like traditional double sided surface mount the surface area of the lead forms is different on gull wing and “J” lead parts. The approximate surface in contact with the gull wing is 0.040” x 0.018” with the “J” lead having less 0.010” x 0.014”. Video sequences have been taken which show successful reflow of SOIC devices on the base of the board. The PLCC devices fall off during the pre heat stages of reflow as any tack strength is lost. Part of the problem here is the weight of the component and the other is the surface of the lead in contact with the solder paste.

If the adhesive can be cured very quickly during transportation between placement and the board turnover unit then the problem of component loss is reduced. Unfortunately surface adhesives used today are primarily thermally cured unlike the original UV materials used in the early days of SMT. Most suppliers do, however, have materials which can be UV cured but that in turn requires the adhesive to be placed outside the component body footprint so the material is exposed to UV radiation. In SMT today the majority of the adhesive dots are placed under the component where the UV light would not be seen to start the curing process.

Recent trials on components have shown that cure strengths in excess of 400g can be achieved on components with very short UV exposures of less than 10 seconds. The trials were conducted using a product supplied by Loctite.

Full reports on the use of double sided reflow soldering and the use of pin in hole intrusive soldering are available from the SMART Group offices. Both reports should be read in conjunction with this report to fully understand the process issues involved with SDSRS.

To improve the process capability a lot of process engineering work needs to be done to achieve successful results with this process. Consider each of the manufacturing stages in a little more detail.

PCB SOLDERABLE FINISH OPTIONS

The board finish is now becoming a major issue in fine pitch printing; conventional tin/lead coatings do not provide the ideal surface for printing. The selective tin/lead solder levelled surface on most circuit boards is uneven across the board. This will result in inconsistent thickness of paste deposits. In the worst case it will not allow direct contact between the stencil and pad surface. Adhesive dispensing is also affected due to contact of height probes on the tin/ lead surfaces. The graph below shows the results from some of the author’s trials and the defect levels in terms of PPM (Parts Per Million) defective.

The results were obtained after running mixed batches of boards through a production process. No attempt was made to optimise the process for any one finish, which of course is possible. The different batches of board were mixed together to make the comparison more revealing.

The graph shows results after inspection at screen printing for different printed board surface coatings. The flat surfaces provided the best performance for printing and Hot Air Solder Levelling the worst.

For fine pitch printing the ideal surface is a flat pad which will assist in providing a consistent paste thickness by allowing the stencil to gasket on to the pads. Some of the available alternative coatings are silver, copper preservative coatings like Entek, palladium, immersion tin and nickel/gold.

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.

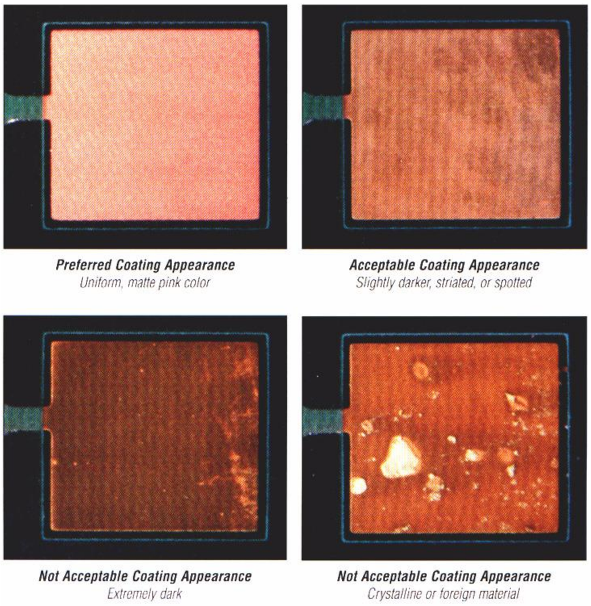
The protective coatings have been used for many years by large volume manufacturers for surface mount products, an example of which is IBM. The limitations of the coating was its general inability to stand up to multiple soldering operations. The coatings are degraded by exposure to high temperature 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 future due to their competitive cost which is much less than any other finish.

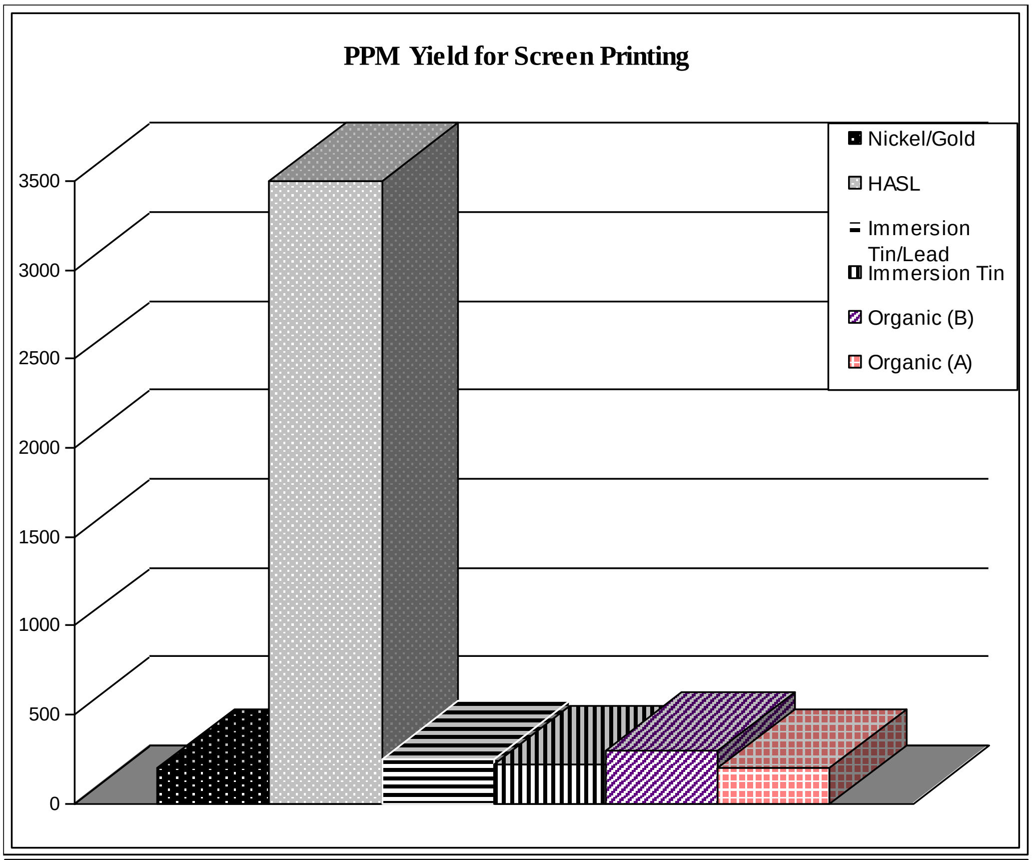
Recent trials have indicated that cooling rates after reflow should be improved to reduce the effects on copper coatings. Cooling the board surface directly after reflow below $80^{\circ}C$ can prolong the solderability life of the OSP coating. Use of nitrogen during first side reflow with an oxygen 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 cooled for second pass printing in a high volume operation.

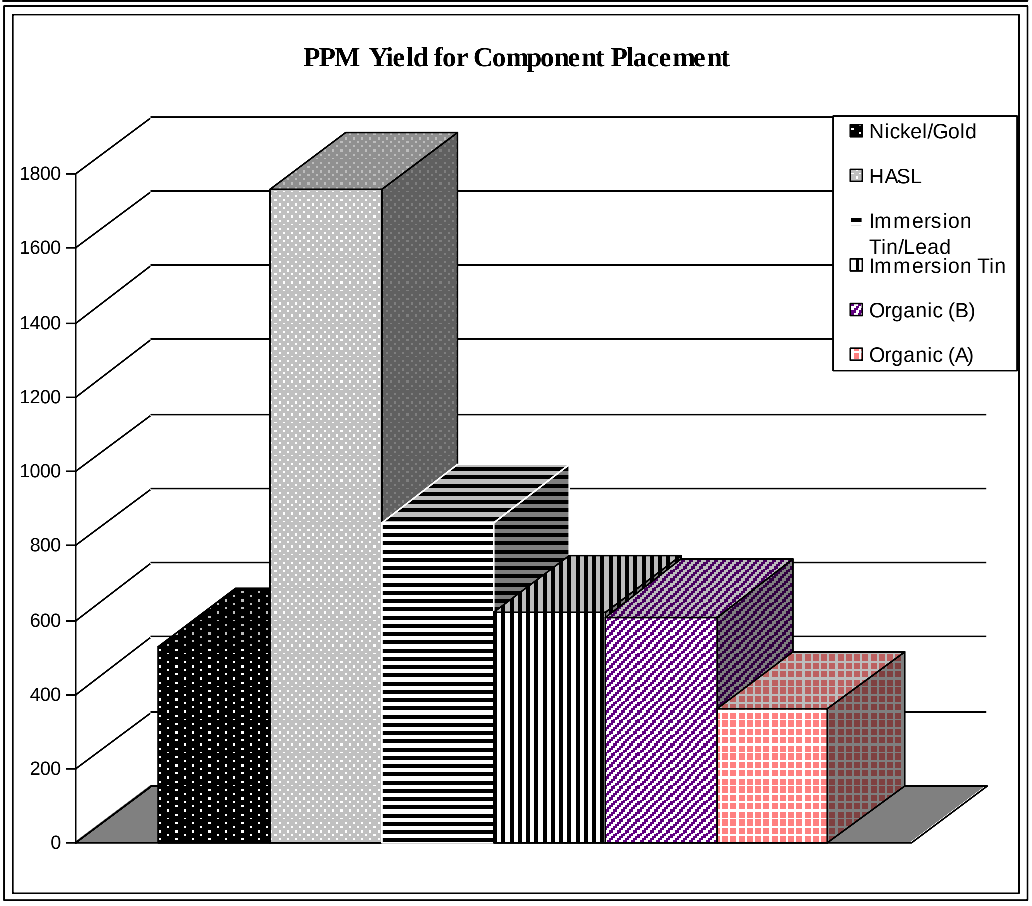
OSP is currently used by divisions of IBM, Siemens, Motorola, AT&T, Olivetti, Compaq and Dell. It is also a common process offered by many printed board suppliers. OSP coated boards were recently shown by Motorola to provide better joint reliability than gold or tin/lead.



