MYDATA®

White Paper

Tackling SMT enemy number one

Raising the standard of solder paste application

Is screen printing technology able to keep pace with rising quality demands and increasingly complex board layouts? Or, is new jet printing technology ready to fill the gap? A comparison study between the two methods reveals some interesting differences. Screen printers offer some possibilities for optimizing solder paste deposits, but optimization is far easier and quicker with the jet printer. At the same time, the ability to print individualized deposits on every single pcb pad may be the ultimate answer to the growing quality challenge.

A challenging future

The role of the solder joint is all important in ensuring the final quality of any printed circuit board assembly. As well as providing a robust electrical interconnection it also needs to ensure a reliable mechanical interconnection. The application of solder paste is key to ensuring high quality solder joints but it is affected by a great many parameters. Currently, printing issues are the most commonly cited cause of error during the surface mount assembly process and account for something like 70% of all soldering errors – mainly opens, insufficients and bridges.

Industry trends are making the situation ever more challenging. Boards are becoming denser, use smaller and finer pitch components and require small and large components to be placed very close together. Modern manufacturers are faced with more complex production, higher quality demands and the need to maximize automation in order to improve efficiency and to remain price competitive. In today’s ultra-competitive environment there is no longer any place for errors!

Automated screen printing

Screen printing with a squeegee blade and metal foil stencil represents the standard method of depositing solder paste in surface mount assembly operations. Despite advances in automation and increasingly sophisticated solutions stencil printing remains a key area of concern. What causes that concern is the sensitivity of the printing process and the fact that so many parameters contribute to the final result. Parameters include: printing speed; squeegee type; angle and pressure; gasketing between the stencil and pcb; separation speed; underside wiping of the stencil; pcb support (especially for second-side printing); and stencil thickness and aperture design. Each of these parameters needs to be carefully optimized to ensure quality. And a new set of parameters needs to be assigned for each new job that is undertaken.

One of the limitations of stencil screen printing is the fact that the volume deposited is, to a large extent, determined by stencil thickness. While stepped stencils go some way to overcoming this issue they also add further costs and complexity. They also mean certain restrictions are imposed on pcb design in order to maintain 'keep-out distances' (the distance required between an aperture in a 'stepped' area and the nearest 'normal' area).

By far the most common situation is for manufacturers to use regular metal foil stencils in their production.

Automated jet printing

Automated jet printing is a relatively new technique that uses a unique ejector mechanism to deposit solder paste onto printed circuit boards at high speed. The non contact printing technology applies no force to the pcb and builds up solder paste deposits in three dimensions (i.e. solder paste droplets can be deposited on top of each other). The process is completely software controlled and default settings are provided for each component based on CAD data. However, the user has the freedom to fine-tune the volume, area coverage, height and layers of solder paste for each individual pad, component or package.

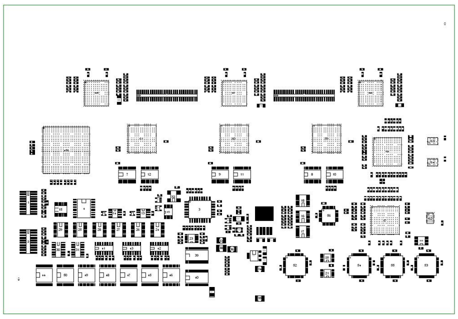
The stencil-free technology provides much faster response times compared to screen printing. There is no lost time ordering, waiting for or cleaning stencils and, as print programs are prepared off-line, setup and changeover times are also minimized. In addition, layout changes or print adjustments are extremely quick and simple to implement.

Fully automated jet printing technology has been developed by MYDATA automation AB, Sweden. The first jet printer (MY500) was launched in 2007 and a new generation of the product was introduced in 2008.

So, how does the quality of the jet printer stand up compared to screen printing in a real life production environment?

Head-to-head study using live production boards

A study to compare the two technologies was carried out at a major European electronics contract manufacturer. The company is a leader in high-tech electronics production and is ISO 9001-2000 & 14001, AQAP 2120-2003, IEC 61508 and TL 9000 certified, with customers coming from the worlds of healthcare, telecommunications, aviation and defence.



An example of one of the customer board layouts that was used in the study.

