NPL Report MATC(A)92

An Review of Electronics Materials Deposition Techniques Including Solder Jetting and Relief Printing

Martin Wickham, Ling Zou,
Milos Dusek & Christopher Hunt

April 2002

SENSOR

April 2002

A Review of Electronics Materials Deposition Techniques including Solder Jetting and Relief Printing

by

Martin Wickham, Ling Zou, Milos Dusek and Christopher Hunt

National Physical Laboratory
Teddington, Middlesex, UK, TW11 0LW

ABSTRACT

This report presents a review of two deposition techniques, solder jetting and relief printing, being considered to complement the traditional stencil printing or needle dispensing. The review of solder jetting highlighted a number of potential niche applications for technique . The majority of these are low volume applications which could avail themselves of the direct write capability of the technology, such as application of solder to substrates for selective deposition for DCA applications or via fill. Wafer bumping in low volume for specialist devices such as ASICs, as well as chip scale package bumping, could also benefit from the direct write capabilities of the process.

Relief printing (printing through thick plastic stencils with underside recesses) of adhesives has been successfully demonstrated with the manufacture of simultaneous double-sided reflow (SDSR) soldered assemblies, but some limits to the technology have emerged. To utilise this process, end users may find that solder paste and adhesives deposition will have to be limited to the larger components or coarser pitch components where large stencil apertures can be used. Adhesive printing proved easier than solder paste printing. However, the technique should be eminently suitable for printing of larger area deposits, such as for coarser pitch components, thermal adhesive under perimeter arrays (QFPs), or die-bond adhesives for chip on board assemblies. In these applications, relief printing may prove faster than needle dispensing.

ISSN 1473 2734

National Physical Laboratory
Teddington, Middlesex, UK, TW11 0LW

Approved on behalf of Managing Director, NPL, by Dr C Lea,
Head, Materials Centre

CONTENTS

1. INTRODUCTION

During the 1990s, surface mount assembly became the dominant electronics manufacturing assembly method. From its early days in the 1960s of small flat packs and chip capacitors, the technology has blossomed into a high yield, high density interconnection method enabling the manufacture of a wide range of products, such as mobile phones and notebook computers, which would simply not have been possible with more conventional techniques.

The increase in the manufacture of surface mount assemblies has led to widespread use of the technique of stencil printing, primarily to deposit solder paste. The technique has been developed for electronics manufacture from silkscreen printing used to print everyday objects such as T-shirts. The increased demands of finer pitch surface mount components, has led to the universal adoption of solid metal foils with etched cut-outs (stencils) for printing solder paste. These have been further developed to improve ultra fine pitch printing by using additive techniques (electro-forming) and by using lasers to cut the apertures. Machine technology has improved by use of fiducial alignment, metal squeegee blades rather than deformable rubber, and the latest enclosed print head technologies. Latterly, stencil printing has been making inroads into other areas, such as wave-solder adhesive application. Stencil printing is now commonplace in the electronics manufacturing environment, being used to assemble components from large power transistors through fine pitch quad flat packs (QFPs) at 0.4mm pitch, down to area array packages and flip chip devices. However, if solder deposition is required at finer geometries, stencil printing of solder paste becomes more problematic. In addition, the technique does not lend itself to printing on non-planar substrates.

Needle dispensing is also widely used within the electronic manufacturing environment. Traditionally used as part of placement equipment to deposit adhesives for wave soldering immediately prior to component placement, the technique has been developed and incorporated into significantly faster stand alone machines to keep up with increased product throughput. More recently it has been developed to dispense the encapsulation materials required by direct chip attach (DCA) components. Solder paste is also deposited using needles, mainly in rework applications or when the substrate is not planar.

When more than one electronics interconnection material is required to be deposited, stencil printing and needle dispensing are often used in conjunction with each other. A classic example of this is for DCA assembly. Initially solder paste can be applied by stencil printing and then after reflow, underfills are applied by needle dispensing.

This report reviews two alternative deposition techniques, solder jetting and relief printing, the latter a development of stencil printing. Neither is intended as a replacement for stencil printing or needle dispensing, but both have potential to fill niche applications and these are investigated here.

2. SOLDER JETTING TECHNOLOGY AS A METHOD OF DEPOSITING SOLDER

In the early 1990s, the technology of ink jet printing, which was primarily developed for colour printing of computer graphics, was adapted to produce a method for dispensing molten solder globules. The solder jetting systems produce a stream of molten solder droplets that are directed at a printed circuit board or semi conductor chip. When the molten stream hits the cool substrate, the droplets solidify, forming a well controlled solder deposit. A schematic of the process equipment is shown in Figure 1. Two basic techniques have evolved, continuous deposition and “drop-on-demand”.

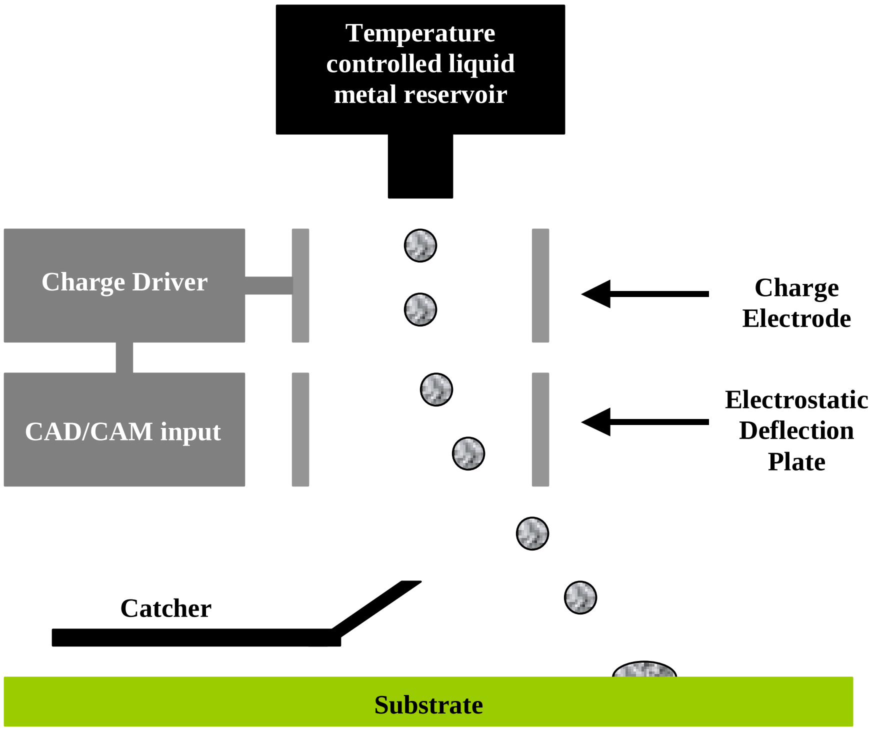


Figure 1: Schematic of continuous deposition solder jetting

2.1. Continuous Deposition Solder Jetting

Continuous deposition solder jetting produces a stream of regularly spaced droplets of molten solder. These droplets pass through an electrode system that charges the individual solder droplets. Electrostatic deflection plates can then control the trajectory of these droplets, and the droplets “aimed“ at the target substrate. When no target is available the stream of droplets can be directed into a catcher and the solder recycled through the liquid metal reservoir. Such systems have been reported to produce deposition rates of up to 44,000 drops per second (Reference 1).

2.2. "Drop-on-demand" Solder Jetting

An alternative approach, known as "drop-on-demand", produces individual droplets only when the target substrate is in the correct position. Thus the stream is not required to be diverted but the substrate needs to be manipulated into the correct position beneath the droplet. This technique is somewhat slower than the other method, with droplet rates under 10,000 drops per second (References 2 & 3).

2.3. Applications for Solder Jetting

When originally developed, the main use of solder jetting technology was envisaged as applying surface finishes to PCBs. However, the relatively slow deposition rates and high capital costs of solder jetting have counted against the technology for this application. But, solder jetting does offer a number of advantages for PCB surface finishes. As the industry becomes increasingly environmentally conscious, it should be noted that solder jetting has none of the disposal problems associated with traditional plating solutions. Additionally, with solder jetting technology there is limited waste as the solder is only applied where required or is easily recycled through the equipment.

Solder jetting technology may provide a solution for the solder coating of small production runs as CAD data can be used to directly write solder to relevant pads. A further application maybe for ultra fine pitch PCBs such as those incorporating direct chip attach (DCA). When ultra fine pitch devices are used in conjunction with their coarser pitch siblings, stencil printing can run into problems as a thicker stencil is required to deposit sufficient solder paste for the coarser components, whilst a much thinner stencil is required to deposit paste for the DCA components. Solder jetting would enable PCBs to be pre-deposited with fine solder deposits in the DCA areas, with stencil printing being used to deposit the coarser paste deposits.

When dealing with ultra fine pitch deposits, printing of solder paste can be problematic below 200μm lines and spaces. Solder jetting has been shown to be viable down to at least 125μm lines and spaces and therefore could provide either a partial solution for mixed coarse and fine pitch assemblies or a complete solution for finer pitch requirements. Table 1 presents a summary of the advantages and disadvantages of the various alternatives for depositing solder as a PCB surface finish.

Table 1: Summary of advantages and disadvantages of solder jetting and its alternatives for depositing solder for PCB surface finishes

Method	Advantages	Disadvantages
Plating	Low cost	Plating solution disposal
	Existing process	Multistep photoimaging process
	High volumes due to parallel processing	
Print and reflow	Existing process	Stencil required for each design
		Printer and reflow oven required
Solder jetting	Environmentally friendly	New process
	Direct write capability	High capital cost
	Selective area coverage possible	Relatively slow deposition

Many of the above comments also apply to component bumping and wafer bumping. In the case of components such as ball grid arrays (BGA) or chip scale packages (CSPs), these consist essentially of ultra fine pitch substrates onto which die have been mounted by wirebonding, flip-chip or tape automated bonding. Thus solder jetting could offer an environmentally conscious, direct write option for low volume µBGA, flip-chip or CSP assembly. Solder jetting has also been used for wafer bumping, direct writing of bumps in low volume applications has been exploited (References 4 and 5).

Other possible applications include selected via fill on PCBs. This would require special resist application for plating or a stencil print and reflow operation. Solder jetting would be ideal for applying solder via fill in particular locations. Again, the direct write capability of equipment could be most effectively utilised in low volume applications.

In summary, possible applications for solder jet technology include :

- Wafer bumping for flip chips
- Ball grid array or chip scale package ball deposition
- Substrate bumping for direct chip attach (DCA)
- Very fine pitch PCB surface finish
- Selected via fill on substrates
- Solder deposits on non-planar substrates (3D substrates)

3. RELIEF PRINTING TECHNOLOGY

During the mid-1990s, both equipment manufacturers and materials suppliers put much work into developing stencil printing for deposition of wave soldering adhesive. One result of this development was a technique that became known as “pump printing”. This technique uses thick plastic stencils to print electronics interconnection material through circular apertures, which have to be drilled rather than etched in the plastic. A typical stencil is shown in Figure 2.

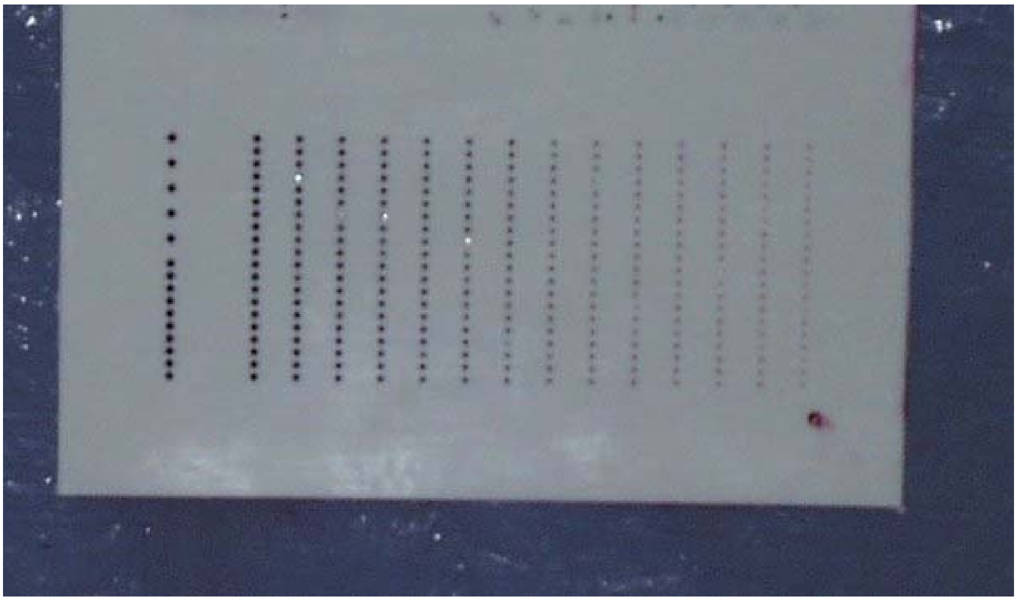


Figure 2: Photo of typical pump print stencil

Because the stencils are typically 1mm or thicker, the stencil apertures take more than one stroke to fill initially, and therefore material has to be “pumped” to fill the aperture initially, thus the term “pump printing”. Subsequently, a single squeegee pass refills the apertures. Because the stencils are thick, not all the material forced into the aperture by the squeegee blade is released from the stencil as the substrate is moved away from the stencil after the squeegee stroke. This is because the adhesion of the material printed can be greater to the edges of the aperture than to the substrate onto which it has been printed. Cohesive forces within the material then dictate the amount of material printed, since, the material will “tear” as the substrate is released from the stencil, leaving the aperture partly filled, with the remainder of the printing material on the substrate. This process is shown schematically in Figure 3.