These guidelines for OSP board acceptability are available from Enthone, the producers of Entek, one the of the most well known OSP coatings in the industry

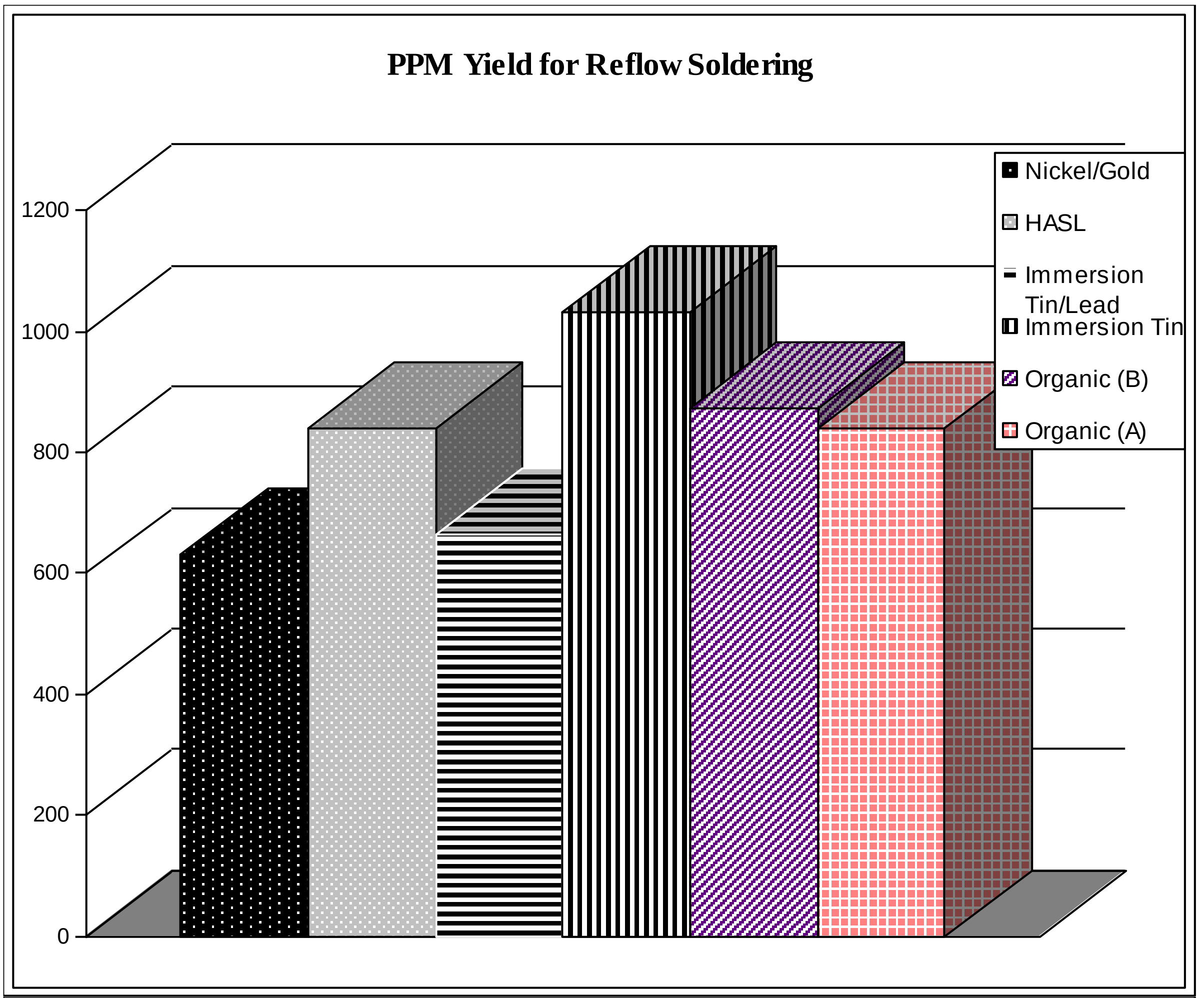
The four graphs that follow are taken from the SMART Group/Shipley Europe Solder Finishes Report produced by the author in 1997 which show the effect on assembly yield when using alternative solderable finishes.

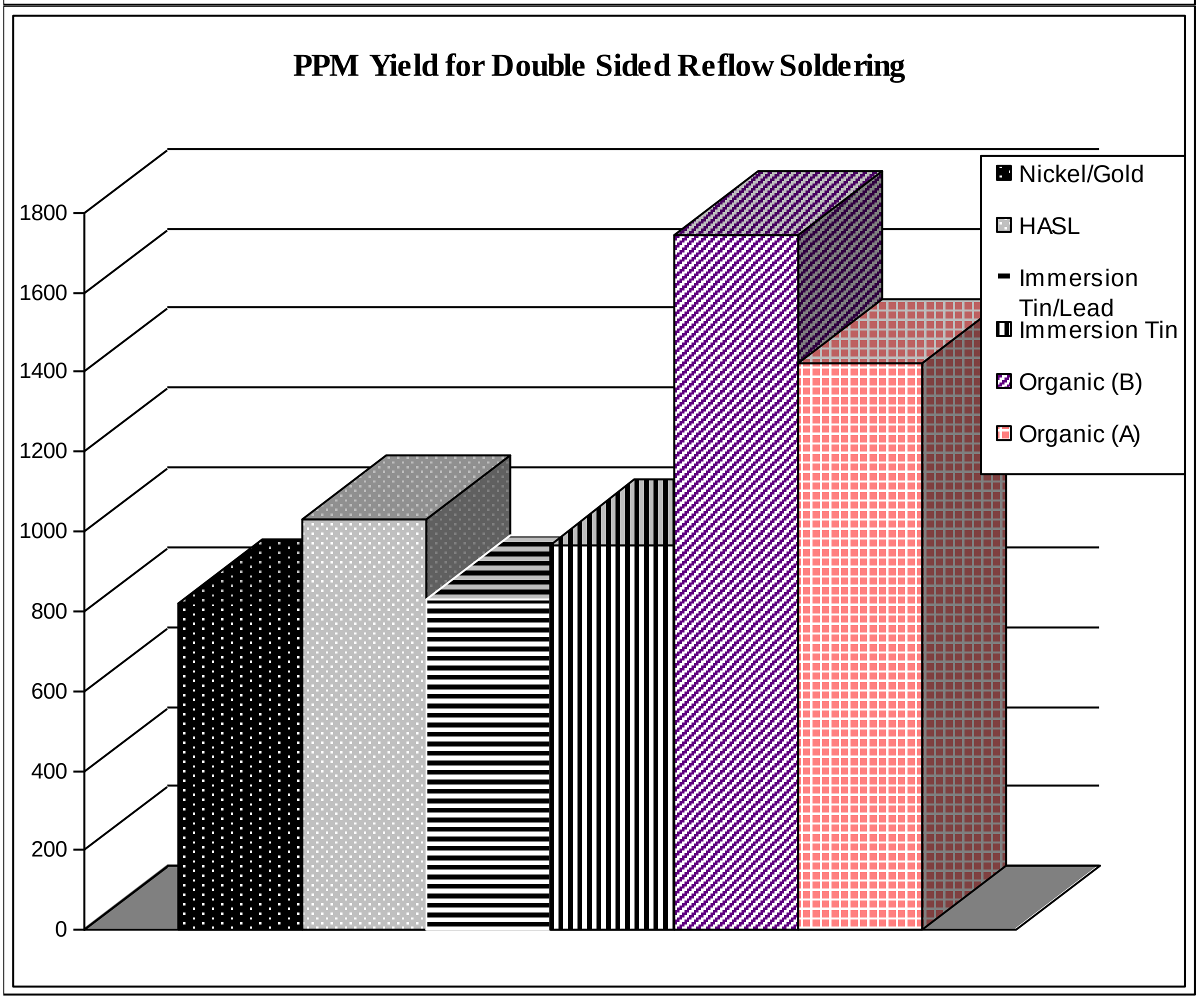




The first graph shows the yield from screen printing which illustrates the highest defect rate on solder levelled boards. The most common defect was halo prints where the solder paste was only around the outside of the pad due to the solder coating coming into the stencil aperture.

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.





These graphs show defect levels after first and second side reflow. Most of the differences are due to the changes in solderability due to the first reflow process.

SCREEN PRINTING

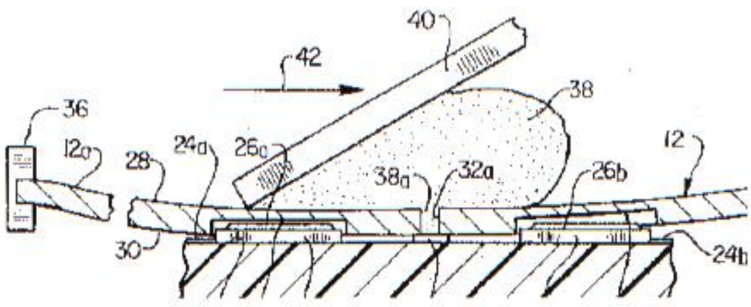
The solder paste printing stage is relatively straight forward as the process is well defined for traditional surface mount. The main area of investigation is the board support system and the board stops. Traditional vacuum or pin supports can be used during the first printing operation which would probably be the least demanding in terms of fine pitch printing.

During second side printing paste, adhesive and components will be positioned on the base of the board so tooling will need to be reviewed. Any component contact by the tooling cannot be tolerated as the components may be dislodged by contact or flexure of the board. The flexure of the board or initial impact of the tooling on the board could also subtly move parts. Most printer suppliers are well aware of these problems for conventional double sided assembly but this process does add an additional burden.

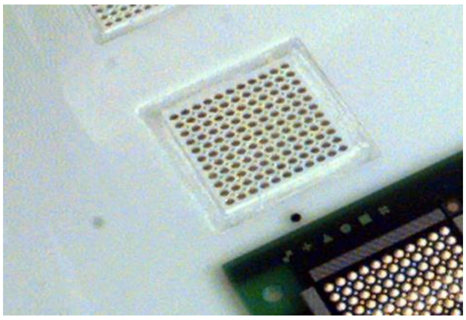
When a board or panel enters the printing process it will be brought to a stop with a soft or hard stop. Generally this refers to the amount of force seen as the board stops prior to optical recognition. The amount of inertia on the components on the base of the board may be sufficient to move or displace parts.

A further alternative process which may be adopted for printing paste and then adhesive utilises the use of stepped or recessed stencils. In this case the first paste printing process is conducted as standard. When completed a second printing process is used for the glue deposit with a recessed stencil. The base of the stencil is etched out to allow the printing of glue without the stencil contacting the paste print area.