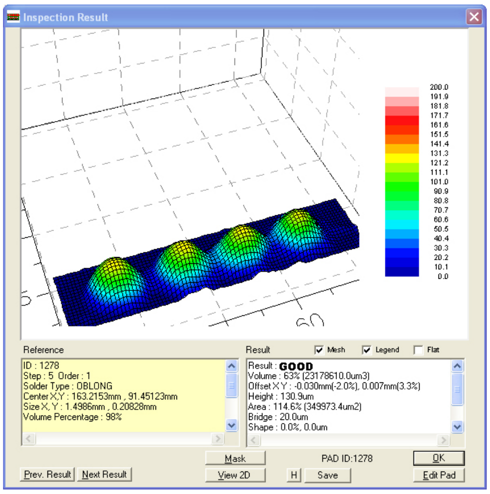
The study lasted for one week, measuring solder paste deposits on customer boards that were produced as part of the company's standard operations. 12 batches were included in the study with approximately 14 boards per batch. Half of each batch was run on a screen printer and the other half on the jet printer. (Note, two of the batches were run on the jet printer only.) Three different MPM screen printers were used along with three different stencil thicknesses.

		JET		STENCIL			
BOARD #	SURFACE	PCB QUANTITY	SOLDER PASTE	PCB QUANTITY	SOLDER PASTE	STN. THICKNESS (µm)	PRINTER
1	Gold-nickel	6	Senju M705-LFAC19	7	Alpha OM-338-T	120	1
2	HASL	8	Senju 2062-AC19F13	7	Alpha OM-5100	150	2
3	HASL	3	Senju 2062-AC19F13	0	n/a	n/a	n/a
4	Gold-nickel	5	Senju M705-LFAC19	0	n/a	n/a	n/a
5	HASL	7	Senju 2062-AC19F13	7	Alpha OM-5100	150	2
6	HASL	7	Senju 2062-AC19F13	7	Alpha OM-5100	150	2
7	HASL	7	Senju M705-LFAC19	7	Alpha OM-338-T	125	2
8	HASL	7	Senju 2062-AC19F13	7	Alpha OM-5100	150	1
9	HASL	7	Senju 2062-AC19F13	7	Alpha OM-5100	150	2
10	Gold-nickel	7	Senju 2062-AC19F13	7	Alpha OM-5100	150	1
11	HASL	7	Senju 2062-AC19F13	7	Alpha OM-5100	150	3
12	HASL	7	Senju 2062-AC19F13	7	Alpha OM-5100	125	2

Breakdown of boards produced during the study. (Note, where necessary, results from the study were weighted in order to compensate for different pcb quantities within certain batches.)

PRINTER
1 = MPM Accuflex
2 = MPM Ultraprint 1500
3 = MPM Ultraprint 2000

After printing, the boards were taken off-line and inspected using a Koh Young Technology three dimensional solder paste volume measurement instrument. All measurements were carried out on site, after which the boards were placed back into the production process. The manufacturer also carried out its usual production and quality checks, both prior to and after component assembly, according to its own stringent quality control processes.



Koh Young Technology is a world leader in 3-D measurement and inspection, offering 100% 3-D Paste Measurement based on the patented Shadow-Free measurement technology .

The study generated measurements from more than 100,000 solder paste deposits. Analysis of the results focused on five key attributes of solder paste application: Overall Quality; Volumes; Repeatability; Predictability; and Accuracy.

Overall Quality – Good to go!

As would be expected from a competent and reliable manufacturer, all of the boards were of a high quality. The manufacturer's stringent quality control processes, based on IPC standards, were followed and any minor errors were picked up and corrected as part of the normal production process. As such, all boards received quality approval and were delivered to their respective customers.

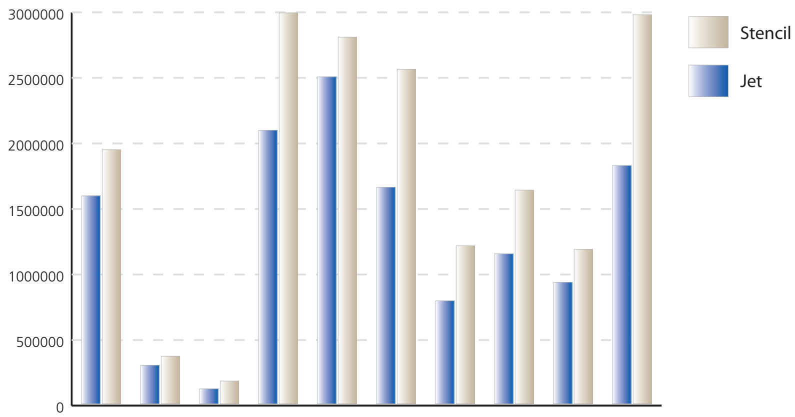
So, if all boards were of acceptable quality, what, if any, were the differences between the two printing methods?