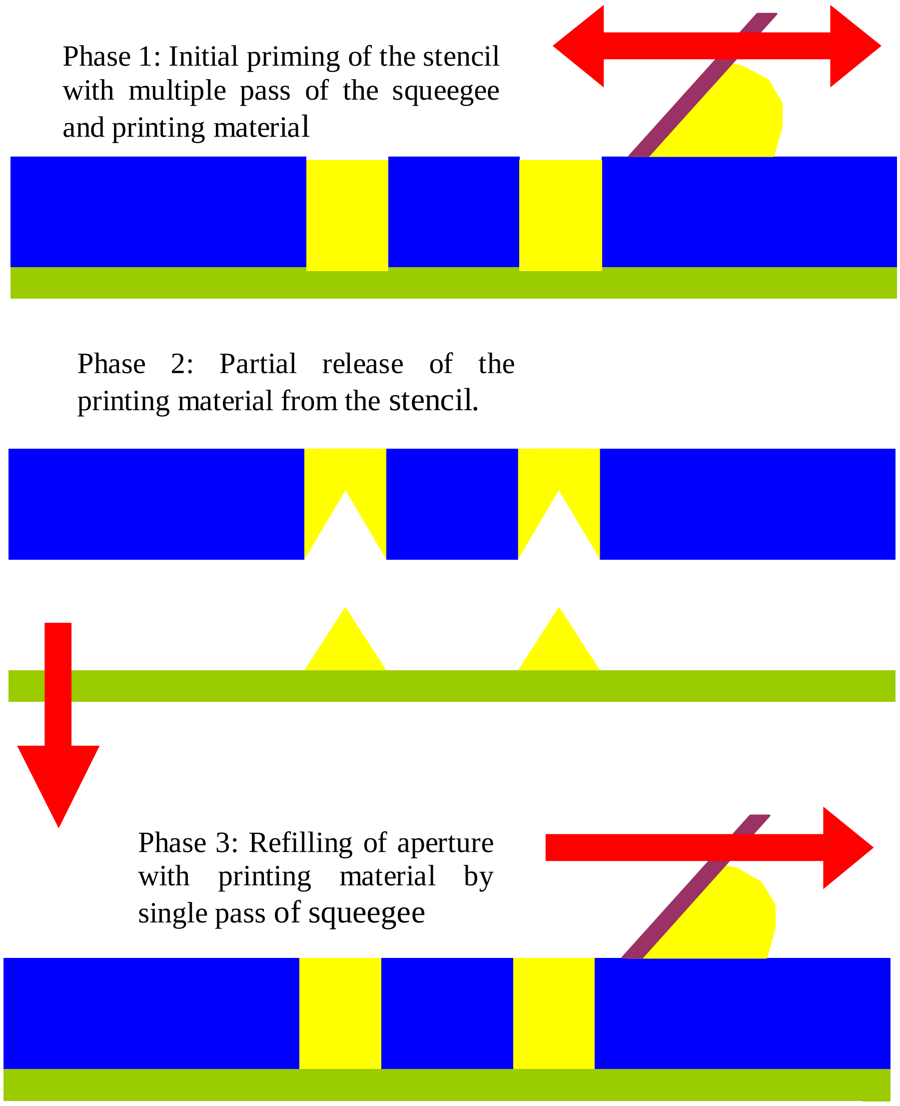


Figure 3: Schematic of Pump Printing Operation

One of the major disadvantages of conventional stencil printing is that the substrate to be printed has to be flat and clear of obstructions. The process of pump printing has been further refined to take advantage of the thickness of the stencil and partially overcome this limitation. By machining cutouts in the underside of the stencil for up to half the thickness of the stencil in the correct locations, the plastic stencil may now be fitted over existing deposits or obstructions. This has become known as relief printing.

A typical relief print stencil is shown in Figure 4. A schematic of how this operation works is shown in Figure 5 based on relief printing of adhesive after printing of solder paste with a normal metal stencil. Relief printing could be used to print a number of electronics interconnection material after initial printing of an alternative materials. A number of these possible applications are detailed below

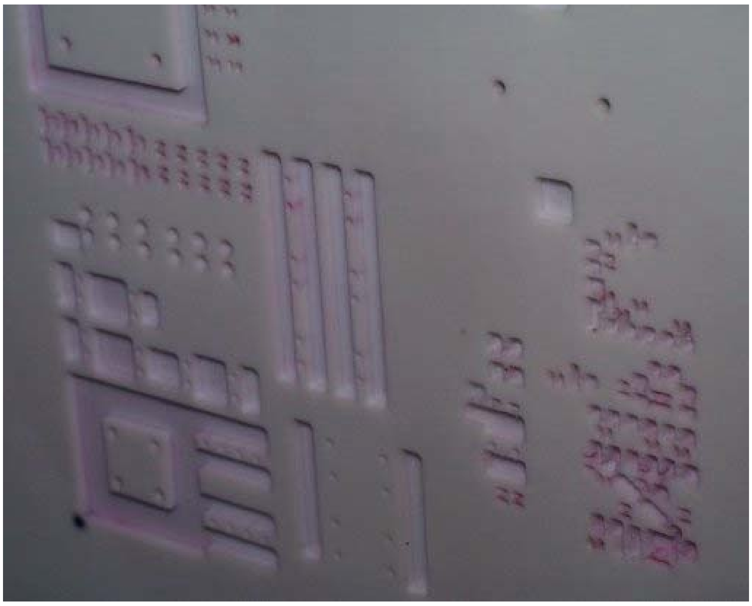
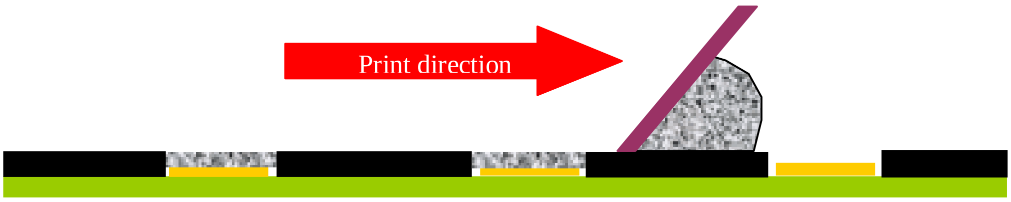
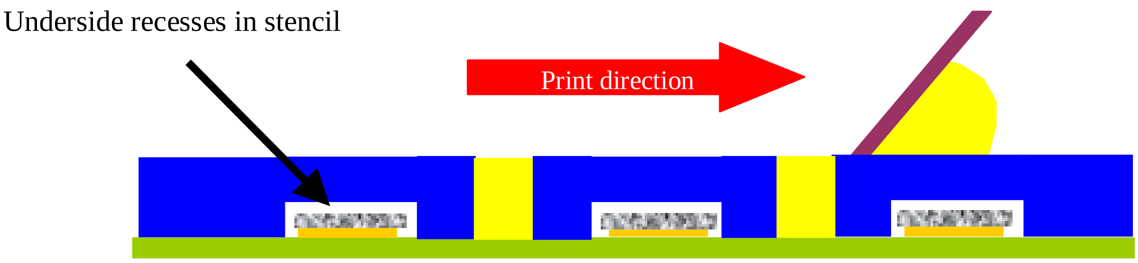


Figure 4: Underside of relief stencil, showing cut outs



Primary print operation using normal metal stencil and material A



Secondary print operation using plastic stencil and material B with underside recesses fitting over previously printed deposits of material A

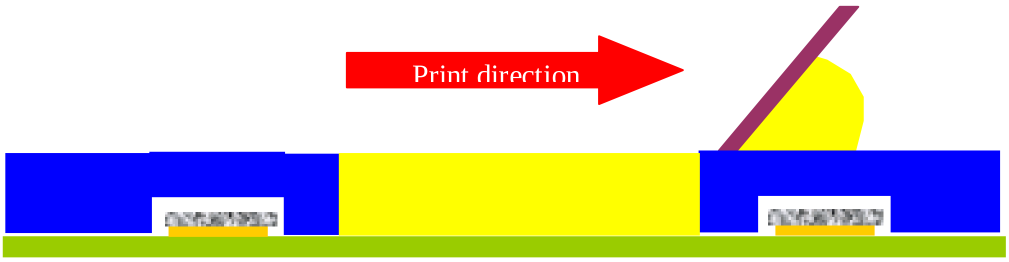
Completed print operation resulting in two different materials printed onto the same side of a substrate

Figure 5: Schematic of relief printing process

3.1. Relief Printing of Thermally Conductive Adhesives and Solder Pastes

Relief printing can be used to print thermally conductive adhesive under SM components such as quad flat-packs (QFPs) to produce better thermal dissipation from the device to the printed circuit board. This is shown schematically in Figure 6. Application of such materials is currently undertaken using a needle dispenser after solder paste printing. Use of relief printing to print thermally conductive adhesive after solder paste deposition could be utilised by a manufacturer who did not wish to invest in needle dispensing equipment, or who would wish to exploit the potentially higher deposition rates offered by stencil printing equipment compared to needle dispensing equipment of similar cost.

An alternative application of the technique could be to print extra thick deposits of solder paste onto an assembly after fine pitch deposits had been printed with a conventional metal stencil. Such applications may be for power devices as part of an assembly that also contains fine pitch QFPs or BGAs. Normally these latter devices would require a thick metal stencil (100 to 125µm), which would not be able to deposit enough paste for the power devices. In some applications, the walls of metals cans are soldered to the assemblies to provide RF shielding. Again relief printing could be utilised to print the thicker solder paste deposits needed for these metal cans. Justification for both these applications would be the same as for the thermally conductive adhesive option above.



Print operation using plastic stencil and thermally conductive adhesive with underside recesses fitting over previously printed deposits of solder paste

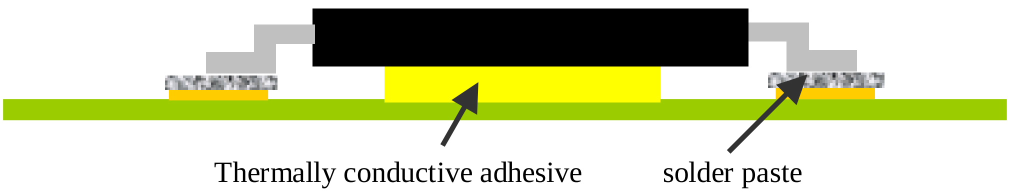


Figure 6: Schematic of relief printing process for thermally conductive adhesive

3.2. Relief Printing of Wave Solder Adhesives After Through-hole Insertion

Figure 7 illustrates a potential exploitation of this printing technique of wave solder adhesive for SM components after insertion of through-hole (TH) components. Automatic cut and clinch operations on through-hole components can be carried out more easily if undertaken on an unpopulated side of an assembly. If SM components have already been glued to the underside of an assembly, carrying out cut and clinch operations on this side of the substrate could lead to damage of the glued SM components or restrictions on where SM components can be placed to avoid the clinching operation. Using relief printing would be an alternative to needle dispensing and the cut-outs on the underside of the stencil would be matched to the already inserted, cut and clinched through-hole components.

Use of relief printing to print adhesive after through-hole insertion could be utilised by a manufacturer who did not wish to invest in needle dispensing equipment, or who would wish to exploit the potentially higher deposition rates offered by stencil printing equipment compared to needle dispensing equipment of similar cost.

