

Solder Paste Jet Printing

A New Approach to Solder Paste Application



A big challenge in modern electronics manufacturing is achieving the right amount of solder paste for each component.

For many years, stencil printing has been the standard method of depositing solder paste in surface mount assembly operations. It has served the Surface Mount Technology (SMT) world well, but there have always been acknowledged difficulties.

One big challenge in modern electronics manufacturing is the huge diversity of size among components and hence achieving the right amount of solder paste for each component.

Another challenge is how to produce prototypes without disrupting series production already running in the line. Product changeovers require adjustments in the printing process. Although the actual printing process is fast, product changeover routines make obvious the lack of flexibility of the stencil printing process for high mix production.

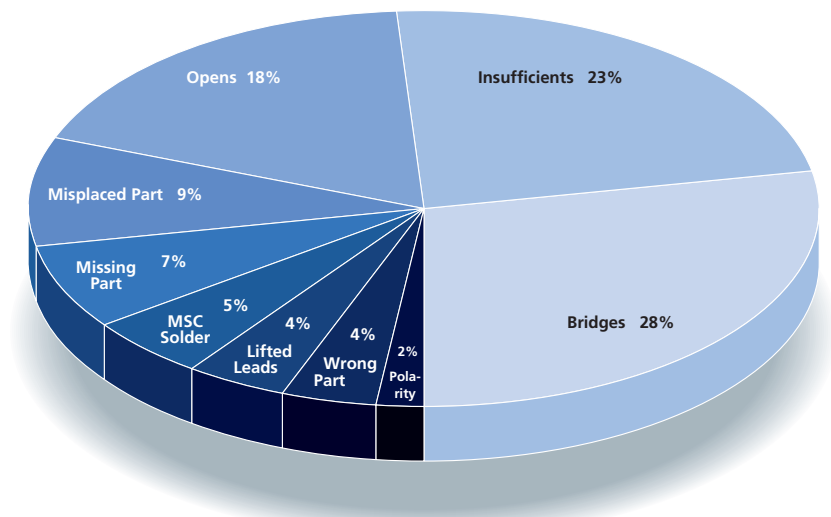
The MY500 jet printer addresses these challenges and provides a superior solution. This white paper introduces jet-printing technology and describes its unique benefits and its components. In addition, the results from verification tests are presented.

The Stencil Problem: The Need for More Flexibility

Although metal stencil solder paste printing is a mature process, it remains the part of the assembly process that has the biggest impact on the soldering result.

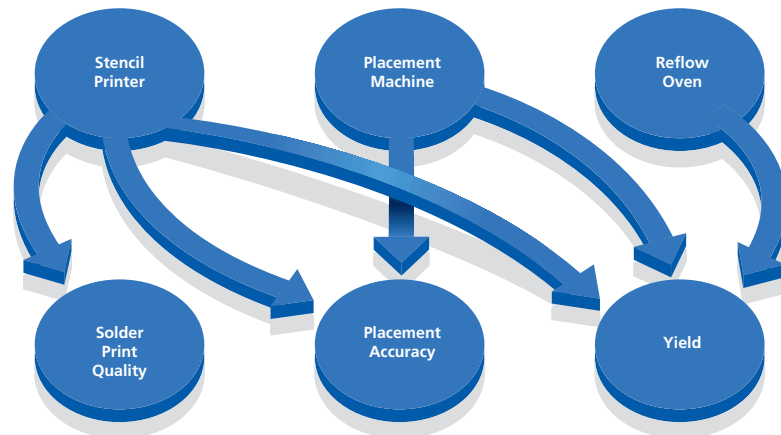
Solder paste depositing is regarded as the biggest source of soldering errors.

In a well adjusted process, stencil printing is often regarded as the source for approximately 70% of soldering errors¹.



Solder errors found after reflow, of which insufficient solder, open joints and bridges most often depend on the result of the stencil printing process¹.

¹ <http://www.kohyoung.com/> ♦ Yield improvement



Each individual assembly processes impacts on subsequent processes².