Recent discussions on smart-e-link, the SMART Group Internet Forum have shown that stencil manufactures like DEK have already had experience of this process. The process has also been granted a patent in the USA to Compaq Computers and can be investigated on Patent Web Site at http://www.patent.womplex.ibm.com/details?pn=us05627108_&language=en



The drawing above has been taken from the patent granted to Compaq Computers. Further details on the patent can be obtained from the web site



Close up of stencil apertures and board after printing adhesive onto the surface of the board with existing paste, deposit courtesy of DEK Printing Machines

Solder Paste Stencils

The solder paste stencils used for SDSRS do not differ from traditional metal foils for surface mount applications apart from the possible use of recessed foils. The design rules and process of manufacture will be dictated by the type and pitch of components in the design. A guide to stencil types and specification is included here for engineering departments new to the use of reflow technology. Further design information and specifications can be obtained from the leading stencil suppliers to the industry like Alpha Metals, Chemtech, DEK and Tecan. Each of the suppliers have information on their Internet Web sites that will be of value.

Solder Paste Stencil Types

Brass

This used to be the most extensively used stencil material but in some areas this has changed to steel. The advantage of brass is that it is easy to etch and is readily available in the standard stock thickness required for the printing of solder paste.

Brass is etched from both sides at once. This is done in order to avoid the opening in the stencil being much larger at one side of the aperture than at the other. The reason for this is that the etch system is a chemical milling process, and is done using an etch resist to expose the areas in the surface to be removed. The etching fluid fills the pattern in the mask which will end up as the stencil apertures and starts to etch its way through the metal.

Unfortunately however, because the top of the brass is contacted for a longer time than the bottom, apertures formed in this way are tapered from top to bottom, and are in fact larger than the original artwork. For this reason, the etching is done from both sides of the brass at once in order to minimise the effect. Control of the etch solutions, temperature of the etching agents and the effectiveness of the spray application can have a major effect on the quality output from a stencil supplier. The basic materials used can also affect the quality of the etching process.

In addition to the brass being chemically milled from both sides at once, the artwork is modified before use by the designer or stencil manufacturer, in order to make the artwork aperture smaller to compensate for the etching effect. This means that the apertures in the stencil will eventually be the size of the apertures in the aperture list.

If artwork is supplied to stencil suppliers it will normally be a 1:1 of the final printed board pad layout. In order to make this smaller, the stencil manufacturer puts the artwork through a process known as "wobbling" or micro modification. This is a photographic process which reduces the pad size but has the effect of rounding off the corners on square aperture pads.

In order to avoid this effect with very fine pitch devices, it is possible with some laser plot bureaux to have them compensate for the etch at the laser plot stage, and actually plot the pads smaller. The stencil supplier can of course also provide this service provided the design engineer has not already taken this into consideration on the artwork, otherwise pads will be compensated twice and end up too small for printing.

The advantage to this approach is that the artwork is modified, but the corners of the stencil pad apertures do not get rounded off. Over the years different viewpoints have arisen, in many cases not substantiated by scientific evaluations. The difference between rounded and square apertures has never been demonstrated to be a problem. The main issue is the visual appearance of the deposit after reflow. If you have a square pad and a rounded stencil aperture there may be evidence of no wetting on the corner of the pad. This will be due to the wetability of different surface coatings and the lack of paste in this area. It is not going to affect the reliability of the solder joints.

On stencils, there may have to be some allowance for a mixture of technologies on the board. Different pitches may require different volumes of paste. The classic answer to this is to differentially etch the stencil. In a board with a mixture of technologies, this would mean that a typical stencil thickness may be 0.006" in one area for 0.020" pitch devices and 0.008" thou overall thickness, to allow for the 0.050" surface mounted devices.

On the fine pitch devices the top of the stencil is locally etched away around the whole component footprint thereby locally reducing the stencil thickness from the squeegee side. This works fairly well, but generally around 3-5mm clearance is needed around the fine pitch component. The reason for this is that the localised reduction in thickness has to be larger than the component in order to allow the squeegee blade sufficient travel to settle into the locally reduced cavity in the stencil. In most modern designs this clearance around a fine pitch device is actually not possible, since the whole idea of using fine pitch in the first place is generally to increase the packing density of the circuit.

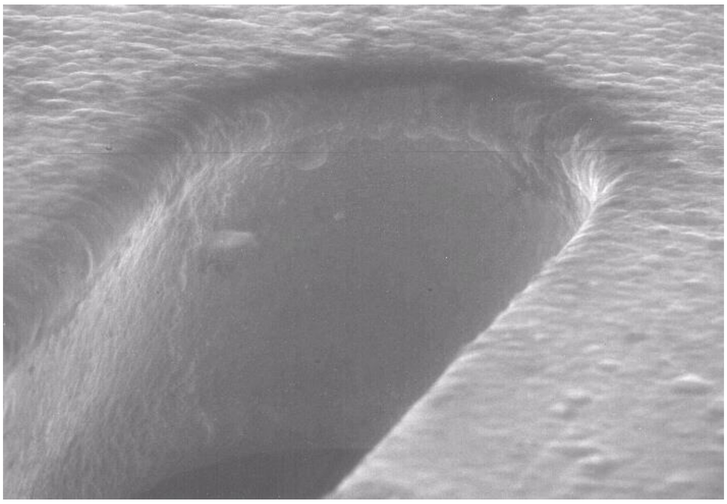
There is another answer that works well with both 50 thou surface mount and fine pitch packages, and once again the answer is in the original designer's software before this is laser plotted.

For a fine pitch surface mounted or TAB device it is feasible, and perfectly acceptable, to reduce the amount of solder paste locally by reducing the cross sectional area of the stencil aperture. This can be done most readily by reducing the size of the pads on the footprint on the CAD system for the solder paste artwork only. Alternatively, some laser plot bureaux are able to handle this requirement at the plotting stage. However, since it is a part of the board design, it is better to lay down these rules as part of the CAD system library for these components since that way there can be no question of whether the artwork needs modification or not.

In practice a 25% overall cross sectional reduction is best for devices at a pitch of around 0.025", and 0.030" to 35% for pitches lower than this. This means in effect that if we have a pad with a length of say 1.5mm and a width of 1mm, then for a 25 thou pitch device the artwork for the paste deposition would have the length of the pad reduced by 25% overall. This reduction should be done equally from both ends of the pad by taking 12.5% from each end. This would give us an aperture that, although smaller, is still on the original geometric centre of the pad but with a dimension of 0.5mm across the pad, and a length of 1.125 mm as opposed to the original 1.5 mm. This practice can be applied to any data that is destined for stencil manufacture for solder deposition on fine pitch devices.

<u>Nickel plating after etching.</u>
The Nickel plating, is said to give a smoother wall to the aperture, and for this reason, a cleaner break out of the paste from the stencil when it is separated from the board. Viewing the aperture walls the use of this technique does improve the appearance and reportedly the paste release due to the lower friction forces. The nickel does not, however, overcome poor etching process due to the thin coating applied. The quality of the aperture must be of the highest quality before nickel plating. In the paste the nickel has also been promoted to reduce oxides from the brass surface affecting the paste and causing problems of solder balling, this has never been proved.

Care should be taken when evaluating any stencil surface for print compatibility; paste has been seen to skid across a surface rather than roll in front of the blade as a bead. Both nickel and stainless steel masks have suffered from this problem if the combination of paste and stencil are not considered together. Both materials may have a very smooth surface with lubricative qualities which can aggravate paste skidding.



SEM section of a Nickel plated brass stencil

Stainless Steel

Stainless steel can be used for etched stencil manufacture, and all of the comments about artwork modification in the above paragraphs on the subject of brass apply. One thing to note however, is that brass is a fairly ductile material, and in use will coin to take up any localised board imperfections which will result in an intimate contact between the stencil aperture and the pad surface. In the case of stainless steel, however, this is not the case, and as such it is a very unforgiving material when used in a printer that has even the slightest of coplanarity problems.

Basically coining is the gradual distortion during use of the metal stencil. It will take up any uneven surfaces of the board and printer support table or tooling and leave a permanent mark. Examples of coining are:

- The edge of the board when not supported by an edge plate
- Any large cut outs in the board or on multi panels
- Tooling pins protruding through tooling holes
- Uneven plating or domed tin lead on large circuit ground planes

It is fair to say, however, if you use too much squeegee pressure you will always get coining on the metal stencil. There are some company technicians in high volume applications like their coined stencil so much that it is sent back for occasional remounting if damage occurs to the mesh or frame. The perfect coining to the particular board design provides a better gasket to the board surface during printing. It is effectively worn into the design.

SEM section of Stainless steel etched stencil

Copper

Copper foil has been used for stencil production in the Far East and is normally always plated with nickel to provide a harder working surface equal to or harder than brass. It is available in a wide range of stock sizes. Copper has been used due to ease of etching fine apertures. It is, however, easily damaged during handling at the stencil manufacturing stage and in assembly and should be avoided. Its use is uncommon in Europe and the United States.

Molybdenum

Molybdenum often referred to as just "Moly" due to the difficulty of the pronunciation is a further choice in the base materials for stencils. The metal has been popular mainly in United States with very little interest in Europe. It is fairly easy to etch, producing very well defined apertures for fine pitch printing. The main concern in the use of Moly from the stencil supplier is the etching process chemistry. The process employs the use of etching solution which raises some further health and safety issues unique to the process. It is also an expensive process with limited availability of the required stock materials when compared to other base sheet materials.

Advantages of Chemical Etching:

Provided the correct supplier is selected the etching process can provide high quality and a very fast turn round. Not all etched stencil suppliers have the large knife edge in the center of the aperture as depicted in many trade adverts. Step etch capability is available from most suppliers in one facility and there are a wide range of materials which can be etched for stencil manufacture.

Disadvantages of Chemical Etching:

There are suppliers who are not very good at stencil etching, leaving over sized apertures or large knife edges in apertures. The thickness of the base material is a limitation to etching process however stencils are becoming thinner rather than thicker.

The apertures are affected by environmental changes in the photographic process. The repeatability between stencil is not as good as with laser cutting.

<u>Laser Cut Stainless Steel</u>
It is also possible to obtain laser cut stainless steel stencils although the cost of such a stencils is generally high, typically being two times the cost of the equivalent brass etched stencil. The stencil is cut using a YAG laser which literally cuts through the stencil material with a spot size of between 0.002-0.003".

The advantage is that the laser system works directly from the CAD disc without any filmwork stage in between. This means that a disc with a Gerber file can drive the laser system directly. This is not necessarily such an advantage as it may first seem, since circuit boards are, of course, produced from filmwork. The tolerances of both need to be considered when evaluating the printing process. No stencil can overcome a poor quality or dimensionally variable circuit board material.