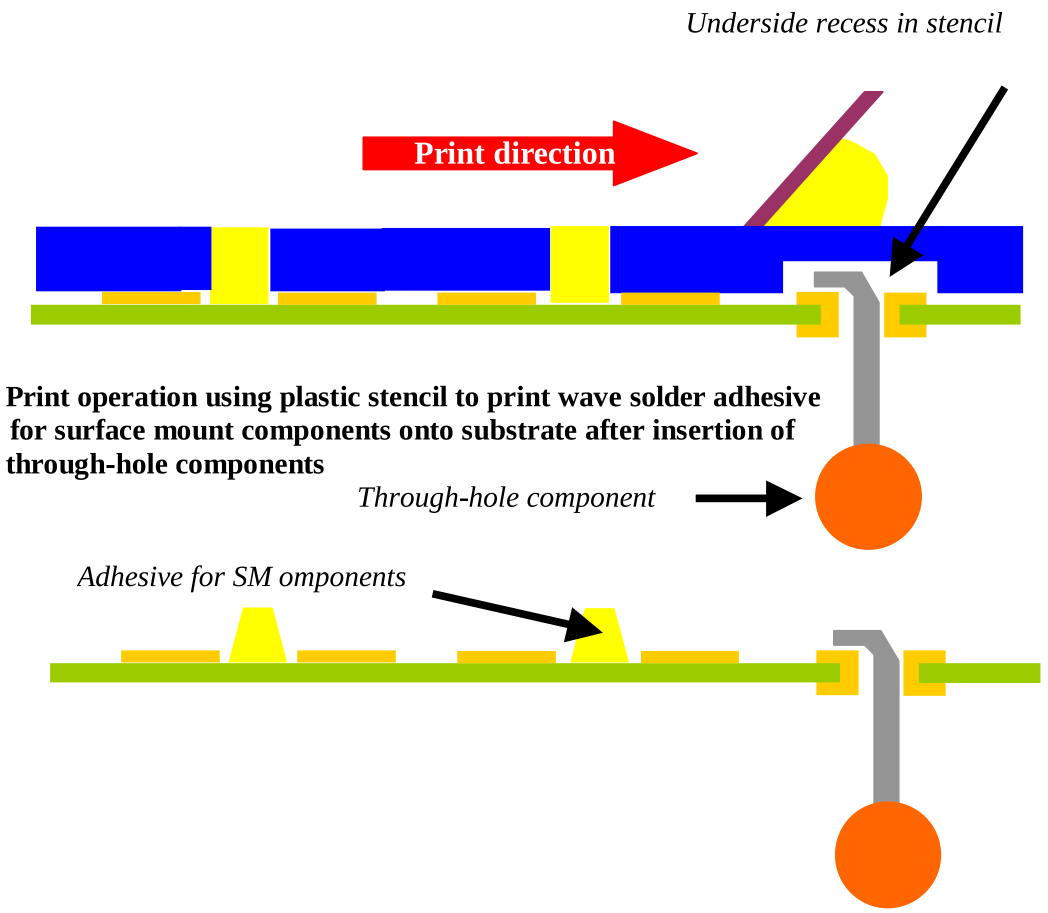


Figure 7: Schematic of printing of wave solder adhesive for SM

3.3. Relief Printing of Solder Paste for Extrusive Reflow

Figure 8 shows a similar application for the thicker plastic stencils although no underside cut-outs are required. This application is to provide solder paste for through-hole (TH) components that are going to be soldered during a reflow soldering operation. This is generally known as intrusive reflow and is undertaken for assemblies which have an overwhelming majority of SM components with a very few number of TH components, all of which will withstand the reflow soldering temperatures. This is typically a TH connector where the additional strength of the TH mechanical connections is required over an SM connector. Normally the TH components would be inserted after solder paste printing and a conventional metal stencil could be used.

In the example shown in Figure 8, the TH component has already been inserted prior to printing. This is known as extrusive reflow. This procedure may be carried out because cut and clinch operations on the TH components may damage the solder paste operation or alternatively, the thicker plastic stencil may be able to deposit better the greater quantity of solder paste required to form a good TH solder joint than that provided by a conventional metal stencil. If ultra fine pitch deposits are required for some SM components then these could be deposited first using a conventional metal stencil, and the subsequent printing for

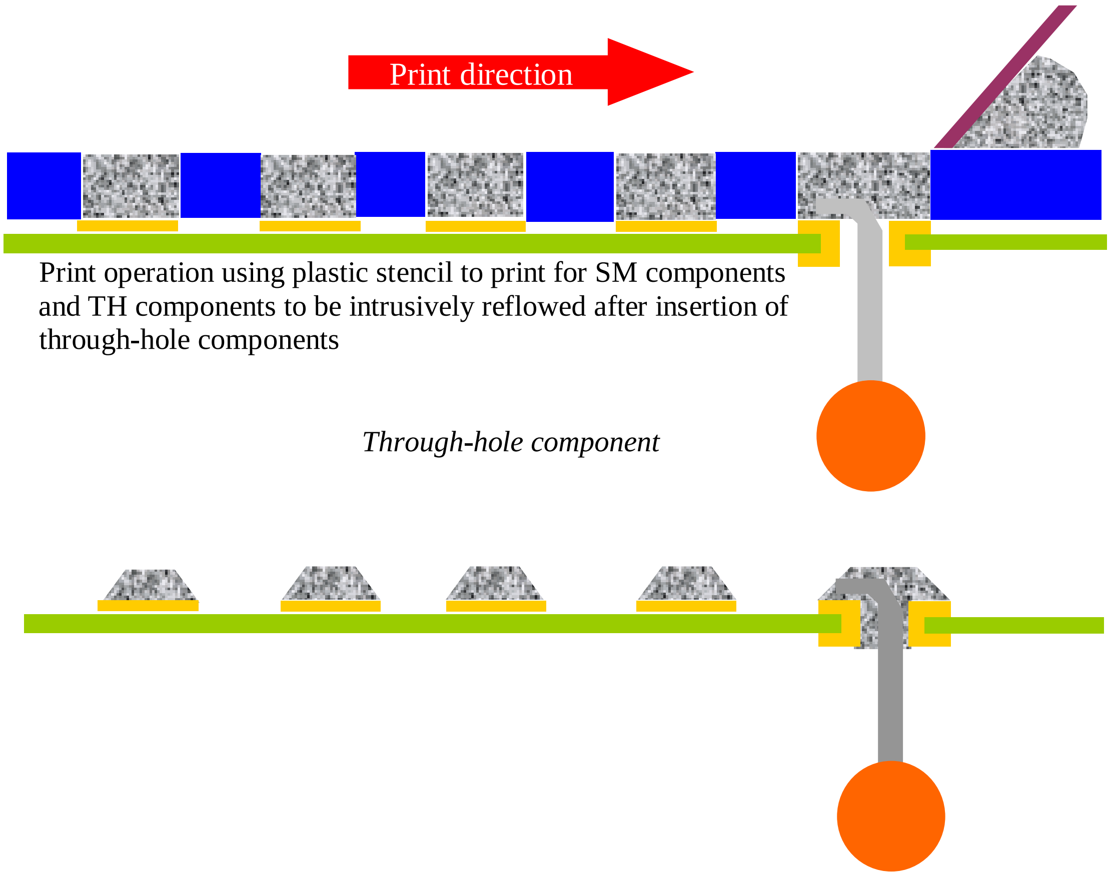


Figure 8: Schematic of printing for extrusive reflow

TH components could be undertaken with a relief stencil with underside cut outs to accommodate the previously printed solder paste.

3.4. *Relief Printing on 3D Substrates*

The thickness of the plastic stencil used in relief or pump printing can also be used to print on substrates with limited obstructions. DEK have reported work undertaken with components specialist Bourns Electronics in battery pack development (Reference 6). In order to save weight, the battery protection circuitry is printed on the inside of the battery case moulding, thus negating the need for a separate printed circuit board. Initially trials to deposit solder paste onto the pads on the inside of the box were undertaken with a needle dispenser, but it was decided that there was insufficient speed for volume production. So DEK developed relief printing to deposit the solder paste through a pump print stencil as shown in Figure 9. The stencil thickness had to be increased to 8mm so that the underside cut-outs could accommodate the side walls of the battery box. An adapted placement system is then used to place components and the whole assembly is then reflow soldered.

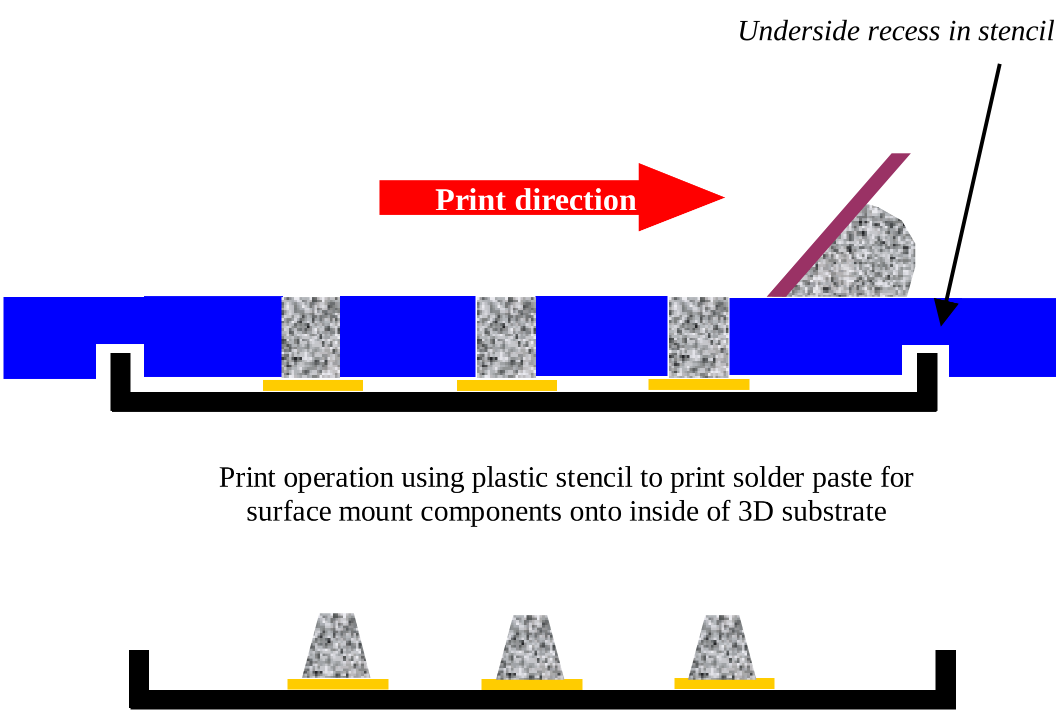


Figure 9: Schematic of printing on 3D substrates

3.5. *Relief Printing in Rework Applications*

Relief stencils could be used in a similar fashion to deposit solder paste onto assemblies which have previously soldered components, such as may be required if components have to be reworked on a significant number of similar assemblies, or a component could not be fitted during original manufacture due to a shortage. Alternatively, the component may not be suitable for soldering at conventional temperatures and needs to be added to the assembly after the main soldering operation using low melting point solder paste or electrically

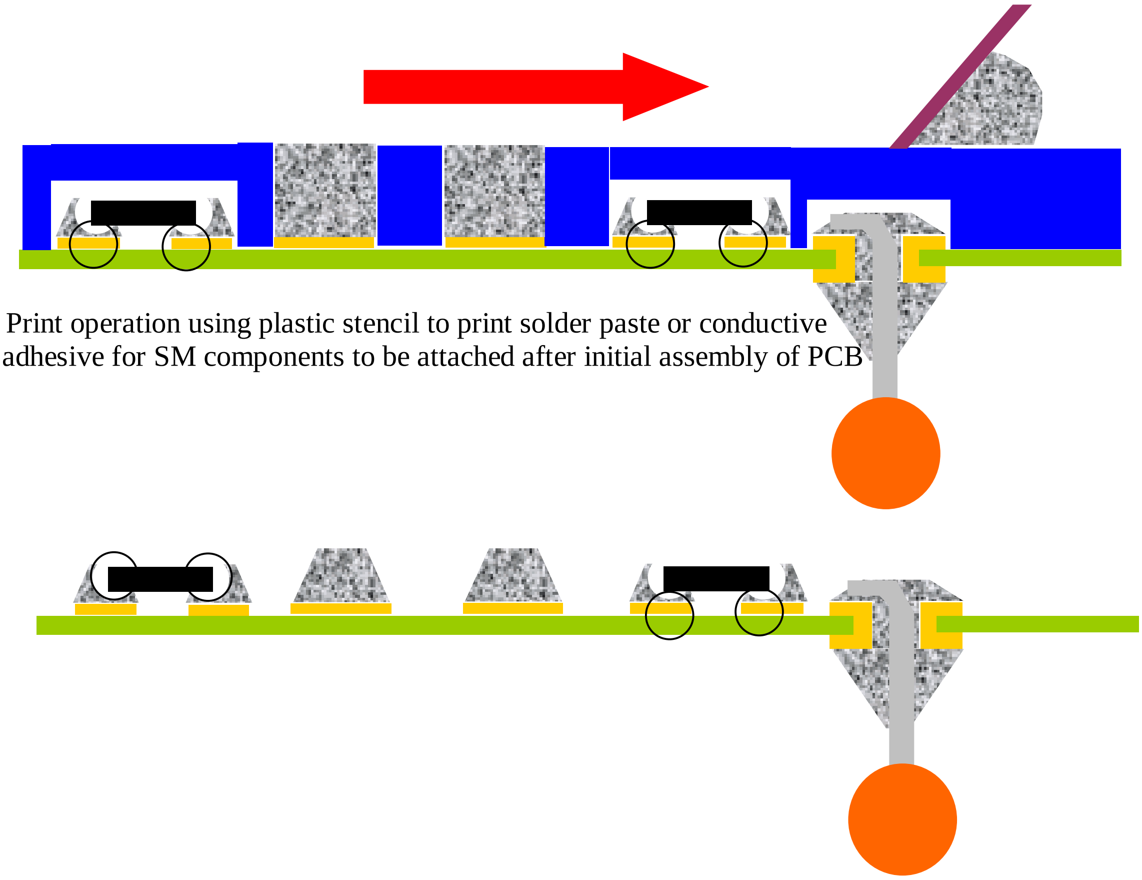


Figure 10: Schematic of printing of paste or adhesive for assembly of components

3.6. Relief Printing of Die-Bond Adhesives

Relief printing could also be used to print die-bonding adhesives under silicon die after SM or TH components have been attached for chip on board applications as shown in Figure 11. The diebonding adhesive can subsequently be cured and the assembly wire bonded. This application is very similar to the thermally conductive adhesive under QFPs. For this type of application, needle dispensing would normally be used to deposit the adhesive. Relief printing could be utilised by a manufacturer who did not wish to invest in needle dispensing equipment, or who would wish to exploit the potentially higher deposition rates offered by stencil printing equipment compared to needle dispensing equipment of similar cost.