In stencil printing with a squeegee blade and a metal foil stencil, there are several parameters that influence the printing result and hence the soldering result including:

- Print speed
- Squeegee pressure
- Angle of the squeegee
- Gasketing ('sealing') between the stencil and the PCB
- Separation speed between the PCB and the stencil
- Support of the PCB, especially when printing the second side on double-sided PCBs
- Stencil thickness and aperture design.

Each parameter influences the amount of solder paste deposited on the PCB and if each parameter is not fully optimized, the soldering result is impacted. Parameter optimizing can take considerable time away from production and when changeovers occur often, it can create a bottleneck to flexible production.

Stencil Manufacturing and Lead Time

There are three methods of producing stencils depending on cost and technology requirements. These methods are chemical etching, laser cutting and electroforming. Each PCB design and design iteration requires a new stencil, thus stencil costs can rise with multiple iterations.

A damaged stencil will result in production delay as producing and delivering a new stencil takes a lead time of at least one or two days.

Compromise Using Different Solder Paste Amount

An ordinary stencil with the same thickness all over the printing area will always be a compromise between the smaller components that need less paste and the larger ones that need more paste. This also has an impact on the soldering result.

Producing and delivering stencils might take up valuable production time.

² Kamen, E. et al. Analysis of Factors that Affect Yield in SMT Assembly, SMT Express Vol.2, Issue 4, 2000.

This leads to the apertures being too small, resulting in insufficient deposits of solder paste and sometime not enough flux. The latter is particularly a problem on CSPs with small diameter bumps and narrow pitch, which can lead to inferior or even open joints. This defect is often very hard to detect even using tilted view X-ray inspection.

Keep-out Distance Imposes Restrictions for Stepped Stencils

Stepped stencils are a way of differentiating the stencil thickness and hence of depositing more or less paste depending on the component size. Stepped stencils will always be more expensive than ordinary ones because they require several manufacturing steps.

Different volume on PCBs are possible with stepped stencils but they impose PCB design restrictions.

The distance between a step-up (more paste) and step down (less paste) edge area and an aperture in the surrounding 'normal' stencil thickness on a stepped stencil is referred to as the 'keep-out distance'. According to IPC-7525A, Stencil Design Guidelines, the minimum keep-out distance between an aperture in a step-up area and the step-up edge is 0.65 mm; while the minimum keep-out distance between the step-up edge and the nearest aperture is 0.9 mm for every 0.025 mm of step height.

As a rule of thumb, the keep-out distance should be 36 times the step height. This implies that a step height of 0.05 mm requires a keep-out distance of 1.8 mm. Adding the two keep-out distances, there needs to be approximately 2.5 mm between an aperture in a stepped area and the nearest 'normal' aperture.

If a stepped stencil is used but the PCB is not designed for using stepped stencils, it may be difficult to maintain the keep-out distances required under IPC-7525A.

Underside Wiping and Gasketing

Another important parameter in stencil printing is the gasketing between the stencil and the PCB. Any difference in height on the board, such as a distorted PCB or labels, jumper wires, etc., can lead to poor gasketing. This will result in smearing of the solder paste out onto the PCB, which in turn can result in solder balling or even bridging.

Poor gasketing can result in solder balling or bridging.

To reduce the problem of smearing, the underside of the stencil is wiped at certain intervals, for example, after every fifth print cycle. Severe smearing decreases the number of print cycles between underside wiping, prolonging the cycle time and increasing the consumption of wiping paper.

