Effect of stencil wall aperture on solder paste release via stencil printing

M. S. Mohamed Sunar1,2, M. Abu Bakar2, A. Jalar2,3, F. Che Ani1, M. R. Ramli1

1Western Digital®, SanDisk Storage Malaysia Sdn.Bhd. Plot 301A, Persiaran Cassia Selatan 1, Taman Perindustrian Batu Kawan, MK13, Batu Kawan, Seberang Perai Selatan, 14100, Penang, Malaysia
2Institut of Microengineering and Nanoelectronics, Universiti Kebangsaan Malaysia, 43600 Bangi, Selangor.
3Department of Applied Physics, Faculty of Science and Technology, Universiti Kebangsaan Malaysia, 43600 Bangi, Selangor.

Abstract. Solder paste printing is a process by which the correct amount of solder paste is applied to the printed circuit board via a stencil. The solder release from the stencil printing process very much depends on the type of solder paste and stencil conditions such as the shape of the aperture, size, and thickness of the stencil. This paper investigates the stencil condition in particular the stencil wall aperture and its relationship to the solder release ability. In this work, two types of stencil wall openings A and B were used, which differ in a different ways of cutting to achieve the wall aperture. The cutting process produced different surface roughness of the wall aperture of the stencil. Stencil printing was performed to print the solder paste onto the PCB pad. The release of solder paste was observed by solder paste inspection (SPI) and analyzed qualitatively and quantitatively. The results show that stencil B gives a better solder compared to stencil A. This is due to the smoother wall aperture compared to stencil A which has a roughened wall aperture. This shows that the performance of stencil printing in terms of solder printing quality is highly dependent on the surface roughness of the stencil aperture. Stencil quality is important as it affects the performance of solder paste printing, and this process is mainly carried out in the electronics industry. Therefore, understanding stencil conditions is important for electronic technology that uses solder printing.

1 Introduction

The electronics market has experienced significant growth in recent years, with the rapid rise of consumer electronics, particularly portable handheld devices, being a major factor. Customers increasingly desire portable, lightweight, durable, affordable, and high-performance products, such as mobile phones. Surface mounting components are becoming increasingly difficult as the trend towards miniaturization continues. Apart from the increased demand for lower prices, the manufacturer must look for cheaper methods of production.

Stencil printing is the process of depositing solder paste on the printed circuit boards (PCBs) before component placement and reflow process in the surface mount technology (SMT) field. Many studies have been published on solder paste printing and improvements in printing [1–5]. Stencil printing is so crucial in SMT that the industry reports 52–71% of fine-pitch SMT defects related to the solder paste stencil printing process [6]. The paste

2Corresponding author: azmn@ukm.edu.my
experiences stress at the aperture walls and pad surfaces as the stencil moves away from the PCB, which defines print quality. At the paste-stencil and paste-PCB interfaces, opposing forces are at work (Fig. 1). The eventual release of paste from the stencil aperture is determined by the net force.

Electroformed stencils are made on a conductive substrate. Normally, a laminator is used to coat this conductive substrate with a dry film layer of photoresist. A photographic film made on a photoplotter is then used to photolithographically pattern the layer of photoresist. The film is next developed, leaving tapering pillars that will create the apertures after the electroforming stage. The conductive substrate is subsequently immersed in an electrolytic nickel or nickel alloy plating solution. The plating process begins when a direct current is applied to the conductive substrate. When the stencil has attained the desired thickness, it can be removed.

Laser-cut stencils are created on a thin substrate, which is often made of metal. After production, this substrate is mesh mounted onto an aluminum frame to hold the foil flat and allow it to fit into the stencil printing process. The stencil is then placed in a high-powered laser cutting machine, which ablates each aperture in a predetermined order.