Volume – Compromise versus individualize

Probably the main difference between the two technologies was in terms of the volumes of solder paste deposited. Screen printing with a standard stencil always requires a compromise between the smaller components that need less paste and the larger ones that need more. The tendency is to choose stencil thickness based on the needs of key components. At the same time it is important to ensure that there is always enough paste on every pad to avoid insufficients or open joints. Invariably, this means that more paste is applied than is necessarily needed.

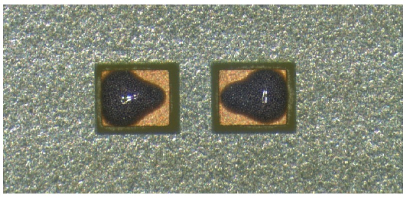
Jet printing, on the other hand, allows volumes to be individualized for every single pad. This means that solder paste application can be optimized for each component and there is never any need to apply more paste than necessary.

SOLDER PASTE VOLUME (nanoliter, nl)



A comparison of the total solder paste volume used for printing each batch. On average the jet printer used just 65% of the volume that the screen printers used.

The difference can be seen in the study results which showed that for all batches the jet printer used less volume. For one particular board the jet printer required just 55% of the paste volume that the screen printer required. Comparing the average volume across all ten different boards, the jet printer required 65% of the paste volume that the screen printers required.



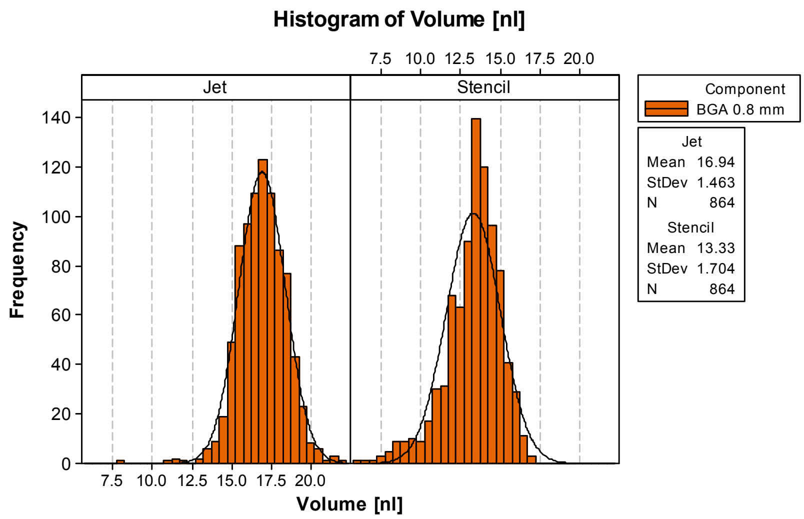
Example of triangle-shaped deposits achieved with the jet printer.

In addition to volume, jet printing also allows printing patterns to be individualized on every pad. Users have the flexibility to adjust the size, position, pattern and height of solder paste. This means deposits can be optimized in order to avoid joint errors or enhance joint functionality. For example, allowing total freedom of heatsink deposit design, ensuring good wetting through proper pad coverage, or allowing triangle-shaped deposits in order to avoid mid ship solder balls.

With each technology being used to produce 70 boards, total volumes translated into 83g of solder paste used by the screen printers and 54g used by the jet printer. However, this does not take into account the additional paste consumed when screen printing, i.e. through spillages and cleaning etc., which the jet printer avoids through the use of solder paste cassettes that simply snap into place.