Handling of Stencils

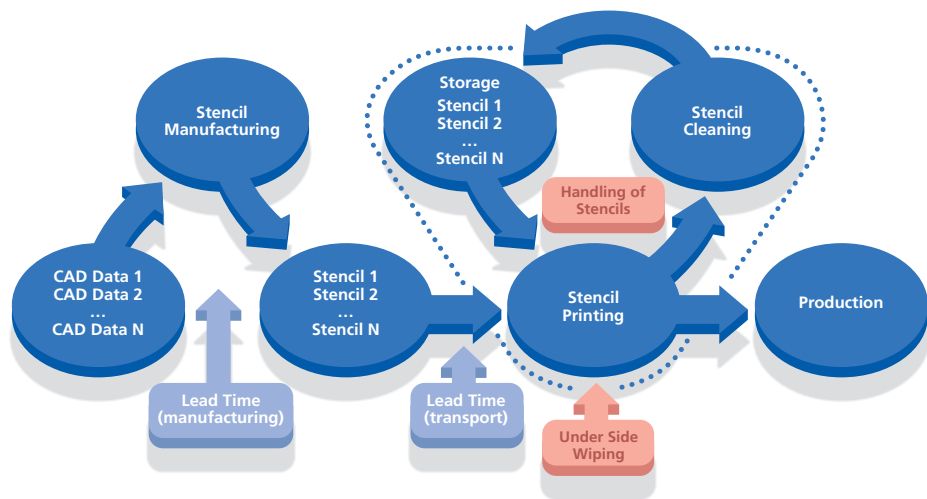
Stencils need to be kept clean and in an undamaged condition. During production, the cleaning is needed to ensure appropriate solder paste deposits. Modern printers are able to perform the cleaning automati-

Stencils require cleaning and careful handling.

cally, but they still need to be reloaded manually with cleaning paper or agents. Prior to storing, stencils have to be cleaned carefully. Thus, the handling of the entire cleaning process requires resources such as:

- Cleaning agents and paper
- Cleaning machines
- Manpower.

This may not sound important until you consider that the more steps there are in a process, the higher the risk of error. It must be said that a good printer operator is very unlikely to make mistakes, but the final result is still operator dependent.



The process of depositing solder paste by metal stencil printing.

The Solution: Stencil-Free Jet Printing

Jet printing technology has been developed to meet the demand for greater flexibility in modern electronics production without loss of throughput speed. To achieve flexibility and speed at the same time, the technology utilizes stencil-free printing and high-speed solder paste depositing using the jet printing method. The technology could be compared to printing a document on an inkjet printer. In other words, this means that the CAD data (or Gerber data) for a particular PCB is compiled off-line and then sent to a jet printer for printing as many "copies" as required.

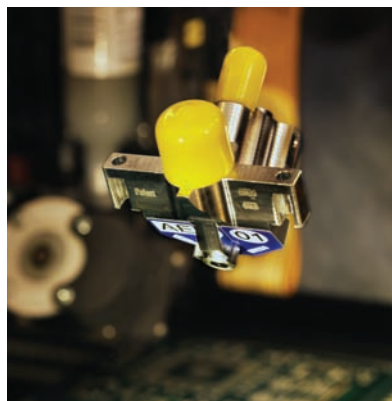
Jet printing brings together flexibility and speed.



Jet Printing – High-Speed Solder Paste Depositing

High-speed solder paste depositing by jet printing is made possible by a unique ejection method. The technique is to eject tiny droplets of solder paste from a cartridge through an ejector mechanism onto the printed circuit board at the positions required by the board layout. The ejector system has been tested for speeds up to 500 droplets per second, which enables solder paste printing on the fly. High-speed solder jet printing involves accelerations up to 3Gs, which demands a very robust design. The MY500 is therefore constructed out of cast stone.

High-speed depositing is possible due to the unique ejector system.



Solder Paste Cartridge

The solder paste used for jet printing is delivered in standard 30 cc Iwashita cartridges containing specially developed solder paste. To develop these jet printing solder pastes, MYDATA initially teamed up with several solder paste suppliers, including NIHON ALMIT Co., Ltd., and Senju Metal Industry Co., Ltd.

Other solder paste manufacturers are also developing solder paste suitable for jet printing.

The cartridge is loaded into a lightweight cassette that is swiftly snapped into the machine. This allows the cassette to be reloaded in a few seconds, so you can switch from tin/lead to lead-free solder paste in a matter of moments.

Barcodes and ID chips eliminate the risk of the wrong paste being used by mistake.

A barcode label on the solder paste cartridge and identification chip in the cassette ensure that the wrong type of solder paste, or solder paste past its due date, is never loaded by mistake. The electronic identification barcode label and cassette memory make machine settings automatic. Once the paste code type is entered, the printing can start.