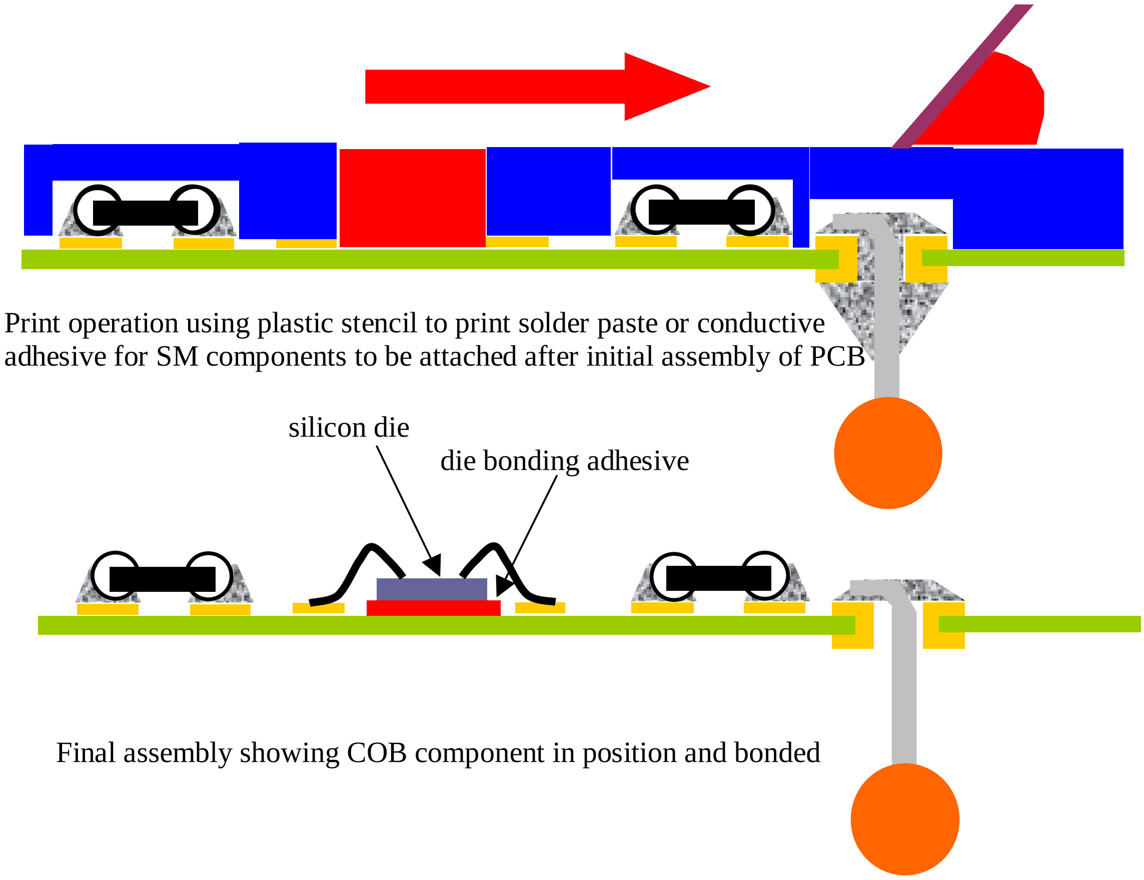


Figure 11: Schematic of printing of die-bonding adhesive for assembly of COB

3.7. Relief Printing of Underfills

Another suggested use for relief printing is the printing of underfills, i.e. materials to seal the underside and improve reliability of flip chips or micro BGAs, after the printing of solder paste. Solder paste would be printed using a conventional metal stencil and then the underfill could be printed between the paste deposits prior to component placement. However, if printed in this manner, the underfill would not be suitable for under-component sealing as the material would need to flow around the solder paste to get a good seal, and as this would happen before the solder joints were formed, it could lead to interference with joint formation. However, if the component were already sealed, such as is the case with µBGAs, then the underfill is only needed as an adhesive to hold the BGA to the substrate, constraining it's expansion relative to the substrate. This will reduce the strain of the solder joints of the µBGA and improve fatigue life. A plan view of the positioning of the solder paste and the constraining underfill is presented in Figure 12. The printing of the underfill after solder paste printing would be the same as shown in Figure 5. Similar issues between dispensing and printing again apply as discussed before.

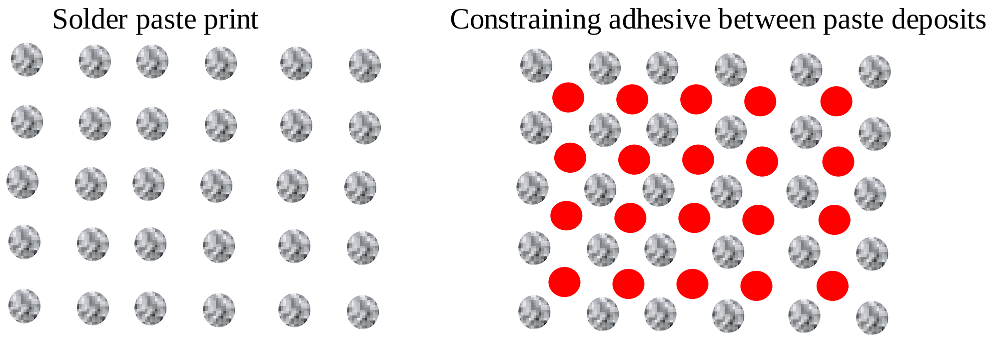


Figure 12: Plan view of relief printed underfill constraining columns

3.8. Relief Printing for Simultaneous Double-Sided Reflow Soldering

The final potential use of relief printing is as an alternative to needle dispensing in the manufacture of simultaneous double-sided reflow soldering (SDSRS). SDSRS is the use of adhesive to hold unsoldered components on the underside of a double-sided assembly so that both sides can be reflow soldered simultaneously. More details of this technique are given in Section 5.

Depositing of the adhesive is normally undertaken with a needle dispenser but an assembler could use the relief print option. Similar issues between dispensing and printing again apply as discussed before.

4. INVESTIGATION INTO RELIEF PRINTING

To examine further the potential of relief printing, an investigation into its capabilities has been undertaken.

4.1. Relief Printing at Different Print Speed and Squeegee Pressures

Two stencils were manufactured from 3.0 and 1.0mm thick nylon materials and comprised a range of circular apertures, from 0.5 to 1.8mm in 0.1mm increments. These two stencils were used for relief printing at different print speeds and squeegee pressures to evaluate the effect these parameters on relief printing performance. The printing was undertaken using an epoxy

SMT adhesive (material A), and a lead-free no-clean type3 solder paste (material B). All the printing reported here was conducted on DEK 265 GSX printer at 24°C/40%RH. A specially designed 45° soft rubber squeegee with 200 mm working length was used for the printing. This squeegee had an angled bottom edge so that the whole blade face was in contact with stencil rather than the normal edge contact as shown in Figure 13.

The adhesive was printed at three different print speeds and squeegee pressures with two stencils. However, printing solder paste in this mode can only be achieved under a narrow range of conditions, and hence these were not varied in the evaluation. The printer parameter settings, stencils and print materials are listed in Table 2. The parameters in bold in each section indicate the variable under investigation.

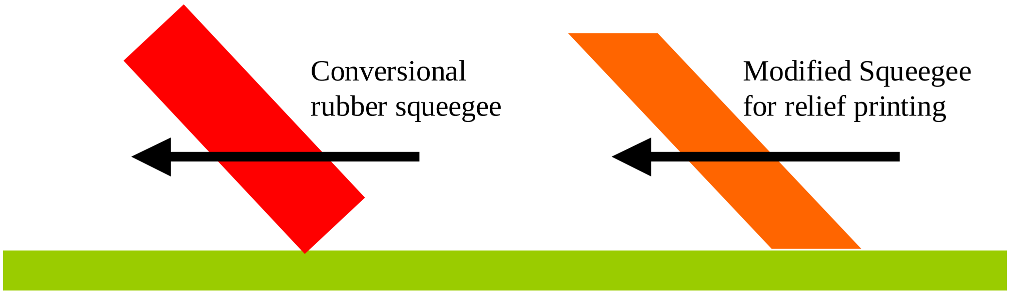


Figure 13: Comparison of conventional squeegee and modified Squeegee for relief printing

Table 2: Experimental conditions for relief printing

Print speed (mm/sec)	Squeegee pressure (Kg)	Stencil thickness (µm)	Separation speed (mm/sec)
SMT adhesive **A** using **1.0** mm stencil			
50	6.0	1000	0.1
75			
100			
75	**5.0**	150	0.1
	6.0		
	7.0		
SMT adhesive **A** using **3.0** mm stencil			
50	6.0	3000	0.1
75			
100			
50	**5.0**	3000	0.1
	6.0		
	7.0		
Solder paste **B** using **1.0** mm stencil			
75	6.0	1000	1.0
Solder paste **B** using **1.0** mm stencil			
75	6.0	1000	1.0

4.2. Evaluation of Print Deposits

A laser scanning profiler with a triangulation sensor was used to measure the deposits. Because the solder paste and SMT adhesive reflect light poorly, a thin layer of gloss white paint was sprayed over the print deposits and board to increase reflection, as shown in Figure 14. Four dots were measured for each diameter aperture, and the average of 4 dots for each aperture size was recorded. Dot volume and height were selected to characterise the print deposit (Reference 7). The dot volume is given as a ratio of stencil aperture volume and absolute deposit volume. The dot height is the maximum height of the deposit dot, as illustrated in Figure 14.

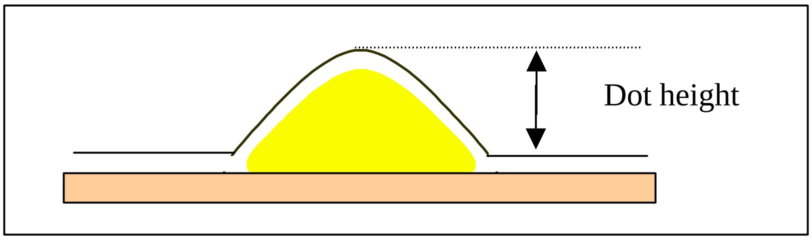


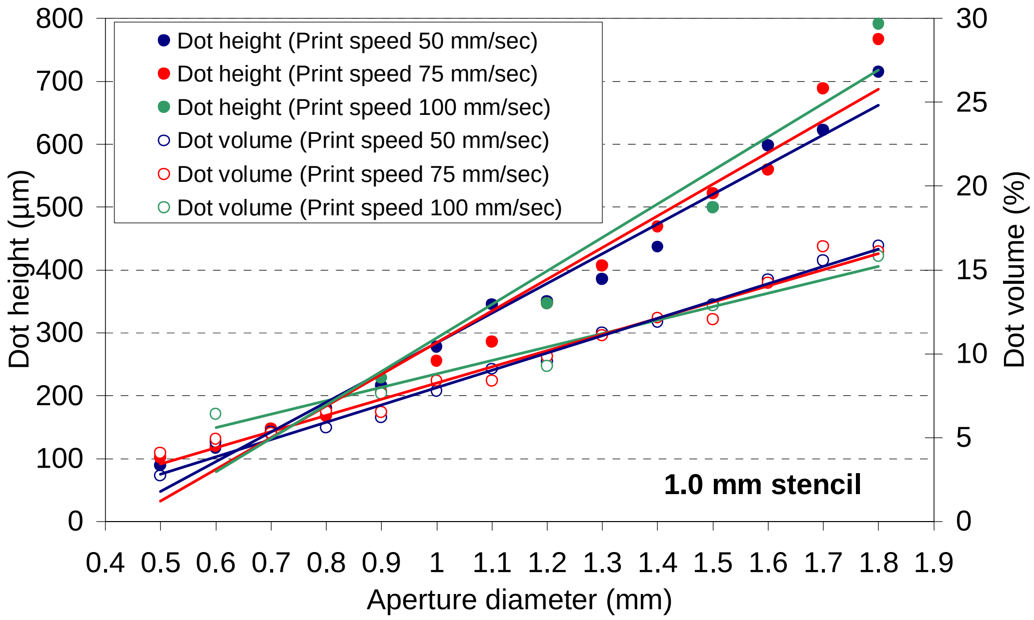
Figure 14: Dot height measurement

4.3. Effect of Print Speed and Squeegee Pressure on SMT Adhesive Relief Printing

The effects of print speed on the dot volume and dot height of the SMT adhesive A are plotted in Figure 15. Figure 16 shows the effect of print pressure on the dot volume and dot height of SMT adhesive A. The following points should be noted:

- Print speed and squeegee pressure did not affect the dot height and dot volume for SMT adhesive A, and the dot volume was much lower for relief printing than that for conventional printing.
- The fractional dot volume significantly dropped with stencil thickness increase, but the absolute volume was approximately constant for the two stencils; i.e. the volume was independent of stencil thickness. Dot height was not affected by stencil thickness.
- Dot height and volume increased linearly with aperture size.

