The trend towards ever smaller components and higher function density continues unabated in the SMT field. To master the challenges this poses, it is no longer enough to simply make the components smaller. Instead, engineers must analyse the interactions between materials and take them into account for their manufacturing processes. In order to achieve good manufacturability, all parties – the designer, the PCB manufacturer, the printer, stencil and solder paste makers, as well as the pick & place equipment manufacturer and the reflow expert – must work together. Only a common effort will ensure high quality products.

01005 components challenge all processes in the PCB assembly sequence because of their size – only 0.2 mm by 0.4 mm. They are almost invisible, at least for the “naked” eye, and extremely lightweight (0.04 mg). With this in mind, it is easy to understand that the whole assembly process, but even more importantly, the materials and the layout of the PCBs, must be designed for these components.

Now which process steps or topics need to be looked at more closely? Of course, the first question one should ask is whether 01005s should be used at all? And, if so, what benefit do they offer? When these questions are answered, the design and layout can begin.

Once the PCB is created, the solder paste application is the next crucial step. Since this step contributes significantly to the quality of the finished (reflowed) board, any effort to be successful here is worth it. For the placement of these “dust-like” components, correct and reliable pickup is the key to a smooth placement process. On the other hand, reflowing 01005s does not seem to be significantly different from soldering bigger passive components.

**PCB design and pad layout**

In SMT (Surface Mount Technology), NSMD (Non Solder Mask Defined) pads are typically used. However, when it comes to small passive components some people tend to use SMD (Solder Mask Defined) pads. In Figure 2 the different appearance of both pad types can be seen. On the left, the NSMD pad with the typical opening of the solder mask is shown and on the right the SMD pad where the copper extends underneath the solder mask can be seen.

Recently a combination of the two pad types, a so called semi-solder mask defined pad, has been invented and it seems to have some advantages over to the other two types.

When comparing the three pad types, the question to ask is: do they behave any differently during solder paste printing, placement and reflow?

When looking into these processes, it was discovered that the three types do not show a significant difference in how close two adjacent components (gap) can be arranged on a PCB without creating any electrical or manufacturing problems. As concerns PCB manufacturing, solder paste printing and component placement, the following was discovered:

- For all three types the minimum gap should not be less than 150 µm.
- The limiting factor for the board is the registration tolerance of the solder mask which today is around +/- 50 µm.
As far as solder paste printing is concerned, it is important that the stencil material remaining between two openings has enough mechanical strength so that no damage occurs.

• The SMD or the semi-SMD pad has the advantage that the stencil sits on top of the solder mask and therefore seals securely.

• The solder mask also stabilises the bottom layer of the solder balls of the printed solder paste layer. This results in a more well-defined print.

The solder mask, especially for such small components, has an important function if it is well designed. It has to reliably cover wettable areas like tracks because uncovered tracks can cause shifted components during reflow.

Figure 3 shows an example of tracks between pads which are not separated by solder mask. Another design error is that the track width is only slightly reduced compared with the pad width. This allows the solder to flow between pads without any barrier during reflow. When the solder is in the liquid state, the surface tension of the molten solder results in a force pulling components out of their initial position and, with such a layout, will cause misalignment. The problem is that if the solderable pad area is not defined or limited by solder mask the movement of such light weight components is uncontrolled. The most desirable situation is that the solderable pad area is well defined by the pad design. This leads to so called “self-alignment” of the components which can compensate for minor shifts in the paste print and/or component placement.

Figure 4 shows the accurate placement of three 01005 components with 120 µm gap. However, the solder paste is too much due to a relatively thick stencil (80 µm) and consequently the solder paste on the pad is a little bit smashed.

Figure 5 illustrates what happened during reflow. When the solder liquefied the components were pulled together out of their predetermined position. The result is a pile of components which electrically would still work fine, but would not be acceptable for any quality inspection. Figure 6 shows what happens when the solder mask opening is too large and portions of the tracks are uncovered. As can be seen, even single components will be pulled to the side where a track is connected into the pad. If pad one of a component has a track connecting to the opposite side than the component on the track connecting into pad two will end up tilted after reflow.

The examples above prove that a good PCB design is essential for quality. Especially for miniaturised components, design for manufacturability is very important.

Solder paste application

The fundamental principle of solder paste printing using a stencil is that the paste has, in the moment it is deposited, less adhesion to the sidewalls of the stencil opening than to the surface of the pad. If
the adhesion to the sidewalls exceeds that to the pad, no printing is possible. Since the adhesion to the sidewalls is a function of their roughness, it is important to select the best combination of material and manufacturing methods for the stencil. Also important is the area ratio which can be calculated when the dimensions of the opening and the stencil thickness are known. Since most of the time the stencil thickness and the opening size are not really negotiable, the only remaining influencing factors are the selection of the stencil material and the method of manufacture. Therefore, it is imperative that a solder paste with good printing properties and a fine enough powder size is chosen.