The advantage of the laser system does is that, since the control system is software based, it is very easy to locally modify pad lengths by sending a marked up drawing along with the disc to the manufacturer. Another advantage to the system is that no etching allowances or artwork modifications need to be made. Also the side walls of a laser cut stencil are perpendicular to the face of the stencil, unlike the side walls of an etched stencil which may be hour glass shaped, and can tend to retain solder paste in the stencil. It is, of course, possible to modify the shape of the finished aperture as is now done to aid paste release. The apertures are now formed with a larger aperture on the base of the stencil to aid paste release just like wet sand in a child's bucket at the seaside. This is generally referred to as trapezoidal apertures.

Combination stencils may be used for fine pitch designs with a reduction in the total stencil price by combining laser cut apertures and etched stencils. This will depend on the way in which the stencil is costed. It is more common to have a combination of laser and etch when a step stencil is required.

<u>Advantages:</u>
Aperture size is very repeatable and controlled from aperture to aperture. The positional accuracy of the apertures and the apertures themselves are very accurate due to the CNC control during cutting. Taper of the aperture can be adjusted along with no practical size limitations. It is also considered to be more environmentally acceptable.

<u>Disadvantages:</u>
A chemical process will be required if any step down sections are needed in a laser cut mask. Some of the laser systems currently used can produce different quality apertures and affect the aperture wall. The laser is affected by the choice of basic material just like the etching process. The laser cut stencil is more expensive than an etched mask.

Electro Deposition Stencils
It is becoming more common to manufacture a stencil by plating up metal as opposed to etching it out. This is known as electro deposition or electroforming and is becoming a popular choice in the industry both in Europe and the United States. One major supplier of stencils with a captive facility offering all the current technology choices produces ?% etch, ?% etch and nickel plate, ?% electroform and ? laser cut. The cost of such a stencil lays part way between the cost of a laser cut and an etched stencil.

Electroforming technology has been used in the industry for many other applications before it was used for stencils. Application like shaver foils, diaphragms and encoder disks were some of the first using the chemical forming. Normally a stainless steel sheet is used as a base to plate which is coated with a dry film photo resist, commonly used in the printed board industry for defining copper circuitry. The resist is approximately 0.004" thick and is hot roll laminated on to the metal support sheet. Some application for electro forming use a liquid coating process which may reduce the cost of the process but the film will have a defined thickness.

The photographic master artwork is then placed on to the surface of the dry film and exposed to ultra violet light which polymerises the resist coating. The artwork consists of a completely black surface, normally referred to as negative artwork with only clear areas corresponding to the required apertures in the final stencil pattern. During exposure the ultra violet light causes the film to become hard and resistant to chemical development.

After the film has been developed the only areas of resist remaining on the steel plate are the resist shapes corresponding to the apertures and their final locations. The plate is then placed in a nickel plating solution until the required stencil thickness is achieved on the plate. The resist can then be striped and the mask removed from the support plate. The stencil then resembles a standard etched foil prior to mounting in a frame.

SEM section of electroformed stencil aperture

As the scanning electron microscope photograph shows there is a characteristic swirl shape to the aperture wall on all electroformed stencil which is caused by the surface of the resist. After the imaging and development of the resist a pattern is left in the surface which is transposed on to the stencil wall. The pattern is left on the stencil aperture wall because the resist is used as a mould for nickel formation. It is also possible to produce multi level stencils using electroform technology in the same way that a two stage etching or laser cut and etching can do for printing different thickness of paste on the board surface.

For some applications where a thick electroformed stencil is required laser cutting can be used after the nickel forming operation introducing yet another choice to the process engineer.

The standard thickness of nickel formed stencils are 100, 125 and 150 microns with a hardness of 200 or 650HV.

Advantages:
Excellent dimensional accuracy and side walls with the possibility of tapered aperture control with any shape of aperture. Almost any thickness of stencil can be produced and the material properties can be changed with the plating bath chemistry.

Disadvantages:
A limited number of suppliers currently provide this service but this changing fast. Any defects in the aperture or the wall are due to photographic process or the mask material being used for the stencil. The cost of the stencil is more expensive than etched but equal to laser cut.

Electropolishing
Basically the electrolytic polishing process is used to overcome uneven surfaces after etching or laser cutting. The need for this process stage is very dependent on the ability of the stencil supplier to control his process in the first place. The process removes some material from the surface providing a smoother wall surface. It does not, however, overcome poor manufacture. It is a common process specified in the USA but not very common in Europe.

Pocket/Recessed Stencils
Although unusual it may be necessary to print paste on to a board which already has some known obstructions on the print surfaces. This requires that cavities are etched into the base of the stencil to allow projections on the board to be present without contacting the stencil during printing. An example of this may be the use of a thick stencil employed for solder paste printing during through hole or intrusive soldering. The original printing process for surface mount parts may be supplemented with an additional print on to the surface of the holes.

It is, however, more sensible to print the paste into the through holes first, then conduct the normal print for the surface mount parts. On the second print an over size aperture will only be required for the through holes in the second print operation. Experimentation with different stencil designs should be able to eliminate a two stencil operation and also obtain effective through hole joints as the example below. This approach has been successful in many facilities in achieving satisfactory solder joints.

SOLDER PASTE

Traditional solder paste can be used effectively for this soldering process provided it meets the requirements of the pitch being printed. Probably the key factors in terms or SDSRS is the tack strength of the paste and the slump resistance but this is no different than traditional requirements. The paste tack strength will be less significant if the UV glue can provide a reasonable cure in a very short period prior to turning the board over.

The tack strength of the paste becomes an issue if the adhesive is not suitable for low temperature or UV curing. It is normally the case that a good paste will hold a component to the board for a couple of hours if suspended upside down. In one experiment the author has suspended a PLCC68 in a printed paste deposit 0.006 thick and run out of video tape waiting for the part to fall from the board. However, as it is heated, paste tends to loose its tack and cannot be solely relied upon to hold the part in place. Reflow experiments have shown that SOIC packages can be retained with uncured glue but PLCC40 packages will drop off.

One of the subtle issues related to this process which can be observed when the solderability is questionable is the lifting effect on the leads, or drop depending on which way you look at. During normal reflow soldering, if leads are slow to wet, there is a tendency for the component and leads to lift floating on the surface of the liquid solder. This would certainly occur if the adhesive was not cured.

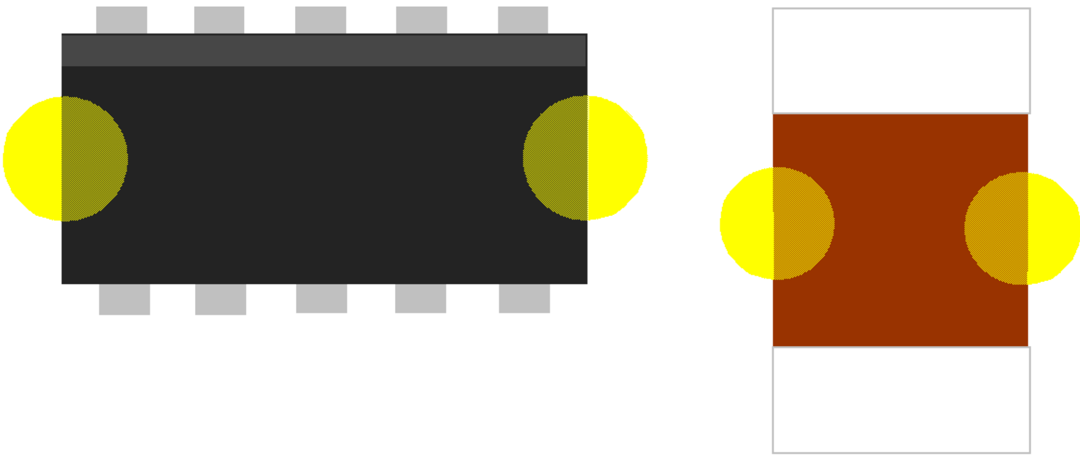
The solder paste must not slump during initial process stages prior to reflow and must not slump during reflow. If the paste selected does slump it will cause the traditional problems of shorts on the second side but it may also mix with the adhesive.

ADHESIVE

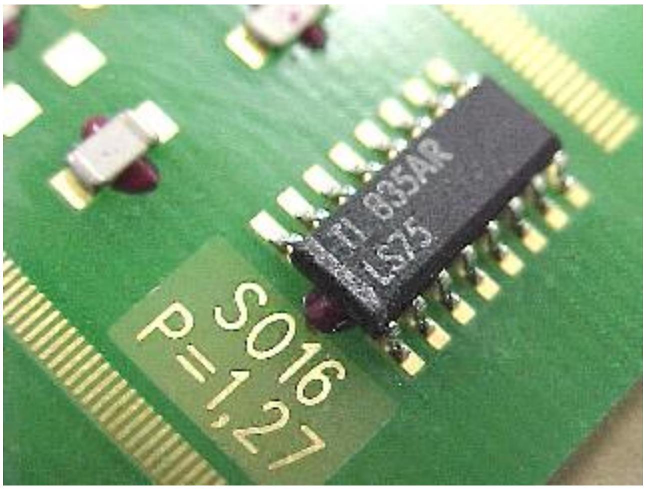
The use of adhesive is generally well understood for surface mount applications but the process of SDSRS puts further burdens on the glue. The material ideally could be cured by a low heat source as it exits the placement process or via UV radiation. The heating process would increase the possibility of paste slump and would not be the preferred process. A UV source, if it could cure the adhesive quickly, would be the preferred method provided it did not in turn generate heat. That would depend on the UV rage, intensity and the formulation of the material.

Acrylic adhesives are UV or heat cured and the initial cure is by exposure to a light source in the long wavelength range. This exposure to the light initiates the material reaction to provide the glue's initial cure strength. Provided this is sufficient to hold the parts in place through board turnover and the first part of the reflow cycle then complete curing will occur during reflow. Care needs to be taken during material evaluation as some adhesive materials do tend to void or outgass if cured too quickly.

The use of UV lamps for curing may be new to some engineers so it is important to work with the material and equipment suppliers to understand the process and the need for process control. UV source can change in intensity over time as the light source decays. Extraction control and the build up of heat in the chamber should be monitored to reduce the possibility of heating the board.



Examples of two dot patterns used for SOIC and chip components when a UV adhesive is being used



Practical examples of adhesive deposits being used on components during testing

General Adhesive Evaluation

The choice of adhesives for effective manufacture is now a little more straight forward for the engineer to assess. The choice of materials for mounting components is still quite wide if we consider the total range of adhesives available, but in most cases many of the available formulations can be discarded due to current production requirements.