In production, poor stencil quality is often not recognized until defects are generated, with an associated cost. Since print quality is dependent not only on stencil quality, but also on the PCB surface, and screen printer programming, paste quality, and many other factors, it is usually time-consuming and costly to trace back poor print results to a low-quality stencil [8-11]. For small pad features, solder paste transfer efficiency is critical to prevent poor solder joints. Solder paste builds up onto the aperture walls, and the bottom side of the stencil leads to insufficient transfer of solder paste onto small pads. By troubleshooting the solder bridging issue, a solder bridging defect at the BGA solder ball’s location was encountered (0.106% yield loss). Root cause analysis found that this defect is attributed to the squeezing of the solder paste during NXT placement, thus causing the solder paste to bridge next to another (Fig. 2).
2 Methodology

A lead-free solder paste (Indium) was used during this study (Type 5, 96.5Sn–3.0Ag–0.5Cu, -500/+635 mesh according to ASTM B214). The solder paste size particles are in the range of 15 to 25 μm. Two types of stencils were used during the study. The stencils were made using nickel foil (4 mils thick). The process used to make the aperture for stencil A is by laser cutting (using laser beam spot of 0.03 mm size), while stencil B is made by electroforming. The solder paste was printed on a printed circuit board using stencil A and stencil B, individually. After deposition of the solder paste, the samples were characterized and analyzed concerning the nature of the stencil performance. Keyence High Optical Scope (200 X) was used to investigate the performance of the stencil to deposit the solder paste onto the PCB pad. A scanning electron microscope is used to observe the microstructure of the stencil aperture due to stencil cutting processes.

3 Result and Discussion

It would be advantageous if the quality of the stencils could be determined by a test method that does not require the printing of solder paste. These studies, which involved subjective visual or SEM inspection of stencil walls, have shown that there is a correlation between the smoothness of the walls and transfer efficiency (Fig. 3). Improvement on stencil design and fabrication method is to minimize the effect of solder bridging due to squeezing, the stencil aperture size has been reduced by 29.4% area percentage (0.52 area ratio, Table 1).

It is acknowledged that, since the original aperture size is 0.75 in area ratio, further reduction of stencil aperture size will supersede the minimum area ratio suggested by the IPC-7252B guideline (minimum 0.55 area ratio). The performance monitoring of the newly revised aperture showed that the yield loss due to solder bridging had been successfully eliminated (0% yield loss, Table 2). The thickness of the lead-free solder deposition was evaluated for both the square and the rounded pad on the printed circuit board based on the optimum parameter settings below (Table 2):
Fig. 3. The surface finish of stencil cut by: (a) Laser Cut and (b) Electroform.

Table 1. Aperture dimension of previous and improved stencil

| Type of Electroform Stencil | Width | Length | Shape | Step Thickness | Area Ratio |
|-----------------------------|-------|--------|-------|----------------|------------|
| Previous stencil aperture   | 10.18 | 8      | Q2    | 3              | 0.747      |
| Improved stencil aperture   | 8.4   | 8.4    | C     | 4              | 0.525      |

Table 2. Optimum parameter setting of the screen-printing process.

| Under Stencil Wipe | Wet/Vacuum/Dry 3rd Print |
|--------------------|---------------------------|
| Squeegee           | 300mm x 30mm x 0.2mm Squeegee blade with 60°. |
| Print Speed        | 25mm/s                    |
| Print Pressure (Front/Rear) | 7kg            |
| Separation Speed   | 1.0mm/s                   |
| Separation Distance| 3.0mm                    |
| Camera Resolution  | 10um                      |
| Threshold          | 40um                      |

Table 3. Yield loss and performance capability index (CPK) of the previous and improved stencil

Significant differences can be seen on the wall surface of the aperture stencils when two types of fabrication stencils are used. The first is laser cut and the second is electroform. The wall surface of the aperture stencil for the laser cut process looks quite rough and uneven when compared to the stencil that goes through the electroform process looks smoother and more even. It is noticed that the process capability index (CPK) of the newly revised stencil (with new area ratio) is below specification (poor solder release). It is hypothesized that this is due to the reduction of solder release capability because of aperture size and area ratio reduction. To improve the solder release capability, the stencil fabrication has changed from laser cut process to electroform process. Based on Fig. 4, we can see the difference of solder deposited on the PCB pad by using stencil A and stencil B. The picture before shows the amount of solder paste is quite large and will cause solder bridging to occur. When changes are made by reducing the area ratio and changing the stencil fabrication process to electroform, the solder volume rate looks good and can reduce the risk for solder bridging occur.
| Stencil material | Cutting method | Area ratio (r/2T) | Yield loss (ideal value: 0%) | Solder paste CPK (ideal value: >2) | Result |
|------------------|----------------|------------------|-----------------------------|-------------------------------|--------|
| Ni               | Laser-cut      | 0.75             | No                          | CAMSTAR-CFD                   | No     |
| Ni               | Laser-cut      | 0.52             | Yes                         | No                            | No     |
| Ni               | Electroforming | 0.52             | Yes                         | Yes                           | Yes    |

Fig. 4. The shape of solder pastes on PCB pad after stencil printing: (a) Stencil A and (b) Stencil B.