The best solder paste release properties were achieved with electroformed Nickel stencils, however, the best overall print results were achieved with laser cut Nickel stencils. This was due to the fact that for both stencil types the Nickel is electrochemically deposited, but for the electroformed stencil the openings are created with a film-based process which is less accurate than laser cutting of the openings.

The solder paste powder type can be selected using the common rule of thumb that five balls of powder should fit through the smallest stencil opening side by side. The conclusion will be to use at least a type 4 or better a type 5 paste.

For good print results it is also important to have a well-supported PCB. If the PCB becomes very thin or below 0.5 mm, a vacuum tooling will be important to flatten the PCB while supporting it from the bottom side. It is also important that the stencil is wiped off often enough so that it is kept clean underneath.

Component placement

For the placement process, the first and most important step is to pick up the component out of the feeder (Figure 7). Preconditions for good component pick up are:

- Good component quality (all dimensions / tolerances within specification)
- Good tape quality (all dimensions / tolerances within specification)
- Feeder unit with sufficient positioning accuracy and repeatability
- Feeder unit with fiducial for exact position recognition

Once the component is picked up, the vision recognition system must have a high resolution and accuracy in order to calculate and correct the position precisely enough. It is also very important that conditions (Figure 8) which would result in defects later on (e.g. face down or bill boarded components) are detected now and corrected immediately.

In addition, a placement system must be able to verify that no components are lost on the way from the feeder to the placement position. Due to the extremely small nozzle tip for 01005 components, vacuum sensing is not a reliable method of detecting the component. Therefore, a laser sensor should be used to detect absence/presence of the 01005 component during the pick and place process (Figure 9).

During touch down of the component onto the PCB the placement force and the velocity of the Z-axis need to be controlled. If the speed is too high, the solder paste gets into contact with the component and may splash. Additionally, if the force is too much the solder deposit will be smashed as well.

Force control is essential to avoid any mechanical damage to the component which is usually very sensitive and may break if more than approximately 2N are applied. Finally, during placement the PCB needs to be well supported so that no vibrations cause already placed components to move out of position.

Reflow soldering in nitrogen is recommended for 01005 components because the solder pastes used are usually of the fine-grained variety (Type 4 and often Type 5), causing them to oxidise more easily than traditional Type 3 solder pastes.

Although one should not depend on the self-centring effect during soldering, reflow soldering in nitrogen improves this effect because it generally improves the solder paste’s wetting characteristics.
As with other components, you must create a reflow profile for 01005s that complies with the usual criteria such as gradual warm-up rate / tomb-stoning, and compliance with the maximum permissible temperature/time limits for components, substrates and solder pastes.

Self-centering does work quite well for very small components even when lead-free solder is used. Due to the fact that 01005 components weigh only 0.04 mg the surface tension of lead-free solder is sufficient to move them. Figure 11 shows an example of a test-board before and after reflow. The components were shifted by approximately 100 µm and are perfectly aligned after reflow. But this self-centring-effect has its limitations. The solderability of the pad, the component terminations and the solder paste have to be good, and the pad, the printed solder paste deposit and the termination of the component have to overlap enough in order to avoid tombstoning. If all conditions are fulfilled, the self-centring works best and most reliably for a shift of up to 50 µm. Since the self-centring effect is not a process element and cannot be controlled, it should not enter in the process plan.

Rework

Rework is not recommended when dealing with 01005 components. This is due to the following facts:

- Even very fine soldering tips are too big for 01005 components.
- A safe mechanical contact to transfer the heat is virtually impossible.
- The danger of causing mechanical damage to the component or pad is quite high.
- The very small structures and components cannot be seen with the naked eye.

Mass production rework of 01005s is not an option. However, if occasionally one component needs to be reworked it could be done by highly skilled workers, equipped with a microscope, a laminar hot air pencil, micro tweezers and some flux.

General comments

The processing of 01005 components not only requires very good process knowledge and planning to design the board/product for good manufacturability, but it also requires that the production is done under controlled environmental conditions. It is not necessary to do it in a clean room, but it must be done under clean conditions. Also temperature and humidity should be controlled and kept constant.

Figure 12 shows a board where a small fibre had fallen across some pads. This would create open solder joints or tombstones if not detected before reflow.