Touchless Depositing Process

Touchless Jet Printing Technology applies no force to the PCB, hence no support pins are needed. In addition, the print program automatically aligns and adjusts to the board stretch based on the fiducial

marks. Temperature control ensure that proper viscosity of the solder paste can be maintained at all times – which in turn leads to higher levels of application accuracy.

Software Controlled

Software allows control of the deposits – in 3D.

Since the process is completely controlled by software, the solder paste volumes can be altered ‘on-demand’. The MY500 jet printing process allows control of solder paste deposits with precision – in three dimensions (3D). It is possible to fine-tune the volume, area coverage, height and layers of solder paste that need to be applied for every individual pad, component and package. If adjustments are needed, or a customer order is revised, the CAD data can easily be changed.

The MY500 can print pads for components with pitches as small as 0.4 mm (16 mil). With this level of control, you can print small deposits next to large ones, e.g. 0201s right next to connectors.

Operator-Independent Process: Improving Quality

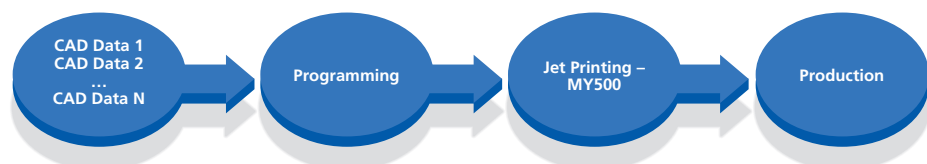
The jet printing process is operator-independent, hence increasing its robustness.

The number of variables (squeegee pressure and speed, separation speed etc.) that can influence the printing result and hence the soldering result is dramatically reduced with Jet Printing Technology compared to metal stencil printing.

For PCB assembly shops, as in all high-tech industries, the less on-line operator involvement needed, the easier it is to achieve and consistently maintain higher levels of quality. For convenience, and to minimize the risk of error, with Jet Printing Technology any operator can run print programs approved for production. To maintain a high degree of flexibility, MY500 printing programs can easily be adjusted at the off-line station if revisions are required.

Reduced number of parameters affecting the final soldering result.

A reduction in the number of parameters and/or process steps, in general, improves reliability. The number of parameters affecting the printing result for stencil solder paste printing, including the manufacturing of the stencil, is in the range of ten different operations. For jet printing, the number of parameters is reduced to two or three. For every parameter removed, the robustness of the process increases.



The process of depositing solder paste by jet printing.

Jet Printing and the Soldering Result

To verify the soldering result after jet printing, three test series were performed. In each test series, a specific type of PCB and specific types of components were used. The components used were as follows:

- QFP with 0.4 mm pitch
- QFP with 0.5 mm pitch
- BGA with SAC305 bumps and 0.5 mm pitch

The components selected are typical packages found on printed board assemblies in most of electronic devices. These components are considered to be demanding in the solder printing process. Hence a good final result would confirm the printing quality of the MY500 Jet Printer as well.

The result was verified after every process step: jet printing, component placement, and soldering. The most important analysis was the volume measurement after jet printing and solder joint appearance after soldering.

Test Method

100,000 solder joints were chosen as the starting point to achieve a proper statistical evaluation of the results. Closer evaluation using the statistical software Minitab³ showed that the confidence interval for statistical calculations was marginal down to 88,000 solder joints.

With regard to the required number of solder joints and the number of leads or bumps per component (QFP128 and BGA132), a total of 20 boards were produced for each component type (see table below).

The PCBs were designed so that the components could be placed in an array of six rows and six columns, i.e., 36 components on each board.

Number of soldered joints				
Component	No. of leads/ bumps	Components on each board	No. of boards	Solder joints in total
QFP 0,4	128	36	20	92160
QFP 0,5	128	36	20	92160
BGA 0,5	132	36	20	95040

The size of the PCBs was 285 mm x 263 mm with a thickness of 1.6 mm with immersion tin metallization. The boards were produced by Multi-Cad AB, in Sweden.