Fig. 5 and Fig. 6 show the difference in solder paste distribution using a different type of stencil in terms of volume after being inspected using an SPI machine. The result in Fig. 6 clearly shows a reduction in volume by using the laser cut stencil due to the wall roughness of stencil aperture cause the solder paste will be adhesive to the wall and release capability will drop. However, observation on the electroform stencil in Fig. 7 shows a smooth surface on the stencil wall aperture will make sure the solder release capability is good.

Fig. 5. Solder paste distribution using stencil A.
4 Conclusion

The results of a study on the effect of stencil design and stencil fabrication method on the printing process, specifically the impact on paste transfer efficiency, were provided in this work. Because the solder paste interacts with the stencil aperture walls and pad surfaces during the aperture filling and emptying sub-processes, it encounters forces/stresses that directly impact the paste flow within the apertures, a thorough understanding of the consequences of aperture design and fabrication method is critical. As the substrate and stencil split, the frictional/adhesive forces on the stencil walls compete directly with the adhesives/pull force on the PCB pads, resulting in inadequate paste transfer or clogged apertures. This means that addressing the issues of printing solder pastes at ultra-fine geometries, such as those necessary for fine pitch component technology, the aspect/area ratio, and the stencil fabrication method is critical. It's also important to keep in mind that paste particle size and other paste qualities have a direct impact on paste transfer, therefore lowering these ratios can aid in enhancing printing performance. Electroformed stencils have been seen to print better than laser-cut stencils in experiments conducted. During testing in high-volume manufacturing, the difference is more visible. Smooth aperture walls produced by electroforming nickel atom-by-atom around photoresist pillars release solder paste more effectively than laser-cut stencil walls.

The authors would like to acknowledge the technical support provided by Western Digital via SanDisk Storage Malaysia Sdn. Bhd through a research grant (RR-2020-004) and collaboration activities with Universiti Kebangsaan Malaysia through a research grant (GGPM-2019-056).

References

1. M. S. Rusdi, M. Z. Abdullah, M. H. H. Ishak, M. S. Abdul Aziz, M. K. Abdullah, P. Rethinasamy, A. Jalar, Int J Adv Manuf Technol 108, 3351–3359 (2020)
2. N. Ismail, A Jalar, M. A. Bakar, R. Ismail, N. S. Ibrahim, Solder Surf Mt Tech 32, 2, 157-164 (2020)
3. F. Che Ani, A. Jalar, A. A. Saad, C. Y. Khor, M. A. Abas, Z. Bachok, N. K. Othman Solder Surf Mt Tech 31, 2, 109-124 (2019)
4. N. Ismail, R. Ismail, A. Jalar, G. Omar, E. M. Salleh, N. K. Othman, I. A. Rahman, J Mater Sci: Mater Electron 29, 12910-12916 (2019)
5. F. Che Ani, A. Jalar, A. A. Saad, C. Y. Khor, M. A. Abas, Z. Bachok, N. K. Othman, Int J Adv Manuf Technol 96, 717–733 (2018)
6. K. Kim, M. Kang, J. Jang, C. Lee, Y. Lee, S. Hong, S. Yoo, IEEE 65th Electronic
Components and Technology Conference (ECTC), 225-229 (2015)
7. R. Kay, M. Desmulliez, Solder Surf Mt Tech 24, 1, 38-50 (2012)
8. O. Krammer, K. Dusel 2019 J Manuf Process 45, 188-193 (2019)
9. O. Krammer, B. Gyarmati, A. Szilagyi, B. Illes, D. Busek, K. Dusek, J Mater Process Tech 262, 571-576 (2018)
10. N. Khader, J. Lee, D. Lee, S. W. Yoon, H. Yang, Procedia Manuf 38, 163-170 (2019)
11. H. Lu, H. Wang, S. W. Yoon, D. Won, S. Park, Procedia Manuf. 38, 108-116 (2019)