The future of 01005s

As a general statement it can be said, that the use of 01005 components make the product more expensive because of the higher costs for materials (components, PCB, solder paste, stencil, nitrogen (reflow), more accurate equipment). The layout and the board design need to take into consideration all process steps to achieve a “manufacturable” product with sufficient quality. Because of the narrower process window, 01005 components are limited to applications where the range of different components is not spread very wide. Therefore, 01005 components are used where miniaturisation is the key driver, like in submodules, sensors and medical devices like hearing aids. In those applications it is not a problem that the electrical performance of the components is also limited due to their small
volume (e.g. limited capacitance). But their use is also beneficial if it is necessary to get the components close to active ones or when they need to be embedded in the PCB. The small size has the advantage that the mismatch of the thermal expansion coefficient does not cause any damage when thermal stress is applied.

The solder joints are reliable, but when the pad design for an 01005 is done, the designer has to keep in mind that the adhesion of the copper pad to the FR4 material may become the weakest point, especially when the packaging density is high. This is due to miniaturisation, and the pad dimensions are getting smaller and smaller.

So far the most difficult process step has been the solder paste application for all who tried to develop and establish an 01005 process in production. But this is not surprising, since this process step has always contributed directly or indirectly to the most defects detected after reflow on Surface Mount Boards. As a consequence of that, it makes sense to use an AOI to find and optimise the process parameters when setting up an 01005 process. It may also be a good idea to inspect the printed boards in a running production.

Nevertheless, the 01005 is a component which is available today, and will be used in electronics production. It will not however replace the most common passive components like 0402 and 0603. It will be used for certain applications and by a limited number of electronic manufacturers.

This article is based on a paper originally presented at the IPC Printed Circuits Expo, APEX and the Designer’s Summit 2008

---

**Pick & Place Redesign Improves Maintenance And Reliability**

Assembléon, a world leader in surface-mount pick & place solutions for the electronics industry, announced that design improvements in maintainability and reliability are reducing lifetime costs for its established MG-1 and MG-8 pick & place machines. The new MG-1R and MG-8R, have been improved to allow for easier maintenance access and better maintainability for the head components. Also, the maximum board size has been increased to 510 x 440 mm (20” x 17.2”), and the operating panel, LCD monitor and keyboard have been relocated to improve ergonomics.

The MG-1R and MG-8R machines are for high-mix production, and place a wide range of components including chips as well as complex and odd-form components. That enables the machines to be broadly used as stand-alone and in-line solutions for industrial, consumer, automotive and other industries. Placement rates are up to 24k components per hour.

A 3-D co-planarity camera and a side view camera help eliminate placement defects. High resolution vision systems recognise, dip-flux, and place even the smallest balls and tightest-pitch flip-chips.

The M-Series is easy to maintain. All modules verify the correct condition of the placement nozzles and have a standard nozzle cleaning system. The modules constantly self-calibrate to eliminate thermal instability from changing production environments. There is off-line setup and feeder exchange trolley verification.

Assembléon
Building HVS-1, PO Box 80067
5600 MD Eindhoven – The Netherlands.
Tel. +31 40 27 22107
Fax. +31 40 27 66541
siu.yuen@philips.com
www.assembleon.com
Faster Fluid Dispensing With Dual-Shuttle Standalone Dispenser

Asymtek, a Nordson company and leader in dispensing, coating, and jetting technologies, introduces the Spectrum S-822 dispensing system with dual-shuttle stages to increase the productivity achieved in such processes as chip scale package (CSP) underfill and other advanced electronics applications. The S-822 is a standalone system that offers the controlled precision dispensing capabilities and many of the patented processes found in Asymtek’s advanced inline systems.

The system allows parallel processing on two shuttle stages for continuous dispensing, which significantly reduces lost time in non-dispensing activities such as pre-heating parts and substrate loading and unloading. The dispenser’s two stages automatically slide in and out through an opening in the front of the machine, allowing the operator to safely unload and load one side while dispensing continues on the other stage. The S-822’s dual-shuttle configuration saves time when a pre-heat process is required and lowers cost-of-ownership without adding another full system to the production line, minimising manufacturing cost per square foot. Options include choice of impingement or contact heated stations for one or both shuttle stages. The S-822 system performs with a wide range of fluids, processes, and substrates and easily integrates with most jets, pumps, and valves from Asymtek.

Asymtek
Tel. +31 43 352 4466
rfsmith@asymtek.com
www.asymtek.com/eu

Electrolube manufacture a wide range of high performance chemicals used extensively by the international electronics and manufacturing industries.

Automotive  Traction  Industrial electronics  Marine  Service
Military  Aerospace  Consumer electronic  Marine  Service
Transport  Telecommunications  Medical  Repair

You need electrolube.com

Electrolube manufacture a wide range of high performance chemicals used extensively by the international electronics and manufacturing industries.

email info@hkww.co.uk  Telephone +44 (0) 1283 222111

International Headquarters - Kingsbury Park, Midland Road, Swadlincote, Derbyshire DE11 0AN United Kingdom. Offices in Europe, America, South America, China and Australasia with our own manufacturing plants in the UK, Brazil and China.