Adhesive Requirements

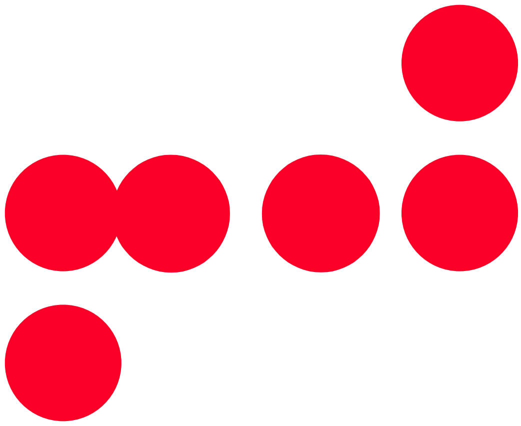
The following are some of the adhesive characteristics which need to be evaluated for any adhesive application:

- Electrical properties
- Dispensing
- Adhesive strength
- Slump/bleeding
- Material compatibility
- Cure time/temperature
- Adhesive life
- Outgassing

Today the use of one part systems is preferred due to the simplicity of supply and use. The one part systems come in epoxies, acrylates and cyanoacrylates. Whichever material is selected process control on the assembly line is still a very important factor. The two methods for in process inspection are shear strength and degree of spread, both can be evaluated using SPC (Statistical Process Control). It is of course necessary to first evaluate the material, to establish control limits for production. If this is not done correctly SPC will not be beneficial. Control limits can easily be achieved with a simple process capability programme.

Process Checks

The parameters which may be monitored are the size of the dot or degree of shear force required to remove components after glue curing. For in process inspection it is also effective to provide a glue test pattern, the inspection staff can visually examine for stringing or the size of dot. Measurements may be taken of the diameter of the test dots with reference to limits which have already been defined using statistical techniques. A test pattern may also be used prior to adhesive cure for bond strength testing with scrap components. An alternative method used by some manufacturers is an active test pattern which may be used as a visual test for accuracy or glue slump.



The adhesive dot pattern above is used to monitor dispense control. The three dot pattern can be used to detect changes in adhesive volume changes

The test pattern is used on glue stations with a two dot pattern needle. The pattern consists of three sets of dots each at right angles to the next, each with a different separation. In the test pattern one dot pattern is touching the next, in the third they are separated. In the case of variation in the process, excessive glue would cause each dot pattern to merge, limited glue would show each dot pattern to be separated. The active test pattern has been successfully run on a number of pick and place systems and again provided simple, but effective quality control.

Shear strength measurement of scrap components mounted on the board prior to cure is a useful test. It can be used to determine correct cure cycle, limited glue dispense and contaminated components or circuit boards. The shear test may be conducted using components placed on the edge of the board or on the scrap area of a multi panel. Adhesive is dispensed as part of the glue pattern with components placed in position specifically for this test. The board is put through the recommended curing operation and measurements are taken for shear force after the board has cooled.

The evaluation results obtained may be affected by the solder resist type, the copper preservative coatings on the PCB, the release compounds on the components, quantity of adhesive, component standoff height and the adhesive quality. Inspection standards are always necessary in production if test patterns are not appropriate. These must be based on the capability of the process and used as a reference for inspection as well as initial training programs. A reference sheet can be extremely useful when quality engineering are collecting SPC data; this guarantees that everyone is working to a common standard and the resulting statistical data is of some value. Inspection standards posters are available from the SMART Group.

Adhesive Dispense

The dispensing speeds for adhesive have increased over the last few years to over 100,000 dots per hour traditionally they were all around the 40-60,000 dots per hour. The speed achievable was partly due to the limits of the adhesives available in the industry and the actual processes used to dispense, time, pressure and needle type. Although not as fast as screen printing they do provide a greater flexibility in terms of volume control and possible changes to the dispense pattern.

There are two types of dispensing technology which are recognised in the industry for adhesive and solder paste they are:

Time Pressure Dispense

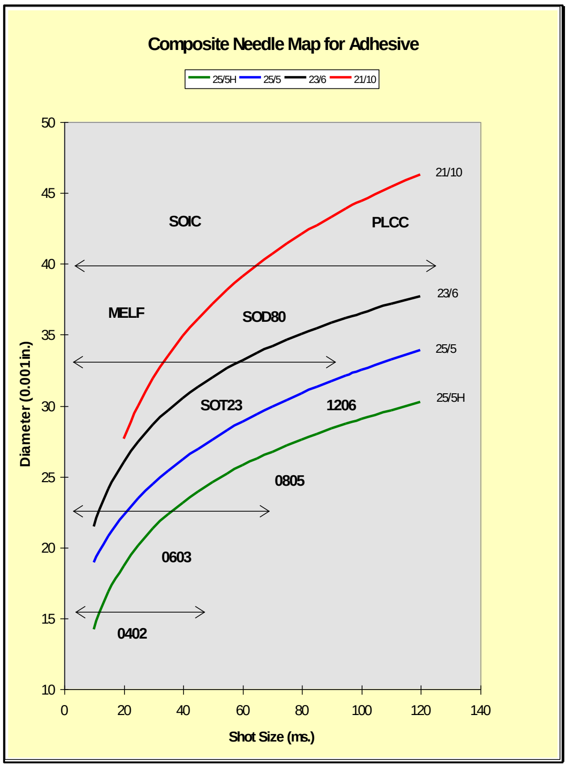
A dot of material is dispensed by applying pressure to a filled syringe for a fixed time. The material volume is controlled by the time the pressure is applied, the pressure and the needle design. This system is the most common in the industry purely as it was the first and used on many production lines. It has been found to less effective as the dot size required has decreased leading to the greater acceptance of the rotary dispense technique.

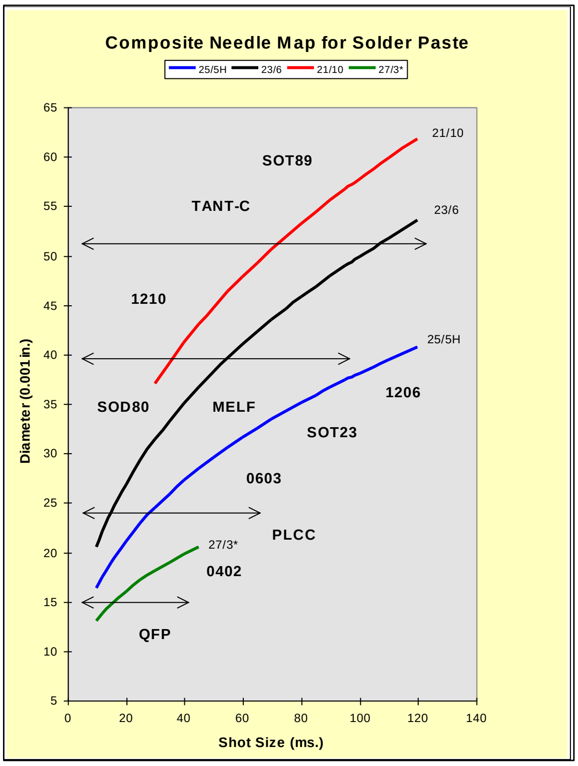
Positive Rotary Dispense

A rotary pump dispense system incorporates a syringe filled with the medium to be dispensed under a constant pressure. The medium then flows out of the syringe into a archimedes screw which is driven by a DC motor. The paste or adhesive then flows out of a needle on to the surface of the board. The dispense volume is controlled by the capacity of the screw, rotation speed, rotation time and needle design.

Solder Paste Dispensing

Over the last few years there has been a greater interest in solder paste dispensing for special applications and for low volume high mix operations. This has been due to the cost and changeover time for stencils. It was also influenced by the space taken up to store framed stencils. Now with the use of frameless foils the issues over storage have been virtually overcome.





These two charts (courtesy of Camalot) are typical of the information provided by suppliers of dispensing equipment to provide initial set-up guidelines

The use of new dispensing systems which can simultaneously dispense in line paste and adhesive makes this an attractive option for SDSRS. The main issue is the speed; can it meet the process requirements as it undoubtedly will have the ability to provided a repeatable process.

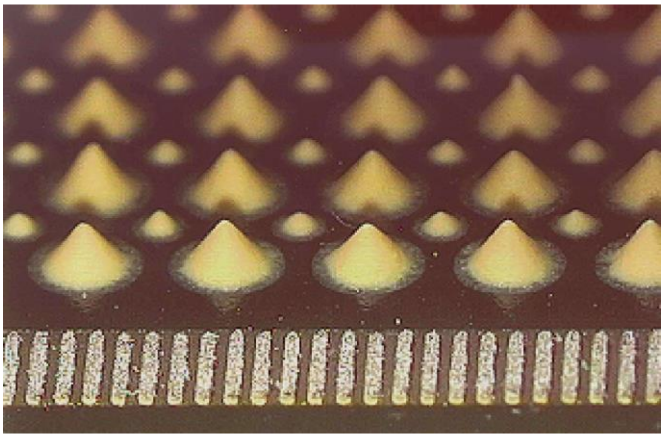
Pin Transfer

Pin transfer, like stencil printing, is very fast but dedicated to a single design which makes it less versatile than dispensing. Dispensing programs can be changed very quickly to modify the volume of material or its position; stencils and pin tooling are far faster than dispensing. There are also a number of other pros and cons for this technique.

Adhesive Printing

It is perfectly possible to stencil print glue onto the surface of the board even after solder paste printing albeit a process complication. Basically the stencil for printing adhesive is thicker than for solder paste and can be produced by multiple etching processes, electroforming or laser cut and etch. To achieve a second print on to the same surface, the base of the stencil is recessed around the printed paste deposit. This prevents the paste contacting the stencil but does allow adhesive to be deposited under the components.

The same technique has also been used by DEK to allow adhesive to be deposited on to the surface of the board after conventional component insertion. In this case the conventional through hole components have been automatically inserted, the leads cut and clinched. The stencil is etched out in the areas around the through hole leads.



This photograph shows solder paste and adhesive deposits which have both been printed using two different DEK stencil designs

Printing adhesive can be a viable process and can achieve a much faster speed than any dispensing process and can compete with pin dispensing in terms of speed. Both material suppliers and stencil manufacturers have developed new materials and processes to meet the challenge.

Component Standoff Height

When evaluating which process to use you also need to consider the standoff height of the components from the surface of the board. An adhesive needs to bridge the gap between the base of the board and the component and have the largest surface contact without affecting any other parameter.