The low dot volume, particularly for 3mm stencil, indicates that for adhesive relief printing the majority of the adhesive remained inside the stencil aperture after print board release from the stencil. ***Thus for adhesive relief printing, performance is dominated by the release of the adhesive from the aperture.*** The previous work had shown that the print volume is related to the ratio of stencil aperture open area and aperture wall area (Reference 8). This is quantified in Figure 17 where the dot volume is plotted against the (open/wall) area ratio for the two stencils thickness. The Figure clearly shows that there is a linear trend between two variables, aperture open area and aperture wall area. Print volume increases with the (open/wall) area ratio. ***Therefore, for relief printing the print volume is dependent on this ratio.*** High print volume could be achieved by increasing the aperture size, but not the stencil thickness.



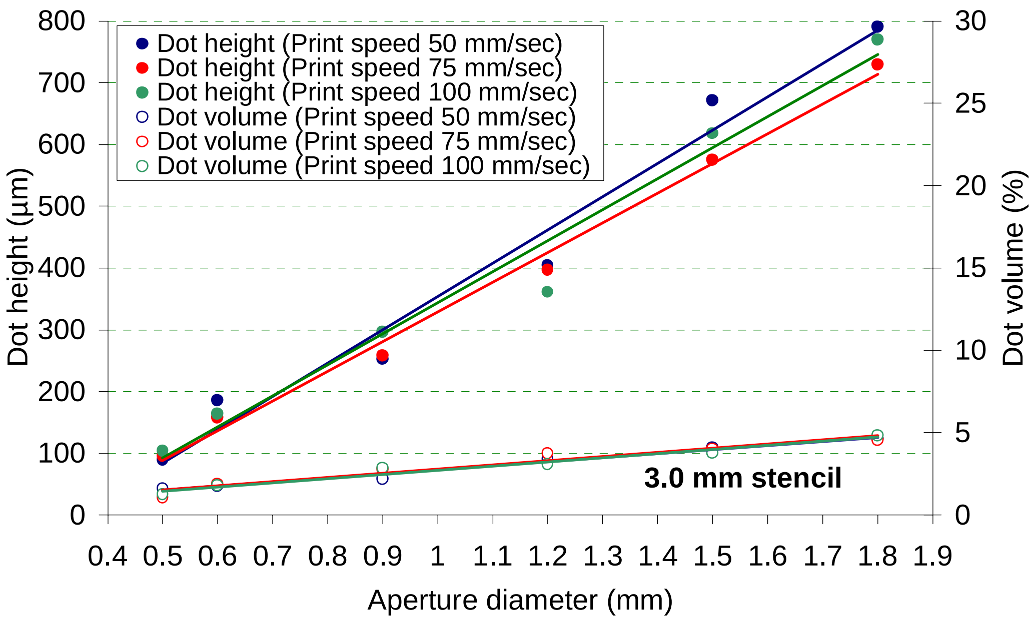
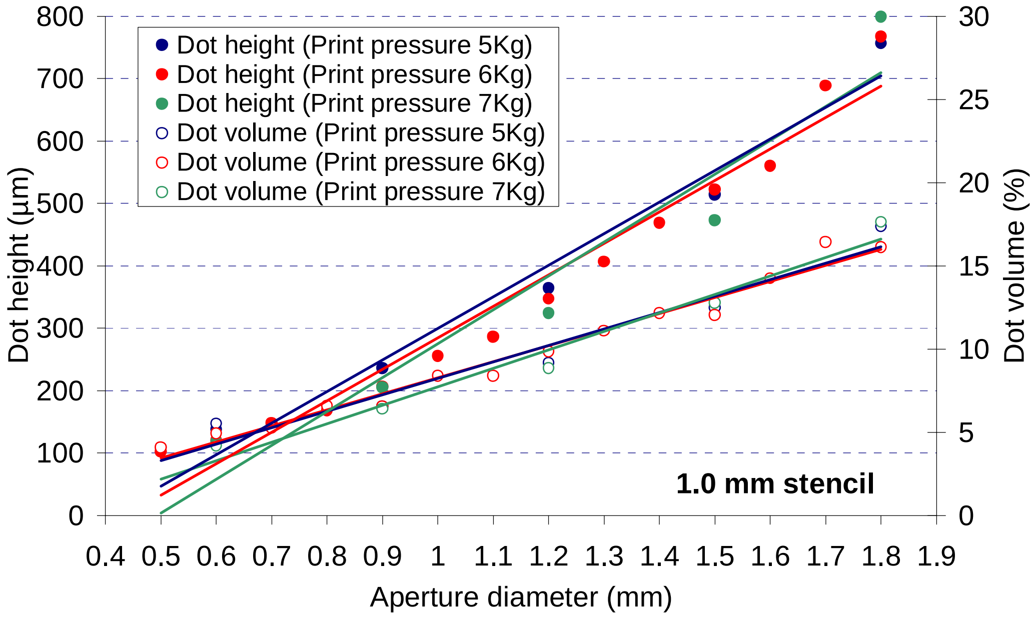


Figure 15: Dot height and volume with print speed
for 1.0 and 3.0mm stencils



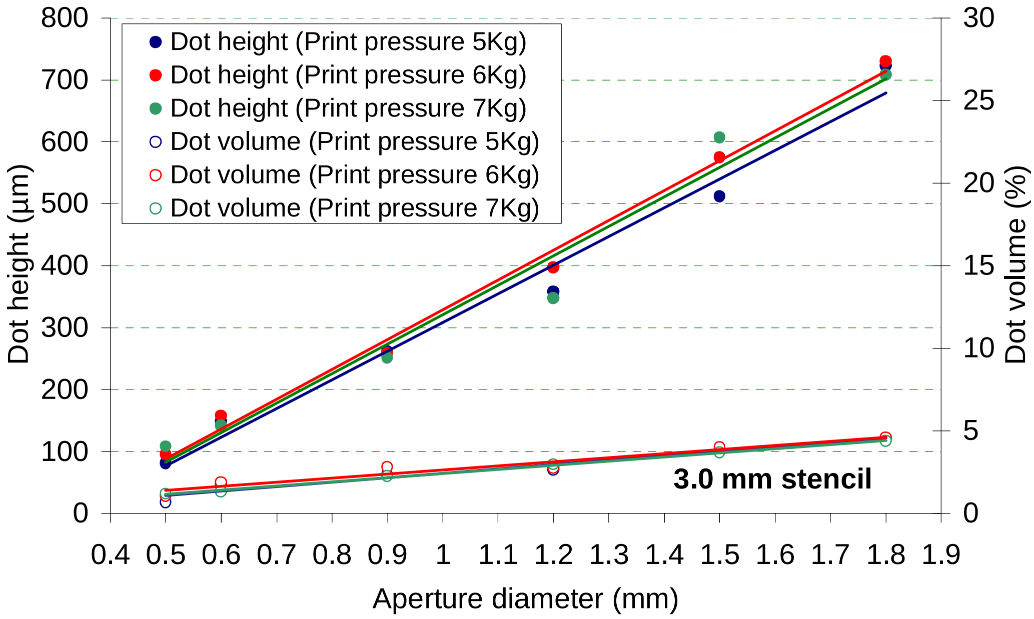


Figure 16: Dot height and volume with squeegee pressure
for 1.0 and 3.0mm stencils

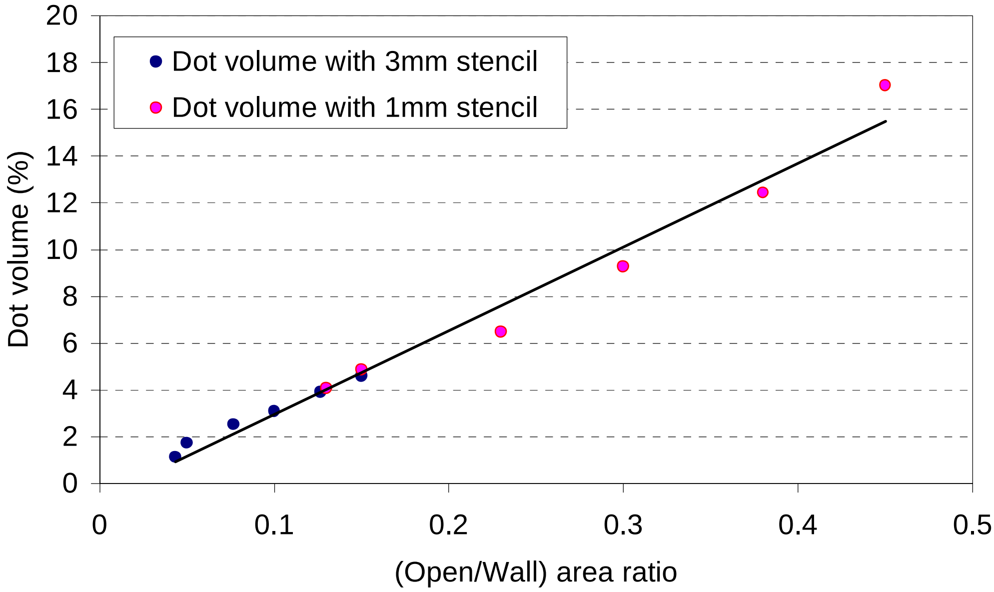
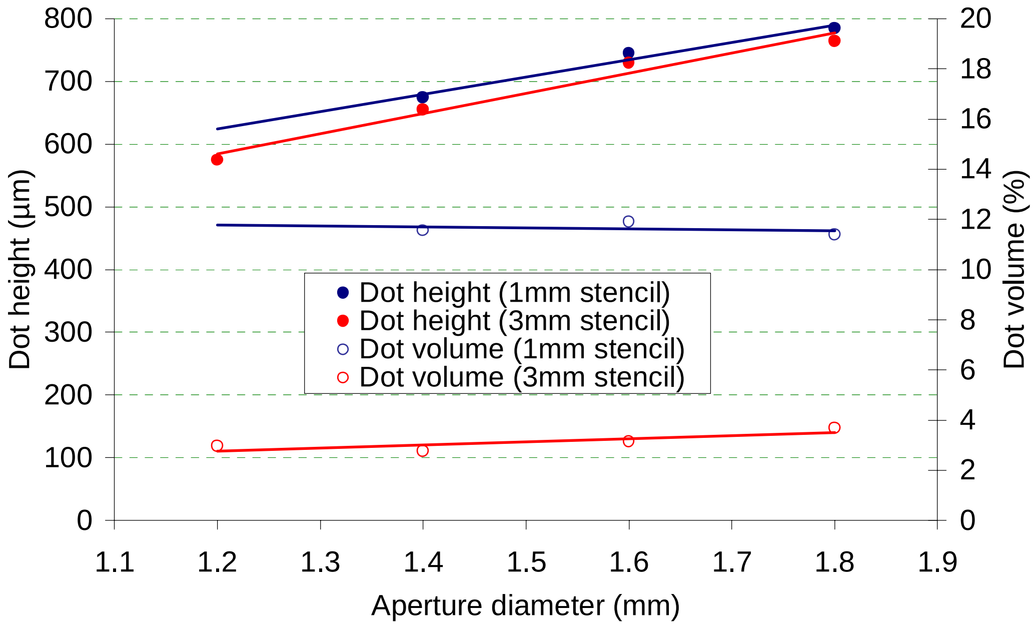


Figure 17: Dot volume with (open/wall) area ratio
for 1.0 and 3.0mm stencils

4.4. Relief Printing of Solder Paste

Solder paste B was also printed using the 1.0 and 3.0 mm stencils. The variation of dot height and volume with aperture size for the two stencils is shown in Figure 18. There are some important points to note from this Figure. For both the 1.0 and 3.0 mm stencils, very little paste was printed when the aperture diameter was less than 1.2 mm. The apertures filled adequately but the cohesive force within the paste was sufficiently high that no paste was released from the stencil aperture.

Dot height increased linearly with aperture size when the aperture diameter was equal or larger than 1.2, but was not affected by stencil thickness. However, the fractional dot volume remained unchanged with aperture size, but the absolute volume was broadly similar for a given aperture for both stencil thickness. Hence the aperture size and the cohesive forces within the paste, and not the stencil thickness, dictated the size of the dot. ***Therefore, for solder paste relief printing, large dot volumes only can be achieved by increasing aperture size, but not stencil thickness.*** Relief printing is not capable of producing paste deposit when the stencil aperture is less than 1.2mm.