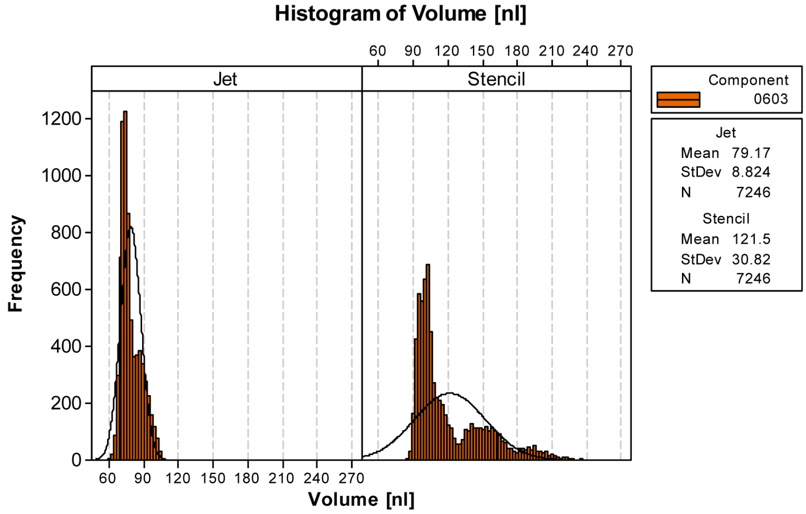
Repeatability – Achieving the same results time and time again

Looking at solder paste volumes for specific components across all batches raised an interesting difference between the two technologies. For key components, such as fine-pitch BGAs and QFPs, the results for the two methods were very similar. When screen printing, it is likely that these types of difficult components determine the choice of stencil thickness, thereby, ensuring consistent results. In fact, when considering all components overall, the technologies achieved relatively similar results – even though standard deviation with the jet printer was lower for 10 out of the 12 batches.

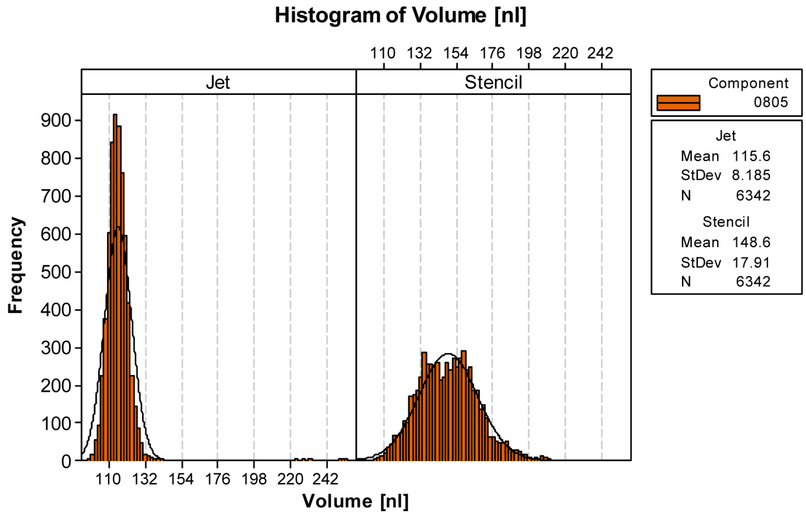


The two technologies achieved similar repeatability results for key components such as BGAs.

However, when it came to some of the smaller components, the screen printers showed less consistency. For example, the histogram for 0603s included three peaks, while the one for 0805s had two distinctive peaks. The explanation for this is that these components were included in different batches that used stencils of different thicknesses, i.e. the optimal volumes for these chips were compromised in order to meet the optimal requirements of other components on the same board.



For 0603s three separate peaks are clearly discernable when screen printing with three different stencil thicknesses, while jet printing achieved very consistent results or this component.



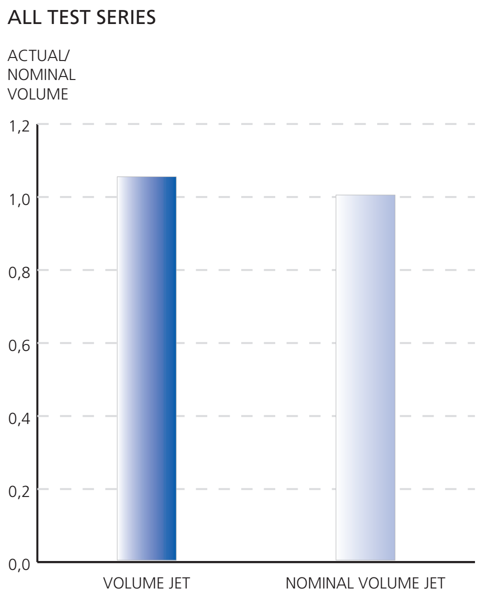
The screen printers produced rather inconsistent results for 0805s. The optimal volume for this component have been compromised by the use of two different stencil thicknesses.