Component	Standoff Height
1206	<0.001”
SOT23	<0.005”
SOT89	<0.001”
SOIC	0.010 - 0.012”
SOIC	0.010 - 0.012”
TSOP	0.006 - 0.010”
QFP	0.012 - 0.018”

Adhesive Curing

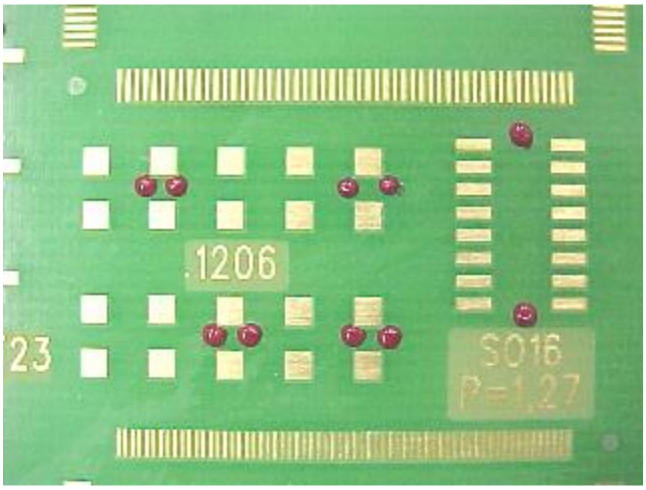
To cure the adhesive the material must be exposed to the UV light source during passage along the conveyor. Realistically the distance between the placement machine and the turnover section will be between 1-1.5m.

Both conveyor suppliers and reflow companies do have experience of UV curing systems which may be added to a conveyor. Alternatively a supplier like Natgraph has experience with UV sources and producing UV bridges used in the printed board industry for imaging and curing resists. The example below shows a UV bridge mounted over the conveyor and the light and control system.

The example UV bridge (courtesy of Natgraph) is mainly used in the PCB industry but the same concept can be applied to a conveyor system for use in the assembly market

The bridge allows boards passing under it to be exposed to UV but covers and prevents any direct light exposure. A key aspect of the exposure is that curing or cure initiation is obtained without any surface heating of the board. Heating the surface of the board could produce slumping of the solder paste. The finer the pitch the higher the possibility of the slump causing shorts after final reflow.

Trials have been run using one adhesive to test the degree of cure that can be obtained in a relatively short space in time. The trials consisted of dispensing dots of adhesive for a number of surface mount components then curing the adhesive using a UV light source. After curing the components were tested by applying a shear force to the side of the parts using a digital gauge.



The example above shows a close up view of the dots included for the initial trial for UV curing for SDSRS

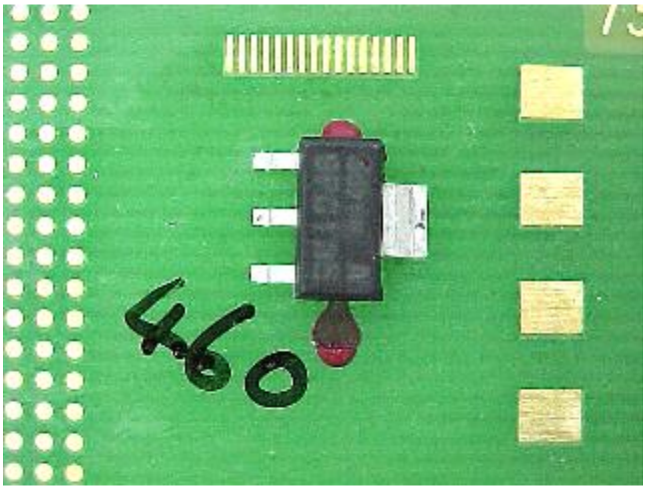
The following forces were obtained during the adhesive trials:

Component	Average Dot Size	Maximum Shear Force (g)
1206	0.025”	125
SOT23	0.030”	95
SOT89	0.025”	140
SOIC	0.040”	522
QFP	0.030	385

The results were variable mostly due to the manual application of the adhesive. This allowed variable amounts of material and position of the glue which prevented effective UV exposure and curing. Shear test measurements in each case were applied parallel with the direction of the body of the device which favoured some part types and their dispense patterns.

Typically the minimum force which is used by many companies for in process control is 500g force to displace the parts. There are other companies who have adopted torque testing with a circular movement to reduce the effect of the dispense pattern.

Curing was conducted using a UV source 365nm wave length with a typical intensity of 100mW/cm^2 mounted above the board. The exposure time for each of the samples was 10 seconds. In all cases where the adhesive was outside the edge of the body the material had started curing but there was evidence that the material was still wet beneath the body of the parts.



Example of SOT89 after testing the cure strength from a 10 second UV exposure

COMPONENT PLACEMENT

Just like the printing process the tooling to support the board is the main issue to consider with this process. Whatever tooling is used it must not allow the board to flex, contact components or subject the board to vibration.

Many placement systems or lines do have integrated dispensing systems for adhesive or have a dispense module like those supplied by Fuji, Panasonic, Siemens etc. Their capability and their repeatability for dot size and positional accuracy between solder paste prints needs to be reviewed particularly for the small passive devices.

The component placement force must be fully adjustable to prevent undue force during contact of the component with the board. Ideally the termination of the surface mount parts must be fully place into the paste deposit. This allows the full tack force of the paste and the adhesive to help the adhesion of the part on the base of the board during transport and reflow.

A further issue which may need to be examined is the effect of placing components into paste and adhesive particularly on finer pitch parts. As the part is placed the body of the component would contact the adhesive deposit before the leads contacted the paste. Any misplacement due to this action may affect the accuracy of lead placement and with it the potential for solder shorts.

Example of high speed component placement systems that provide the flexibility for double sided and SDSRS (courtesy of Siemens)

REFLOW SOLDERING

During reflow soldering a solder joint is formed between a component termination and the pad on the surface of the printed board. With SDSRS it may now reflow both sides of the board, top and bottom simultaneously. Normally the termination on a component is tin/lead plated which provides an easily solderable. The solder paste is produced with a 63/37 tin/lead alloy, which is the most popular alloy used in all soft soldering electronic applications.

There may of course be changes in the next few years away from traditional alloys with the reduction of lead and the move to lead free solders. Europe is leading the way with the proposed ban. The two alloys which seem to be favoured at present are the tin/silver or the tin/copper materials. Tin (Sn) 96.5%/ Silver (Ag) 3.5 % with a liquidus temperature of 221°C or Tin (Sn) 99.3% and copper (Cu) 0.7 % at 216°C.

Most chip terminations are a metallised silver palladium surface with a nickel layer and tin/lead as the final protective coating. A leaded part will either be copper with a tin/lead coating or kovar with a solder coating. In recent year some suppliers have moved away from tin/lead to a solderable palladium finish for the leaded parts. This was due to the problems of lead forming and cutting, jamming the tooling with the soft tin/lead coating. In the case of the printed board the most specified surface finish is still tin/lead applied by solder levelling. Gold over nickel, copper (OSP), palladium, and silver have all recently taken some market share. Alternative solderable coatings have been introduced to resolve the problem of uneven tin/lead surface coatings seen with traditional solder levelling.

To reflow a traditional board assembly solder paste is applied to each of the pads on which a component is to be soldered. Components are then placed into the paste surface to assist the wetting of the side terminations. The paste provides some adhesive qualities during transportation of the board to prevent component loss prior to soldering. As the paste is reflowed it changes into a liquid state at 183°C for a 63/37 alloy, a reaction takes place between the tin and the surfaces being soldered. The tin from the paste and the copper or nickel on the lead or circuit pads form another alloy referred to as an intermetallic. This new alloy is the joint which forms during any soldering operation and must form if a reliable joint is to be produced. Tin reacts with most materials to form a bond, copper dissolves far quicker than nickel.

How effective this reaction is will depend on the solderability of the surfaces being joined. To make this process feasible a fluxing agent is incorporated into a paste to clean the lead, pad and the solder particles that make up the paste. When the solder is in a liquid state there should be nothing to prevent the soldering reaction taking place. It is possible to use low activity or high activity fluxing agents in solder paste formulations but that will depend on whether cleaning is to be conducted after assembly.

Controlling the temperature, pre-heat, atmosphere and solderability of all the materials being joined is the job of the process engineer. Normally profiling is conducted using one of the market leading profilers produced by either Datapaq, ECD or Kick using temporary or removable temperature probes. The mounting of cables and the use of profiling systems is featured on the SMART Group Reflow Soldering CD-ROM.

Determining a profile is normally a single side operation, some companies also need to monitor the bottom side in an attempt to reduce second side reflow. With SDSRS the requirements are somewhat different, the process needs both sides to reflow at the same time.

Reflow Soldering System

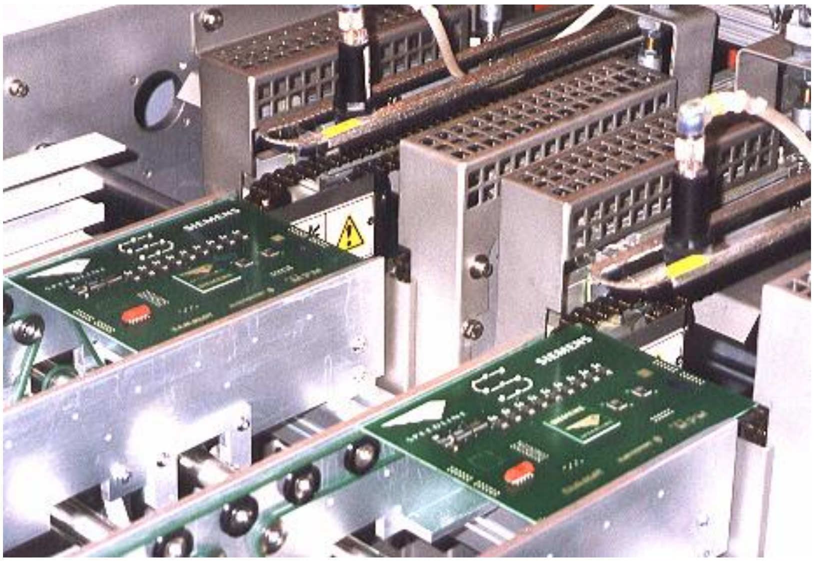
Conveyors

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

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

The latest industry trend is for two pin conveyors to be incorporated into a single reflow oven . The dual lane process is particularly interesting for companies in high volume with limited factory space or a limited mixture of board widths. It also lends itself to companies producing one board width in volume and a variety of other widths in smaller volumes. The smaller volumes can be processed down one line with the high volume down the other fixed lane. The only process parameter which can be varied is the speed of the conveyor as it is not possible to change the zone setting for each conveyor.

Fortunately with this new technology, standards have already been defined so that reflow systems are compatible with placement systems and outfeed conveyors.



Example of a two lane conveyor system feeding a dual lane reflow system (courtesy of Electrovert)

Two issues need to be considered here for SDSRS: the relative position of the conveyor/heaters above and below the board and if the conveyor or the panels can be moved to alter the heat input. The use of a board support system for thin panels can, like pin conveyors, take heat away from a section of the board. If the SDSRS process is to be used special consideration will be required for support to avoid problems.

Pin Transport

Pin transfer is becoming the most popular conveyor as it is more versatile for both in-line and batch production. Using a linked mesh system would be fine for batch production but not ideal for in-line operations.