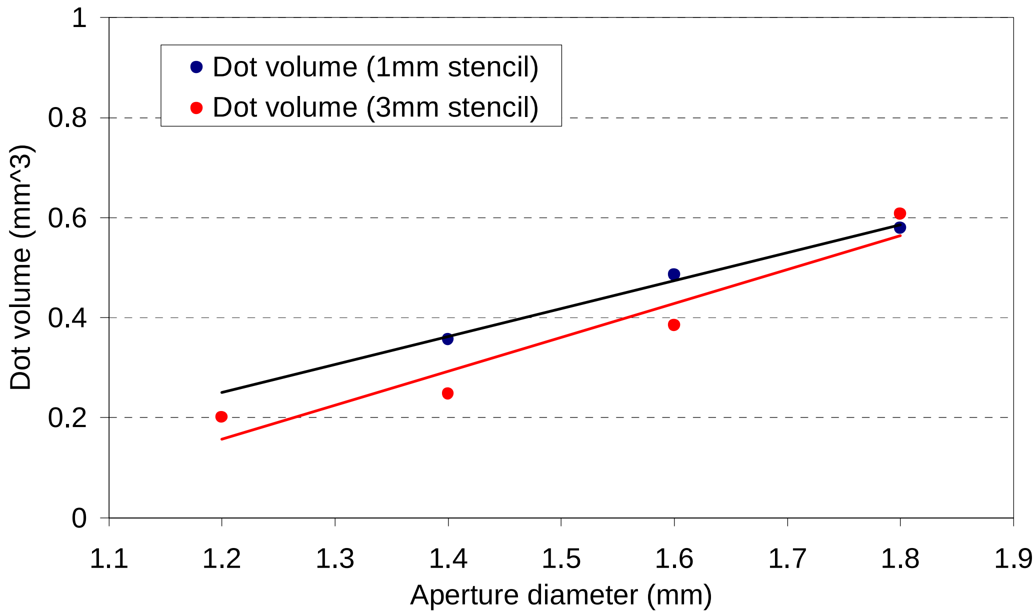


Figure 18: Absolute and fractional dot volume and dot height with aperture size

5. INTRUSIVE REFLOW SOLDERING

Intrusive reflow soldering is a production process allowing the assembly and reflow of through-hole components at the same time as SM components. This cuts down costs related to secondary operations i.e. hand, selective or wave soldering, of these parts. Typical parts that can be intrusively reflowed include pin connectors and IC sockets (PGA), but the technique can easily be extended to other components such as DIL packages, through-hole resistors, capacitors, or any other leaded components, provided that such parts can withstand reflow soldering temperatures. The solder to form the joint between the PCB and the TH component is applied by stencil printing solder paste on the lands around the mounting holes and into the TH itself, as part of the normal paste printing operation for SM components. The pin components are then manually or automatically inserted into the PCB together with placement of SM devices and the whole assembly is reflowed.

5.1. Intrusive Reflow Assembly Issues

There are five critical aspects that effect intrusive reflow :

Length of component pins
When a component is available in different pin lengths, the pin length should be carefully chosen to match the thickness of the PCB. Too short a pin will cause problems with inspection. If the pin is too long, then some solder paste will be pushed away from the TH on the end of the pin during insertion and therefore not be available to help form the joint.

Diameter of PTHs
The diameter of the mounting holes should be as close as possible to the pin diameter, allowing for tolerances needed for automatic or manual insertion. If the pin is significantly smaller than the PTH, then less of a joint will be formed for the fixed volume of solder paste available.

Stencil aperture design
The stencil design should allow sufficient paste to be printed, since this will critically influence the printed solder paste volume, and hence the resultant solder joint. The printed area usually covers the conductive pad around the component lead. The apertures could be locally plated to achieve a thicker stencil around the location of the TH component, and hence more volume of solder paste can be printed.

Printing process
Printing adequate volume of solder paste may demand higher printing pressure to force extra solder paste down into the PCB holes. Hence 45° squeegee blades or enclosed print heads would be beneficial.

Reflow profile
Voiding in intrusive reflow solder joints can be an issue. This is because on heating the paste in the hole, the solvents cannot escape side-wards because they are surrounded by the impenetrable plating of the hole. The gas can only escape upwards or downwards, taking longer to do so and therefore some solvent may remain in the joint when it solidifies. Care therefore needs to be taken in a preheat stage of the reflow process to remove as much solvent as possible to reduce voiding in the joints.

The above parameters are very application dependent and should be discussed with component and stencil manufacturers. Details of the effect of aperture shape and size on intrusive reflow joint formation are given in Reference 9.

5.2. Intrusive Reflow Measurement Issues

The volume of paste needed to form an intrusive reflow solder joint is significantly affected by the amount of solder paste forced into the hole during the printing operation. It is therefore a process control requirement periodically to measure the fill of the hole prior to pin insertion.

Automatic and semi-automatic measurement systems for solder paste are normally based on laser displacement sensors. Two different types of sensor have been assessed for their suitability for measuring solder paste deposits inside PTHs.

5.2.1. Differential or Dynamic Focusing Sensor

A differential focusing sensor projects a laser beam vertically downwards onto the target and focuses this beam to a predetermined spot size. From this the distance to the target can be calculated. However, this system did not function well in the evaluation because it was unable to cope reliably with large rapid changes in measurement height associated with the change from the land area of the PCB to the paste inside the hole (shown schematically in Figure 19). No further evaluation was undertaken of this sensor type

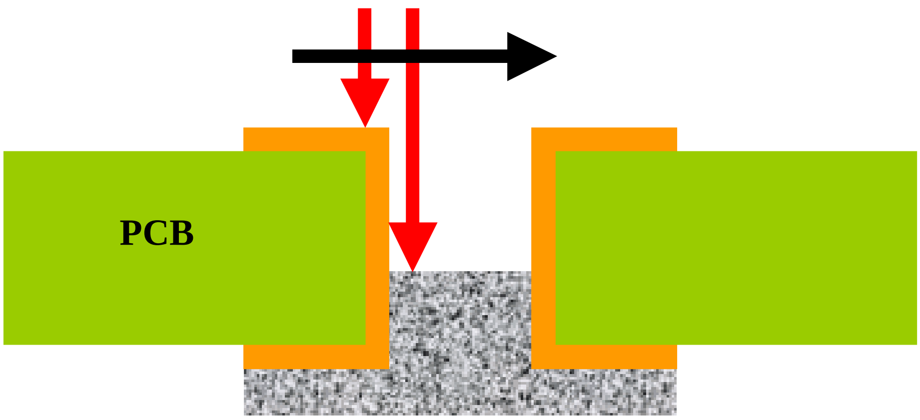


Figure 19 : Schematic showing rapid height changes experienced with intrusive reflow measurement

5.2.2. Triangulation Sensor

An alternative to the differential focusing sensor is a triangulation sensor. This uses an angled detector to focus the reflected beam of a vertically incident laser, shown in Figure 20. The detection angle is dictated by the minimum separation distance of the incident beam and the detector, and the close vicinity of the sensor to the workpiece to maximise accuracy of the measurement. Other sensor heads are available for the system but these have lower resolution. In the sensor head used here, these limitations resulted in a fixed reflection angle of 40°. Thus the depth that can be measured with such a sensor is limited by hole diameter if the reflected beam is to clear the edge of the hole and be detected. The following Section describes work undertaken to evaluate the maximum depth, which triangulation sensor can measure for the different hole-sizes.

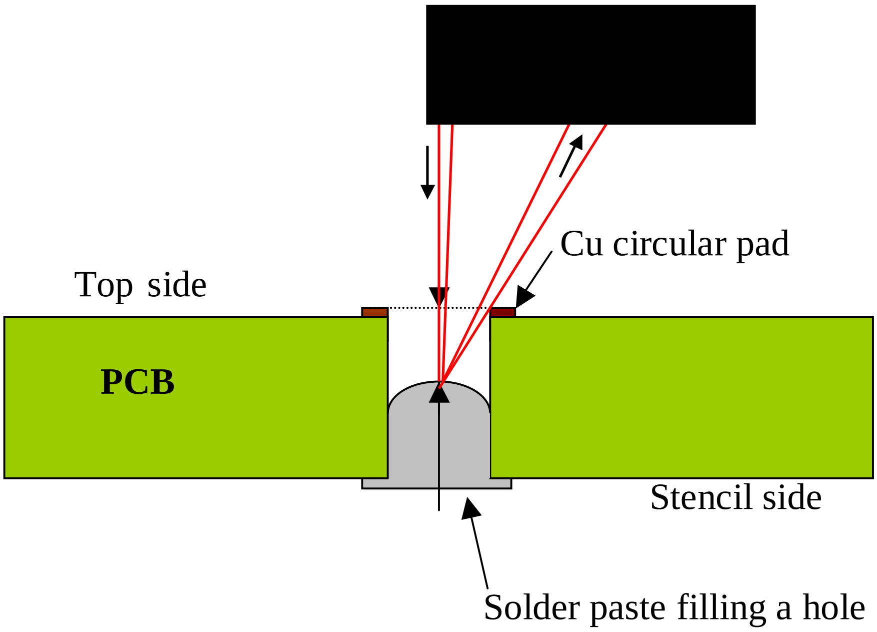
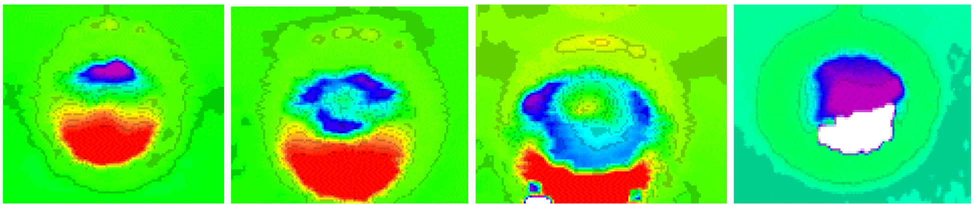


Figure 20 : Schematic of height measurement of solder paste filling a PCB hole

5.2.3. Triangulation Sensor Maximum Depth Measurement

For the work here the board thickness was 1.6 mm and a range of holes was manually printed using a no-clean type 3 solder paste. The diameters of the holes in the board were 0.55, 0.70, 0.80 and 0.97 mm. The holes were filled with paste to different depths by adjusting the print pressure. Examples of colour contours are plotted in Figure 21, and the blue end of the spectrum indicates low areas and red indicates high areas of the scan. The maximum measurable depth as a function of hole diameter is presented in Figure 22. This Figure represents the maximum depth that can be measured over half the area within the hole, as this is considered to be representative of the whole area. Additionally, should the whole area be required to be measured, an additional measurement could be undertaken with the PCB rotated through 180°. The printed PCB with connector inserted is shown in Figure 23.



ϕ 0.55mm ϕ 0.7mm ϕ 0.8mm ϕ 0.97mm

Figure 21 : Example of contour maps of scanned holes in a PCB filled with solder paste (as a function of hole diameter)

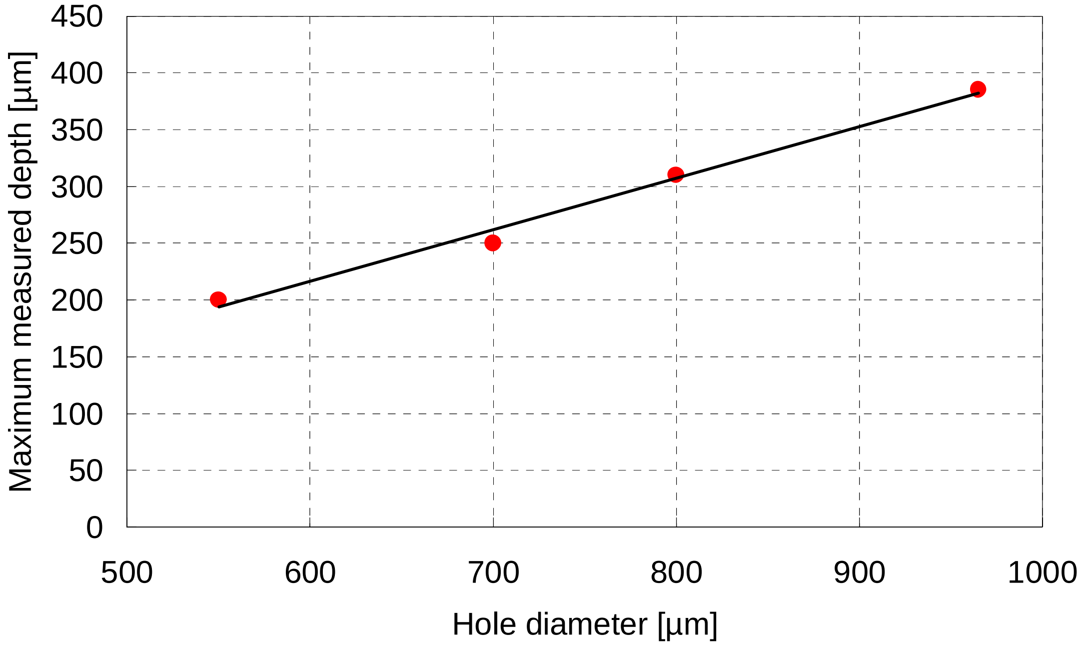


Figure 22 : Maximum available height measured from top of PCB to solder paste surface

From Figure 22 it can be noted that even for hole diameters approaching 1mm, it is only possible to measure depths of 0.4mm, a quarter of the possible depth required to be measured (PCB thickness of 1.6mm). It is therefore considered that the current equipment sensor is unsuitable for measurement of solder paste fill in PTHs. A sensor with the detector placed at a smaller angle to the incident beam would be required, but such sensors require a larger sensor to work-piece distance and have a correspondingly lower resolution. Based on Figure 22, it is estimated that a sensor with a reflected angle of less than 10° would be required to measure down the full depth of a PTH of 0.55mm diameter in a 1.6mm thick PCB.

6. PRACTICAL EXPERIENCES OF SIMULTANEOUS DOUBLE SIDED REFLOW SOLDERING (SDSRS) MANUFACTURE

To explore the potential of relief printing, a demonstrator of the technology was built using SDSRS assembly techniques.

6.1. SDSRS Test Assembly Design

The demonstrator was based on an original design by R. Willis of EPS, and was designed to incorporate an active circuit comprising a series of timed flashing LEDs as well as a number of other typical surface mount components to assess their suitability for the process. Both sides of the circuit are shown in Figure 23. The components incorporated into the assembly are listed in Table 3.