With the jet printer it does not matter what the component mix on a board is as volumes are always optimized for each individual component. So, even with the small components, the jet printer achieved consistent results for repeatability across all batches.

Predictability – Does theory match reality?

When comparing the actual volume of solder paste used to the amount expected to be used, the screen printer results were reasonably accurate, despite the many parameters involved.

The jet printer, on the other hand, achieved truly excellent results with the actual volume being extremely close to the nominal value. Several factors contribute to this including the fact that there is less operator intervention with the jet printer, i.e. volumes are set during programming, after which solder paste feeding is completely software controlled. It is also a closed system, so all of the paste jetted ends up on the pcb. In addition, as jetting is a non contact technique it is not affected by warped boards.

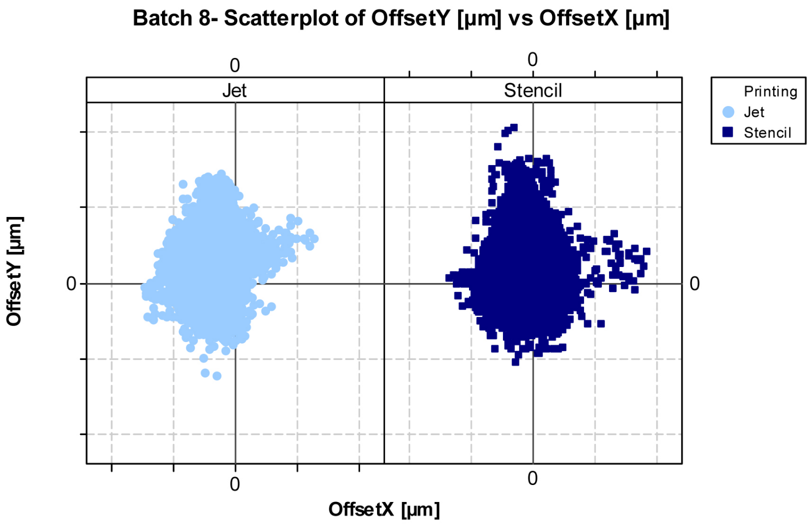


The expected volume required by the jet printer matched the actual volume used extremely closely.

In the case of screen printing, both manual and technical issues affect the actual amount of solder paste used. For example, stencil quality, pcb support, squeegee pressure and the solder mask can all affect printing volumes. Also, paste is prone to sticking in apertures when the stencil is lifted, particularly with fine pitch applications. More frequent cleaning of stencils and/or vibrating stencils prior to lifting may be necessary, with the associated risks of manual adjustments and errors.

Accuracy – Hitting the right spot

Deposit accuracy is obviously important in order to ensure good solder joint quality. In the production study the two technologies achieved very similar results.



The non contact principle, along with the use of board fiducials (reference markers on the pcb used by the jet printer's imaging system), mean that the accuracy of the jet printer is not affected by aspects of board quality. Accuracy is the result of software-controlled x-y movements of the jetting mechanism.

Screen printing accuracy, on the other hand, is directly affected by pcb or panel quality, including how the board is cut, stretching, warpage and solder mask quality etc. Furthermore, stencil quality and stencil alignment are two additional parameters that are very important in determining screen printing accuracy.

Conclusions

Each component type has its own individual characteristics such as lead geometry, land pattern and weight that set different requirements for solder paste deposits. Achieving the correct solder paste volume and coverage per pad is essential for high quality solder joints across the pcb.

While screen printing using metal foil stencils has served the electronics industry well, it remains a cause for concern. As shown in this study, it is possible to achieve good quality screen printing, although certain compromises are always necessary. Also, due to the many parameters involved, optimizing conditions for a specific job can be very time consuming and requires a good deal of operator experience. As with any process, the more parameters and steps involved the harder it is to ensure consistency. At the same time, the more operator involvement there is, the higher the chance of introducing errors.

In contrast, MYDATA's MY500 jet printer is a touchless, closed system that greatly reduces the number of parameters involved. As illustrated in the live production study, the jet printer matched or bettered the high quality screen printing in all aspects.

With full software control the jet printer allows far greater scope for optimizing solder paste application. Operators have complete flexibility to fine-tune the volume, area, pattern, height and layers of solder paste for every individual pad, component or package on the pcb.

Having easy access to this level of control and flexibility is likely to become increasingly desirable to manufacturers as the industry continues to turn to smaller and finer pitch components on denser, high mix boards.