The pin transfer system provides the best solution for in-line production. The linked chain with 5mm pins is the most common, the board just sits on the surface of the pins for transportation through the process.

An alternative clamping system is offered by reflow suppliers like BTU and Soltec. In this case the board enters the reflow oven and the edges are clamped on each side limiting warp and twist of the board as it passes through the reflow process. Thin boards of less than 1mm or standard 1.6mm boards which have poor copper balance can benefit from this type of conveyor. The boards expansion in the x & y direction is not restricted. The clamping technique utilises the normal 5mm clearance designed into the edge of all well designed boards. The clamping design also reduces sag of the board during soldering.

Some suppliers of reflow ovens have noted that the conveyor section heat sinks some heat from the board just like a board placed into a reflow soldering pallet drops the temperature by 5-10^{o}C. To overcome or compensate for this, heaters are built into the conveyor section in specific locations to allow process engineer to compensate.

Linked Chain

The mesh belt has been used for many years in the industry for adhesive cure and reflow applications. It has also been used for reflow soldering in the printed board industry when fusing tin/lead. The belt provided support for the whole board as it passes through the process and reduces the possibility of warpage. The weight of the board holds it flat as it passes over the glass transition temperature of the base laminate.

The mesh is formed in such a way to provide a standoff which reduces the thermal loss from the board to the conveyor. They also assist in allowing the air to move freely around the base of the board.

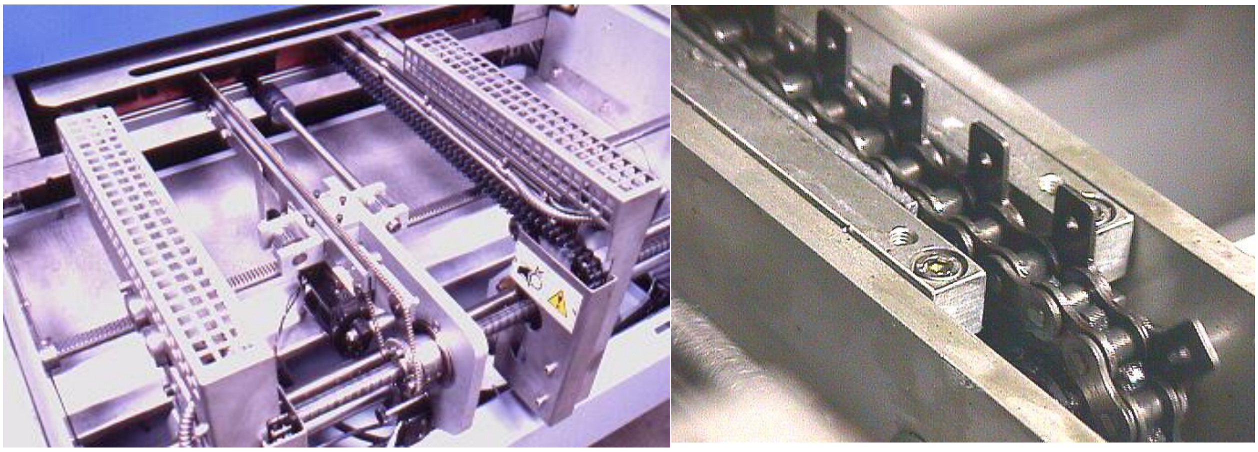
Mesh belts are used for soldering single sided surface mount boards and should not be used for double sided surface mount products. The board can be supported off the belt with no contact between components on the base of the board. Realistically they are just not suitable for SDSRS.

Centre Board Support

With the greater use of thin boards below 1mm and panel widths above 120mm there is a tendency for boards to sag during reflow soldering if used on pin conveyors. For many years a centre board support has been used on wave soldering. Either a wire or solid knife edge has been employed to support the board in the centre position during wave contact. Similar techniques are also used during reflow soldering and available from most suppliers. The systems are either wire, vertical pins on a linked chain or simply a linked chain. The ideal method is one which takes up the minimum contact space on the base of the board.

These support mechanisms can be dropped when not required on double sided assemblies with a drop distance of 25-50mm below the board. They are also capable of being moved into different positions like the conveyor itself. The use of board supports when soldering both sides of the board does rely on the design engineer leaving a support area in the centre section of the board. Normally this is a clear line 3-4 mm wide. The use of centre board supports tends to be used on the first side of a double sided assembly. There does tend to be a rigidising effect when the first side is already populated and soldered. This is true if the solder joints do not move into a liquid state during second side reflow. SDSRS has different support requirements.

Keeping the board flat is important for final product acceptance but also for future process stages. For example if the board is not flat, printing of solder paste and placement of components on the second side will be a problem. A more recent problem with warped boards has been the increased use of flying probe test systems as opposed to bed of nails fixtures. The flying probes do not have the same capability of overcoming uneven boards.



Two examples of centre board supports incorporated into a reflow oven to support the board during processing. The centre support prevents sagging during reflow and reduces the possibility of warped board

Machine Zones

The number of zones in an oven is generally dependent on the throughput requirement of the process. Selecting a 7 zone system over a 10 zone unit will provide the engineer with more control of the process, but at the speed that is required for the number of boards that needs to be processed per hour.

An oven normally includes zones which are often referred to as pre heat, soak, reflow and cooling just like the stages of a temperature profile. There may be one or two pre heat, two soak and one reflow with a forced air cooling section. More substantial systems with seven zones will incorporate dedicated cooling zones with forced cooling. Remember it is not just the number of zones it is the effective length of these zones which is important during reflow.

In the case of SDSRS the bottom and top heaters can be independently adjusted to obtain the same surface profile on both sides of the board. This may not be possible on a standard oven due to the configuration in the reflow zone. Both the oven and profiling software may not be compatible with this type of process. In the case of profile prediction software it cannot use the bottom heater setting higher than the top. The prediction algorithm in the software may not be compatible. Machine software may only have a single points for sensing oven temperature or over heating.

Zone Sections

Pre Heat

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

During pre-heat any volatile material in the paste can be driven off as well as start the initial cleaning action of the flux contained in the paste. Some materials do not activate until they reach substantially higher temperatures. During pre-heat every effort is made not to create too much of a temperature differential on the board surface as it may be difficult for this to be overcome during the soak period. A temperature rise in the initial pre-heat zone can be between 80-150°C.

Soak
The soak period allows all temperatures on the surface of the board to normalise. It is inevitable that during pre-heat some termination areas on the surface of the board will heat up more quickly than others. Today, with the greater sophistication of reflow technology, the surface of the board can see limited temperature variations but at the component termination interface the temperatures will vary. As a guide under a plastic ball grid array the differential temperature can be 10-15°C. Under a through hole connector body the temperature difference can be 15-20°C lower than the surface of the circuit board. At the end of the soak the temperature can be between 150-170°C depending on the product and the type of paste materials being used.

Reflow
The reflow zone is where the board temperature moves over the liquid temperature of the solder paste. The paste is normally 63/37 or 62/2/36, the first reflowing at 183°C and the second silver loaded alloy becoming a liquid at 179°C.

The aim of the reflow zone is to allow reflow in the shortest time possible to produce reliable joints normally within 30 seconds. This is normally dependent on the temperature differential of the board entering the reflow one. If the assembly temperature is less than say 140°C and there is a 20°C differential, all joints will not make reflow temperature.

Normally this zone is top heater dominant so further control to obtain a stable top and bottom side process temperature could be a problem for SDSRS. This must be achieved during each stage of the process.

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

Some engineers suggest that you cannot conduct reflow reliably in a reflow oven of less than five zones but that is not true. Provided the board assembly can be reflowed with the minimal differential temperature across the board, not subject the components to a thermal shock outside their specification and have the joints in a liquid state for a period of less than 30 seconds, any number of zones is acceptable.

Single zone reflow processes are sold on tabletop batch ovens. The new tower reflow equipment is effectively a single zone system. Two and three zone ovens have demonstrated their capability of providing reliable reflow on double sided assemblies, BGA and Pin In Hole reflow, albeit at a reduced throughput speed to larger ovens. The limited number of zones does reduce the flexibility to the engineer during profiling of a complex board design.

Machine Extraction

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

Normally extraction ports are positioned during the pre-heat zones and the reflow or just before it enters the final soak zone. Initially vapours will be seen from the paste in the pre-heat phase. Additional fumes will be seen just before reflow of the paste and any material coming from the laminate or the solder mask. Some laminates and solder resists will give off fumes between 80-140^{o}C.

Many suppliers have experienced problems with flux fumes affecting the cooling process. With the use of SDSRS the fumes will increase so care needs to taken when looking at the quantity and the capability of the oven to deal with it.

There may be specific issues like the use of laminates that give off different vapours which can condense out and drip down from exhaust ports. An example of this is the phenolic resin associated with paper based substrates when used during wave soldering in nitrogen. The phenolic does cause a problem for the extraction.

Cooling

Cooling systems vary from the basic fans which blows ambient air on to the surface of the board to assist cooling. There are high volume convection systems which increase the cooling rate or the nitrogen systems which cool in an inert atmosphere. There are a number of practical issues why a board assembly may need to be cooled before it exits the oven.

Handling manually or automatically may be an issue due to the residual heat in the board. Operators would need to wear gloves if manual unloading were used. The solder must not be in a liquid state as the board exits the oven and moves onto a conveyor this could cause part displacement.

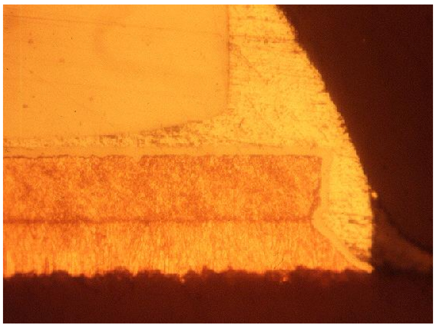
If the board is being fed directly into a second paste printing or dispensing operation a warm board will affect the paste possibly causing it to slump.

Cooling the solder quickly after reflow can reduce the intermetallic at the joint interface. The thicker the intermetallic the weaker this area is reported to be. The period of reflow today should ideally be less than 30 seconds in a liquid state. With the possible introduction of lead free products based on current thinking the temperature of the ovens will increase. This increase for lead free products will be between 15-20°C which will of course put greater strains on the cooling process

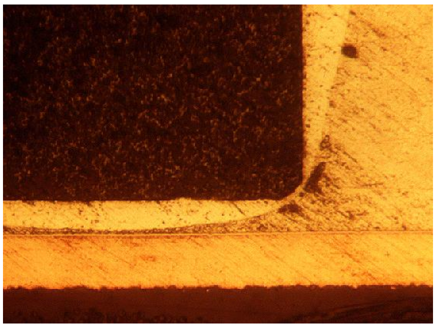
Maintaining the board at high temperatures for a long period of time will cause the solderability of the pads and terminations to be degraded. Cooling the surface quickly to below 50°C has been shown to be beneficial particularly for copper boards coated with Organic Solderable Protectors (OSP).