Table 3: Components incorporated into SDSRS design

Side	Number	Component type	Comments
A	10	R0402	0402 Chip Resistor
A	34	R0603	0603 Chip Resistor
A	7	C1206	1206 Capacitors
A	1	QFP160	160 pin 0.65mm pitch QFP
A	1	3002	Battery Clip
A	1	CR2032	Lithium Battery (coin)
A	1	PBGA256	256 Plastic BGA perimeter array 1.0mm pitch
A	13	SOT23	PNP Transistor IBMT4403D1
A	1	SOIC8	Small Outline IC (8 Pin, 555 Timer)
A	1	µBGA	µBGA 46 pin 0.75mm pitch
A	18	LED	Chip LEDs
A	6	SOIC14	Small Outline IC
A	1	TA006TCM106KBR	Tantalum Capacitors, 10uF
A	2	TSOP32	Thin Small Outline Package 0.5mm pitch
A	10	SOT223	Power Transistors
B	15	SOT323	Mini Small Outline Transistor
B	1	PBGA256	256 plastic BGA perimeter array 1.0mm pitch
B	1	QFP160	160 pin 0.65mm pitch QFP
B	12	R0805	0805 Chip Resistors
B	10	C1206	1206 Capacitors
B	6	SOIC14	Small Outline IC 1.27mm pitch
B	10	SOT23	Small Outline Transistor
B	10	R0402	0402 Chip Resistor
B	1	µBGA	µBGA 46 pin 0.75mm pitch

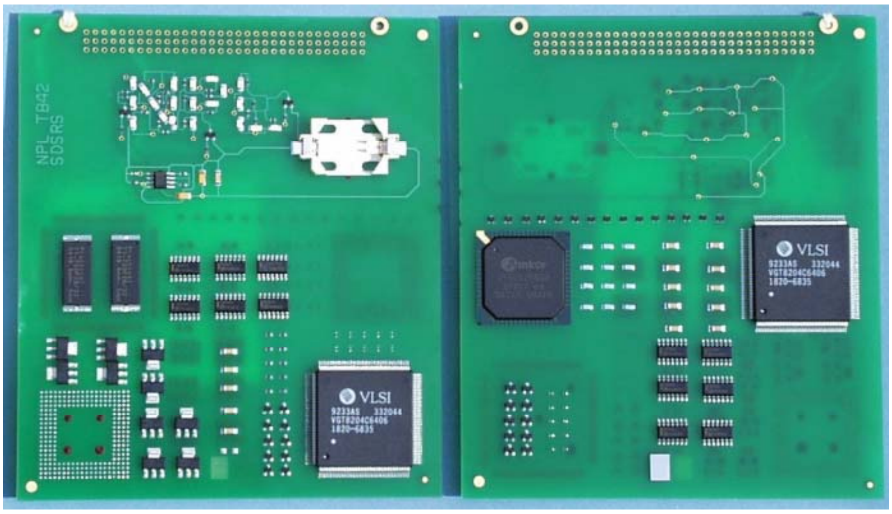


Figure 23: SDSRS Test Assembly (NPL Test Board TB42)

6.2. *SDSRS Assembly*

Assembly of the demonstrators was undertaken using the stages illustrated in Figure 24. Side A was initially printed with a no-clean type 3 SnPbAg solder paste using a 125μm thick laser cut stainless steel stencil on a DEK 265 stencil printer. A wave solder adhesive was then printed using a 3mm thick plastic stencil with the underside milled out to place over the already printed solder paste. The relief stencil, as shown Figure 4, was 3mm thick. Several prints were initially required to fill the stencil apertures completely with adhesive before satisfactory printing of the adhesive was achieved. Special squeegee blades with a large flat edge as previously discussed in Section 4, were used to ensure apertures were adequately filled.

Components were then placed onto side A and the assembly placed in an air circulation oven at 75^{o}C for 30 mins to partially cure the adhesive. After cooling to room temperature, the assembly was inverted and again printed with the same solder paste, this time on side B. The DEK 265 printer has an edge conveyor so that no components on side A were disturbed during the printing operation. All under board supports were removed so the board was only supported along two edges during printing. After printing, side B was manually populated and then the whole assembly was reflowed with side B uppermost in a 5 zone convection reflow oven. During reflow, four corner pillars supported the assembly, so that the components on side A were not in contact with the mesh belt of the reflow oven.

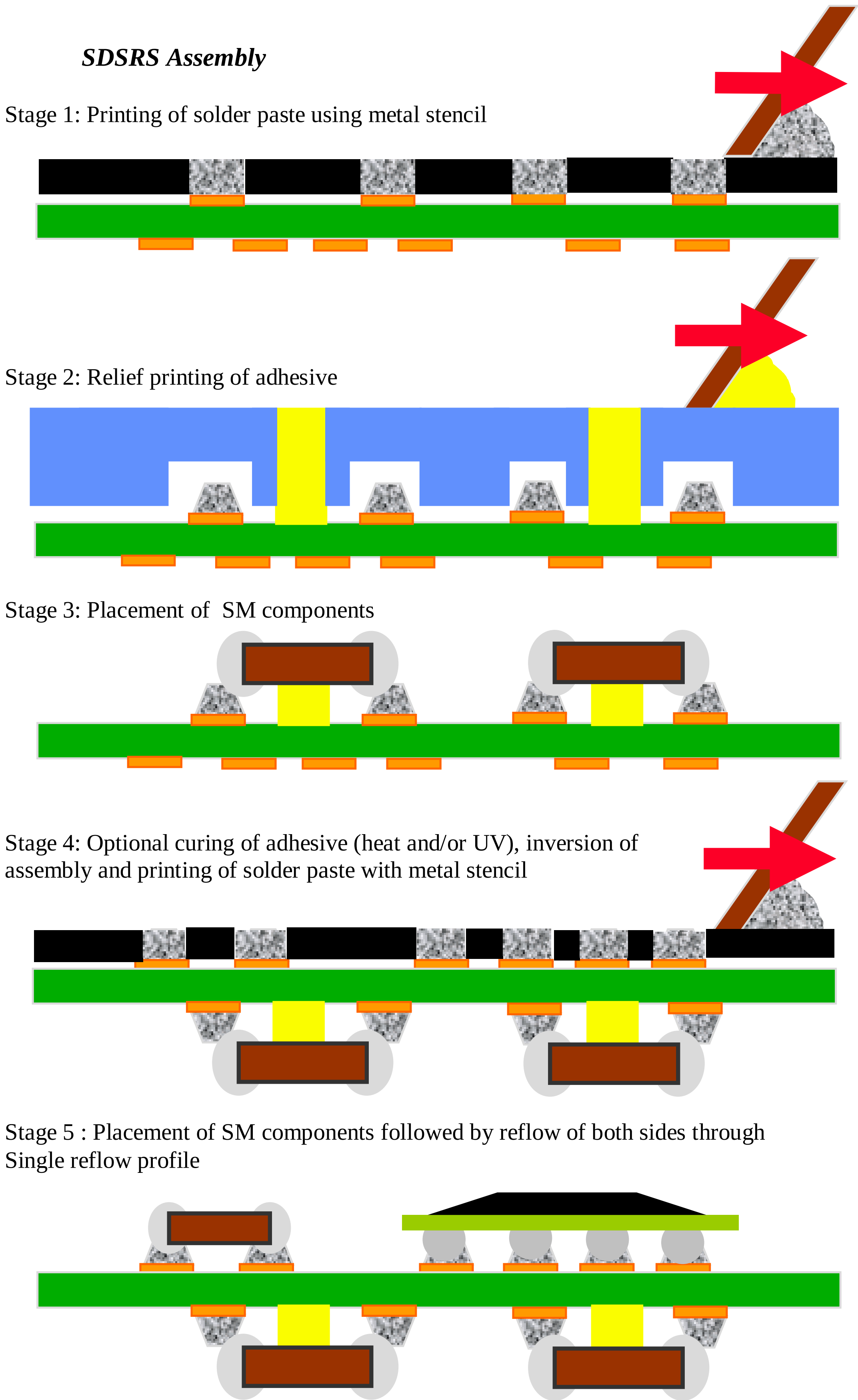


Figure 24: Schematic of SDSRS assembly using relief printing

6.3. SDSRS Assembly Results

Paste printing: No problems were encountered with either paste printing operation, which were no different from conventional solder paste printing.

Adhesive printing: Adhesive printing through the relief stencil proved relatively straight forward for the larger components as can be seen in Figures 25 to 28. With plenty of space between solder paste deposits, the apertures for the adhesive could be relatively large and therefore forcing adhesive through the aperture into contact with the board was possible. Adhesive was successfully relief printed for PBGAs, QFP160s, SOT223s, battery clips, TSOPs and SOICs.

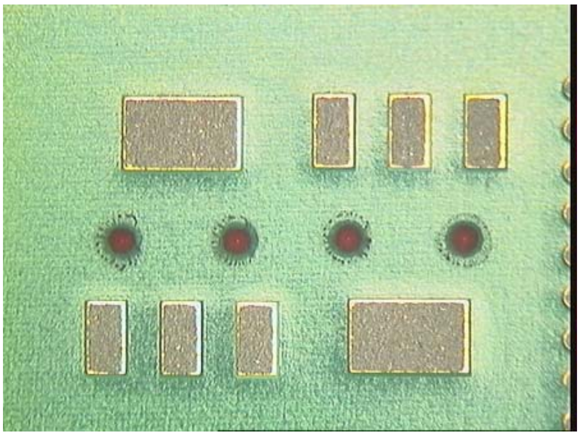


Figure 25: Paste and adhesive for SOT223 component

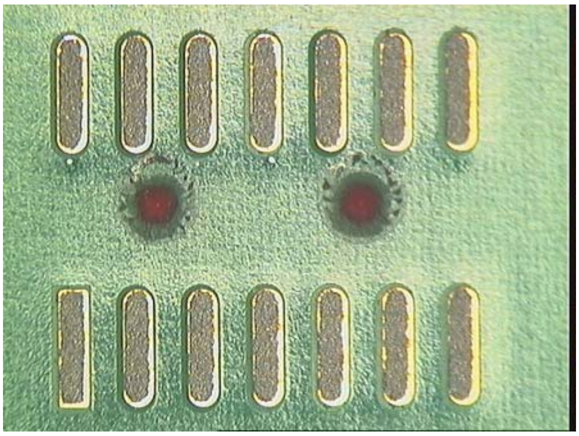


Figure 26: Paste and adhesive for SOIC14 component

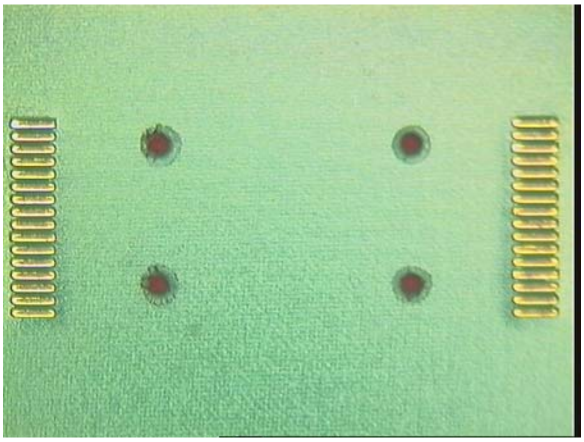


Figure 27 : Paste and adhesive for TSOP component

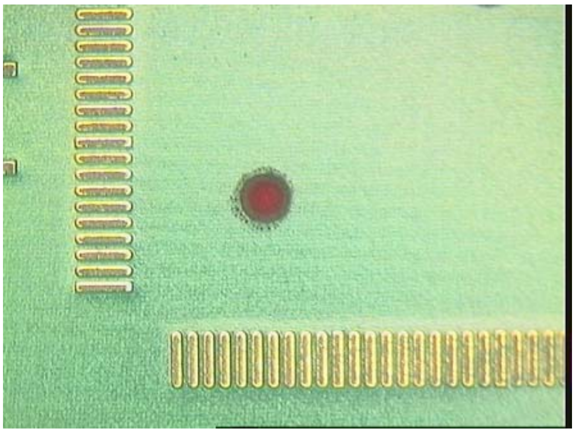


Figure 28: Paste and adhesive for QFP component

With smaller components, adhesive printing was more problematic. Figures 29 & 30 illustrate the irregular shaped dots achieved, and also the contamination of the solder paste on the pads with adhesive. Printing of adhesive for R0603 components was not possible.