The test series performed was as follows:

1. Depositing of solder paste with a MY500 Jet Printer
2. Placement of components with a MY9 pick & place equipment
3. Reflow in a Heller oven
4. Inspection of QFP and BGA solder joints by AOI and X-ray, respectively.

³ www.minitab.com/

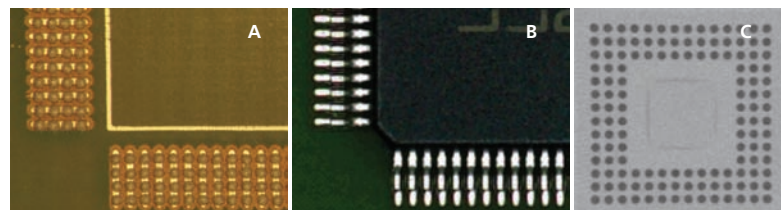
The printing, placement and soldering and part of the inspection were done at MYDATA in Bromma, Sweden. Optical inspection of the QFP boards was done by a Marantz AOI at Frontside in Mölnlycke, Sweden. X-rays of the BGA joints were done at Saab Microwave Systems AB in Mölndal, Sweden.

Test Results

During inspection of the 279,360 solder joints, no solder bridges or open joints were found.

The results show that a MY500 Jet Printer offers a stable process that facilitates high quality in the finished product. The inspection results are summarized in the table below.

Summary of the inspection results				
Component	Inspection method	Solder joints in total	Solder bridges	Open joints
QFP 0.4 pitch	AOI	92160	0	0
QFP 0.5 pitch	AOI	92160	0	0
BGA 0.5 pitch	X-Ray	95040	0	0



Samples from the tests. A) Jet printed deposits on QFP with 0,4 mm pitch. B) Soldered joints on QFP with 0,4 mm pitch. C) X-rayed BGA with 0,4 mm pitch.

Additional benefits of jet printing

With metal stencil printing, a product changeover involves the removal of solder paste from the old stencil, removal of the old stencil, loading of a new stencil and new paste applied to the stencil followed by some kind of calibration between the stencil and PCB.

As there is no need for stencils with Jet Printing Technology, the MY500 allows off-line programming for changeovers to be accomplished in seconds. So every production run can be started and completed as efficiently as possible.

The MY500 uses the same CAD or Gerber file data as is used for the pick and place machine. The machine preparation software uses that data to automatically generate a 'virtual stencil'.

An 'auto match' function determines how much solder paste should be applied to each pad. The operator can choose from a library of shapes

— or create new design solutions for difficult or unusual components. When the operator is finished, MYCam generates a print program and sends it to the printer.

To maximize uptime, a lot of effort has gone into the development of a user-friendly interface that, together with a touchscreen, enables a stoppage time of less than 60 seconds.

In one prototype run, the MY500 allows you to use several different deposit volumes and shapes on different boards on the same panel, producing several different iterations. This helps to yield the perfect combination of shape and volume for the solder paste deposit, saving time in the prototype stage.

Although it is theoretically possible to alter the aperture size and shape and to use a stepped stencil to vary the deposits on different boards on the same panel in a prototype run, it has never, as far as we know, been tested in practice.

The absence of stencils means other cost-reduction benefits such as:

- No need for special paper for underside wiping
- No need for stencil cleaning machines
- No need for storage of stencils
- No risk of damaging the stencils during handling.

Damaged stencils results in faulty solder joints that need touch-ups or reworking of the soldering. The lead time for a new stencil to arrive delays production and may delay delivery to the customer.

Conclusions

Jet Printing Technology:

- Combines high-speed printing with flexibility and the possibility of depositing the right amount of solder paste for each component type
- Is operator-independent and reduces the number of printing parameters, hence improving reliability and the robustness of the printing process
- Offers swift product changeovers, which results in stoppage times of less than 60 seconds.

Read more

Visit the MY500 home page at: <http://www.mydata.com>

You can also call your local sales representative for a detailed presentation of the MY500 Jet Printing Process: <http://www.mydata.com>