Nitrogen

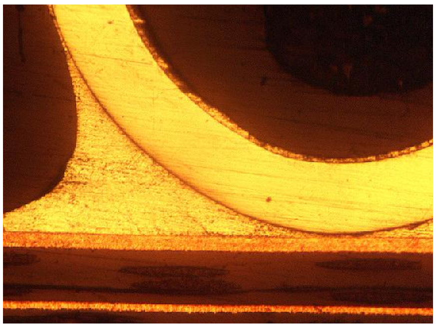
The use of nitrogen has become popular over the last few years particularly when using low residue solder pastes and copper OSP circuit boards. The nitrogen is being used to displace the oxygen and open up the process window during reflow soldering. Oxygen levels in reflow soldering at or below 100ppm have been shown to improve soldering performance and prove economically viable if copper boards are being used. The use of copper boards coated with an Organic Solderable Protector ("OSP ") are cheaper than gold or tin/lead and the saving can offset the use of nitrogen. Hence the quality and process improvements obtained with the use of nitrogen are free, or nearly free.



Microsection from a nickel gold board after SDSRS



Microsection from a nickel gold board after SDSRS



Microsection from a nickel gold board after SDSRS

The above microsection were taken from side one board samples reflowed with adhesive holding the components in place during SDSRS. Microsection one and two were reflowed on nickel gold board the third sample had an OSP finish.

Flux Only Reflow
A process that has been reported to work effectively for double sided assembly is flux only reflow. The boards being assembled have been specified with plated tin/lead which allows the printed board supplier and customer to define how much solder is present on the pads. The flux allows the cleaning operation on the component terminations and pads.

This process has been used for high reliability circuits in the past for boards containing Leadless Ceramic Chip Carriers (LCCC) which are one of the heaviest components in a surface mount format. The process of assembly involved placing tacky or concentrated flux on the pad surfaces and then placing the LCCC parts. The flux concentrate was produced by allowing the solvent in a high solids flux to evaporate off. The boards were then turned over and the process repeated prior to reflow. One company used a stencil process to apply the flux to the pads rather than dispensing.

The reflow process in each case was conducted by vapour phase reflow obtaining a simultaneous reflow on both sides of the board. The flux protects the tin/lead pad surface as the coating moves into a liquid phase forming the joints. With the recent interest in VP, soldering companies like Seho may be able to advice users on the possibilities of conducting SDSRS with condensation soldering.

Note: The author has not conducted this experiment and although having discussed with some industry experts remains sceptical due to the weight of the LCCC. The use of a tack medium would have moved under the device by capillary action providing a degree of adhesion and may have allowed this process to be successful.

Profiling
Profiling a printed board is essential to guarantee that peak temperatures for components are not being compromised and that the soldering process is in line with paste suppliers' recommendations.

A temperature profile is simply a plot of temperature rise against time which can then be related to a particular printed board design and oven. A profile should always be conducted on a loaded circuit pattern to guarantee accurate information.

A procedure for profiling a new printed circuit board should provide a basic guide to successful reflow soldering. A simple procedure is included on the SMART Group Reflow CD-ROM.

CONVEYORISATION

The conveyor design guidelines will not change from a conventional product. Clearance down two sides of the board or panel are required to support the board. This will vary between designs but a clear area would normally be 4-5mm. This edge clearance is required for tape, pin transportation and location in load/unload cassettes. Vibration during transportation could be problem particularly the acceleration and deceleration of the board during transportation. The effect of the conveyor movement will be eliminated if the adhesive is cured. The normal speed of a conveyor is quoted as 18m/min by suppliers like PAF.

PCB turnover unit which would be used after placement prior to second side assembly. This section could be used to house the UV source eliminating an additional conveyor section (courtesy of PAF)

Flipover systems where the board is inverted and turning stages are used could potentially be a weak link in the chain. The amount of board movement and the vibration could be more than the paste and adhesive tack bond could stand hence the need for UV curing. Generally speaking the most vibration will be transport between conveyor and machine and then returning to the next section of the conveyor.

When designing the conveyor system thought needs to be given on where to place the UV section. It would need to be placed after the placement stage prior to board turnover. A second UV section can be placed after turnover if the material dictates additional UV exposure prior to entering the second side print stage. The boards should pass through the UV without stopping as this will provide a more consistent process. The boards should not stop prior to any stage as stopping in a UV section increases the possibility of heating the board.

As an alternative to having a dedicated UV stage it could be incorporated into the board turnover unit. The maximum speed for the turnover unit is 12 seconds for a complete cycle. With a slower speed for entry curing could still be conducted prior to board turnover reducing the conveyor space required. The maximum operating temperature for the conveyor section is 50°C which should never be approached during normal use.

FINAL INSPECTION

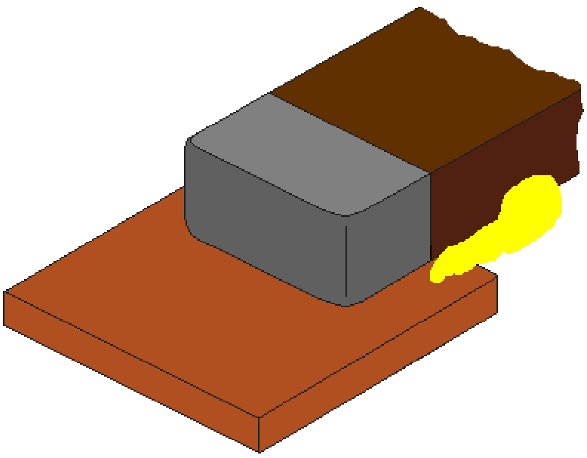
Inspection criteria for SDSRS should not be significantly different to those standards currently being used in industry. IPC documentstend to be the standards most often referred to in contracts and on general training courses. The IPC 610 document covers both surface mount and conventional soldering quality. They do not, however, cover the issues associated with gold or OSP boards or the use of intrusive reflow.

Two issues which may need to be re-examined are adhesive contamination on the surface of pads and the standoff height of component leads after reflow.

Adhesive Contamination

Most standards do not allow adhesive to be present under the component and on the surface of the pads. In reality as long as the minimum soldering standard is achieved this should still be acceptable. The process which causes the glue to be present on the pads needs to be addressed but the parts should not be reworked.

Unfortunately most inspection criteria is for the final solder joints, guidelines should be set for post placement and post soldering. As an example the minimum criteria for adhesive prior to placement may say that adhesive may touch the pad but not be present on the pad surface. After placement the adhesive may be present on the pad provided a minimum solder joint can still be achieved.



In this example there is still the opportunity to solder the joint and achieve the minimum solder standard of 75% coverage even with evidence of adhesive

After soldering, provided the minimum solder joint has been achieved, then no rework is required, however the process should be re-examined for corrective action. These examples show that when using current assembly process rework is often done unnecessarily.

Lead Standoff Height

During reflow soldering lead or component float can occur lifting the part to the crest of the joint. During soldering the solderability of the termination will affect how quickly a joint is formed. If the termination is slow to wet initially the lead will sit on the crest of the paste or solder and eventually pull back into the solder.

The criteria for soldering is that the solder should wet up the side of the pin to half the thickness of the lead. In the case of simultaneous reflow the only thing holding the component is the tack of the glue and the paste or if the adhesive is cured the original height of the termination when cured. That is also dependent on the placement force applied during assembly and whether or not the terminations were forced fully into the paste. This is generally not the case due to the possibility of shoring the paste.

SDSRS PROCESS DEFECTS

Finally let’s consider the type of defects that may occur when using SDSRS and how we may be able to eliminate the possibility of these occurring.

Lost Components

At the outset this is the problem that most engineers will envisage during process introduction. During recent trials lost components have been seen to be an issue but with correct design and selection of parts for first side placement, like DSRS this can be avoided. If parts are missing the assembly would be rejected and another part soldered in place or the product will not work !!!

Lifted Leads

As half the components are soldered upside down it is possible for the component not to be flat to the surface of the board. Components can be off the surface of the board provided that minimum soldering standards are still achieved. The normal requirement is that the solder on J and Gull wing leads reaches up to half the lead thickness. The chip termination’s minimum standard is 25% of the height of the component termination.

Lead lift is unlikely to be a problem with chip parts but can be an issue with leaded parts which may be heavier. If the adhesive is cured prior to going into reflow there should not be an issue with lead of termination lift.

Glue Contamination

The criteria for adhesive joint contamination is tight for traditional surface mount. It does not allow glue to be present on the pad surface which is fine for 1206 and 0805 but when you go below these sizes it may occur. It is far more sensible to have the criteria to stop the process of dispensing but not to cause the boards to be reworked or wiped off. The criteria should be linked to the soldering criteria which often has a minimum criteria of solder around 50-75% of the lead or termination to be acceptable.

Misplaced Terminations

The normal criteria of misplacement is acceptable and should easily be achieved. The normal standard applied in companies is between 50-75% of the termination must be on the pad surface.

Solder Balling

Solder balling should not be more prevalent in SDSRS than in any other process. There may be evidence of solder balls on the surface of the glue after reflow on chip terminations. This is due to the glue surface causing solder balls from the paste to bind to the surface or if the paste slumps, which hopefully will not occur.

This type of balling has been seen in the past when one large contract manufacturer used double sided reflow with adhesive on one side of the board. The reason for the adhesive was to allow a solder joint to be formed on the bottom of the board before insertion of through hole and wave soldering. Reflowing the board first guaranteed a solder joint without solder skips during wave soldering. The contractor was trying to avoid the use of a chip wave and the resulting increase of solder dross.

FURTHER TRIALS

The area which probably needs the most investigation is the adhesive. If a glue can be found or developed to cure at low or ambient temperatures or use a UV curing mechanism that would be ideal. The material would not have to develop a full cure strength but the cure would need to be achieved without slowing the speed of transport between placement and turnover for second side placement.

General trials will need to be undertaken on the amount of vibration which occurs during processing and transportation. This is normally an issue with double sided assembly and may already have been addressed.

Further trials need to be conducted on reflow systems and profiling equipment to obtain the best set of process parameters. Ideally trials on lead free products will also need to be undertaken especially with so much European interest in the use of lead free solders.

Any updates to this initial report will be provided to those companies purchasing the initial copy of the report.

FURTHER REFERENCE MATERIAL

The author has yet to find any other information on SDSRS but would welcome a copy of any papers to list in any future updates of this report.