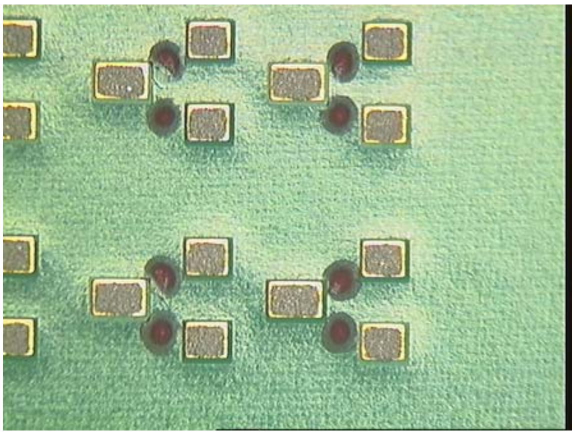


Figure 29: Paste and adhesive for SOT323 component

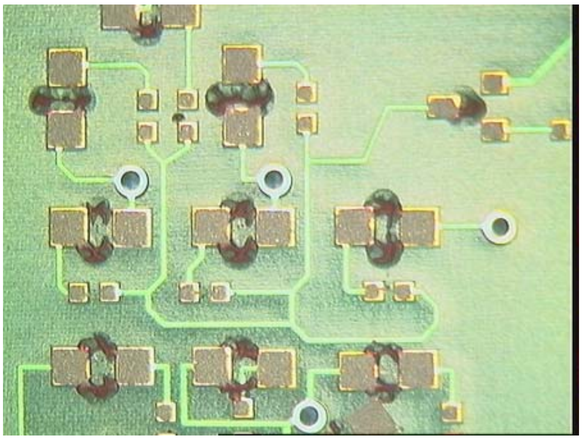


Figure 30: Paste and adhesive for chip LED, SOT23 and R0603 components

Table 4 below lists the aperture sizes used for adhesive on this assembly. From this list and Figures 25-30, it is evident that it was not possible to print apertures below 0.6mm diameter with a 3mm relief stencil and this adhesive. However, with multiple manual strokes and a squeegee 0.5mm apertures could be printed. A slightly thinner stencil would have helped this problem.

Table 4: Aperture diameters used for printing adhesive under SDSRS components

Chip LED	0.3 and 0.4mm
R0603	0.3 mm
R0805	0.3 mm
Tantalums	0.3 mm
C1206	0.3 mm
SOT23	0.5 mm
μBGA	0.5 mm
SOIC	1.2 mm
SOT223	1.2 mm
TSOP	1.2 mm
Battery clip	2.0 mm
PBGA	2.0 mm
QFP160	2.0 mm

6.4. SDSRS Reflow Soldering

Both sides of the assembly were reflowed together during a single pass through the reflow soldering oven. Generally the resulting joint quality was acceptable with the majority of the components, as demonstrated in Figures 31-35. All the active parts of the circuits were functional after soldering with the exception of some chip LEDs, which were not placed accurately during manual placement. Some of the solder joints on side A did exhibit limited solder balling as a result of the solder paste being subjected to 30 mins at 75^{o}C (to partially cure the adhesive), but many commercial users would consider this acceptable. The worst solder balling was evident behind the heel of the SOIC lead seen in Figure 32. Such solder balling could be eliminated using a UV curable adhesive. But the relief stencil design would have to be changed so that the adhesive dots were placed on the edges of the components, rather than underneath, to ensure that their cure could be initiated by UV.

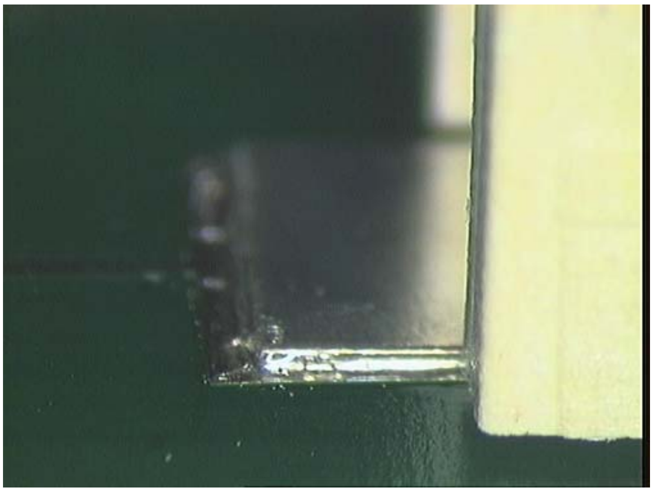


Figure 31: Reflowed solder joint on battery connector on side A

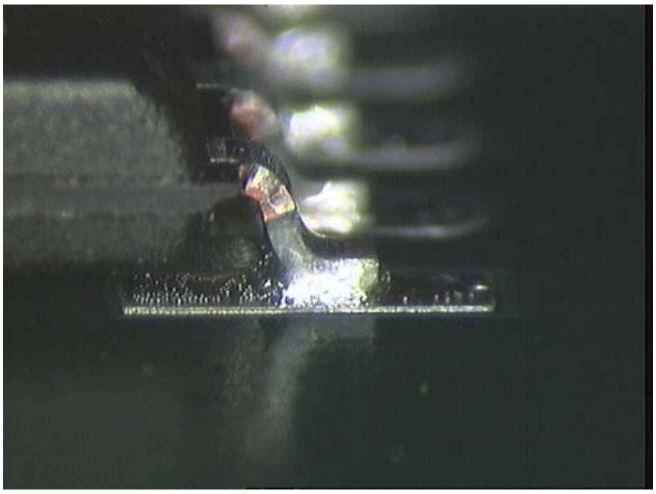


Figure 32: Reflowed solder joint on SOIC on side A

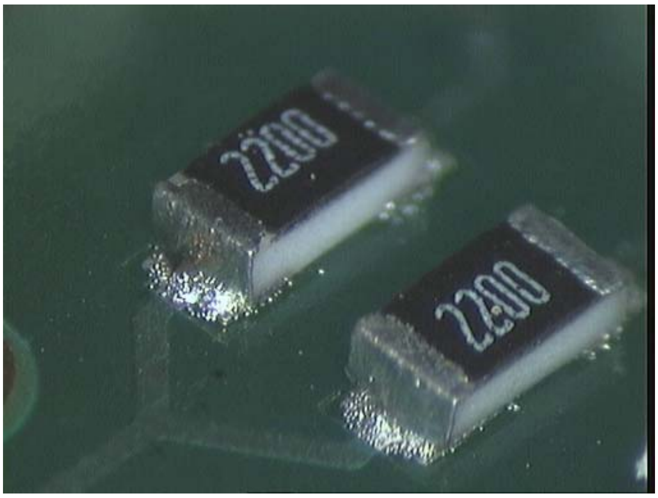


Figure 33: Reflowed solder joint on R0603s on side A

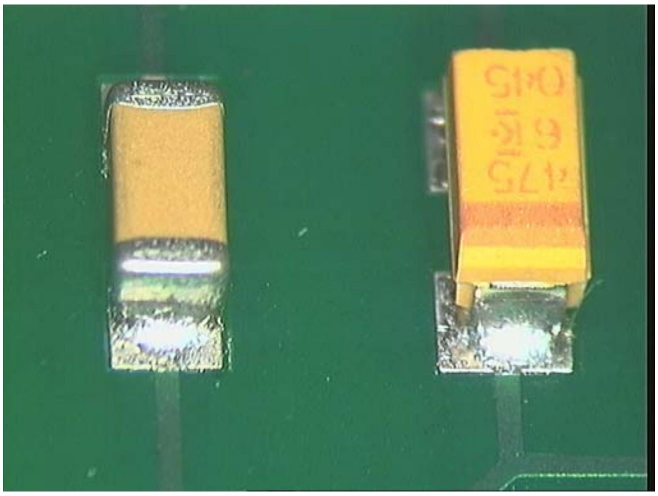


Figure 34: Reflowed solder joints on C1206 and tantalum on side A

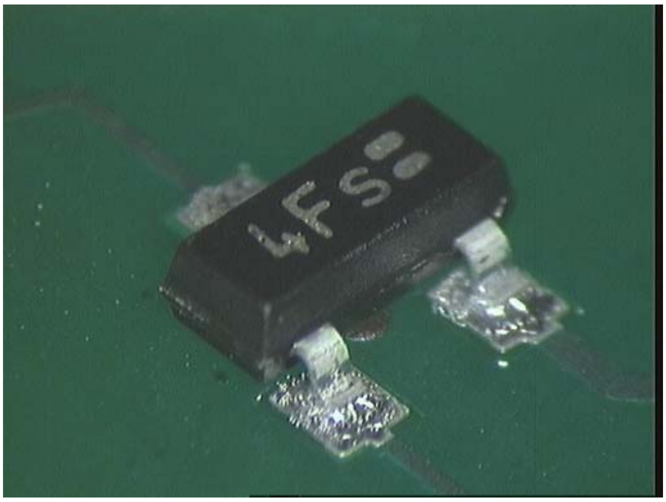


Figure 35: Reflowed solder joint on SOT23 on side A

The only components poorly soldered were the µBGA and the PBGA. In all cases these fell off the underside (side A) during the paste printing of side B or the subsequent reflow operation. The PBGA did not remain attached because the adhesive dot height was insufficient to bridge the gap between the underside of the component and the PCB. Possibly for the PBGA increasing the aperture size and hence the dot height would have helped, especially since this was a peripheral array and there was sufficient room under the component. The µBGA did not remain attached since the adhesive dots were poorly printed due to (a) the 0.5mm aperture used, and (b) the fact that they needed to be printed on the perimeter, since they were full array solder components. The small 0.5mm aperture design was selected, since this provided the closest centre point of a dot to the component. Generally, it is advisable to use the thinnest stencil compatible with the rest of the process. The only reason to increase stencil thickness is to accommodate particular features; otherwise a thinner stencil will give better printing.

An attempt was made to reflow side A upside down without the use of adhesive. Side A was printed with solder paste as before and then manually populated and inverted prior to reflow soldering. Table 5 shows the results of this trial. Components as large as SOIC14 were successfully reflowed. However, TSOP32, QFP160, BGA, µBGA and battery clips did not remain attached to the assembly.

Table 5:
Results for reflow of components inverted without aid of adhesive

Component type	Successfully reflowed inverted without adhesive
R0603	Yes
C1206	Yes
QFP160	No
Battery clip	No
PBGA256	No
SOT23	Yes
SOIC8	Yes
µBGA	No
LED	Yes
SOIC14	Yes
Tantalum Cap	Yes
TSOP32	No
SOT223	No (75%)

6.5. SDSRS Summary

- The process worked well with larger aperture designs
- The thinnest relief stencil compatible with the process and material is recommended

- Stencil printing of the three stages was straightforward
- A number of board and design issues arise that are not normally encountered
- Sometimes the requirements of SDSRS are at the manufacturing limit of polymer stencils
- Cleaning glue from polymer stencils can be problematic, because of high aspect ratio apertures

7. CONCLUSIONS

7.1. Solder Jetting

The review of solder jetting undertaken has highlighted a number of areas, which could potentially prove successful niche applications. The majority of these are low volume applications, which could avail themselves of the direct write capability of the technology, negating the need for costly masking or photo-imaging techniques. Such niche applications could include application of solder to substrates for select deposition for DCA applications or via fill. Wafer bumping in low volume for specialist devices such as ASICs, as well as chip scale package bumping, could also benefit from the direct write capabilities of the process

7.2. Relief Printing

Relief printing of adhesives has been successfully demonstrated with the manufacture of simultaneous double-sided reflow soldered assemblies, but it is evident that there are some limits to the current technology. With the 3mm stencil used, it is clear that for aperture diameters greater than 0.6mm, adhesive was printed much more successfully than with apertures below 0.6mm. With solder paste, it proved difficult to print with apertures less than 1.2mm in diameter and this was because of high cohesive forces compared to adhesive forces. Thus when designing assemblies to utilise this process, the designer may find that solder paste and adhesives deposition will have to be limited to the larger components or coarser pitch components where large stencil apertures can be used. However, the technique should be eminently suitable for printing larger area deposits, such as thermal adhesive under perimeter arrays (QFPs), or die-bond adhesive. The technique also looks like finding commercial exploitation in printing of 3D substrates, although only with coarser pitch SM components.

In all potential applications, relief printing could prove to be a viable alternative to needle dispensing should the end user not wish to invest in needle dispensing, or require a higher application rate than is available through current dispensing technology.

8. ACKNOWLEDGEMENTS

The work was carried out as part of a project in the Measurements for the Processability of Materials (MPM7.4) Programme of the UK Department of Trade and Industry. The authors are grateful for many useful discussions with R. Willis of EPS on SDSRS.

The authors are especially grateful to DEK for the loan of a DEK 265 GSX printer and plastic stencils, and to Henkel Loctite for materials.

9. REFERENCES

(1) Application of Eutectic Solder Using Solder Jetting Technology: Rick Godin, MPM Corporation, Franklin, MA : p739 –744

(2) Recent Advances in Highly Controlled Molten Metal Droplet Formation From Capillary Stream Break-Up with Applications to Advances Manufacturing: Melissa Orme and Anthony Bright: 2000 TMS Annual Meeting, March 16-20, 2000 Nashville TN

(3) Microjet Printing of Solder and Polymers for Multichip Modules and Chip-Scale Packages: Donald J Hayes, David B Wallace and W Royall Cox : IMAPS '99

(4) Solder Jet Printing For Low Cost Wafer Bumping: Donald J Hayes and David B Wallace: http ://www.microfab.com/papers/ishm96/ishm96.htm

(5) Solder Jet Technology Update: David B Wallace and Donald J Hayes : Proceedings, ISHM 97

(6) http://www.emtmagazine.co.uk/techfeatures/200111%20November/technicalarticle_Case_Study_DEK.htm

(7) A Novel Measurement Technique For Stencil Printed Solder Paste: M. Dusek and C. P Hunt, NPL Report MATC(A) 08

(8) Characterising Stencil Printing of Surface Mount and Conductive Adhesives: L Zou, M Dusek, M Wickham and C. P Hunt, NPL Report MATC(A) 55

(9) Compatibility of Lead-free Solders with PCB Materials: Milos Dusek, Jaspal Nottay and Christopher Hunt, NPL Report MATC (A) 89 :