01005 Assembly Process – From the Board Design to the Reflow Process

Norbert Heilmann
Siemens AG
Munich, Germany

Abstract
The trend towards ever smaller components and more function density continues unabated in the SMT field. Manufacturers and users must increasingly coordinate their activities to develop usable and cost-effective solutions. Progress doesn't stop, especially in the world of electronics. Electronics are used in a wide variety of applications. More and more functions are being crammed into ever smaller modules. To master these challenges on the path from SMD technology to the world of microelectronics, it is no longer enough to simply make the components smaller. Instead, the engineers must analyze the interactions between materials and take them into account for their manufacturing processes. In order to achieve good manufacturability all parties, starting with the designer, the PCB-manufacturer, the printer-, stencil- and solder paste maker, as well as the Pick & Place equipment manufacturer and the reflow expert should be consulted. Only a common effort will ensure good quality.

Introduction
01005 components challenge all processes in the assembly sequence because of their size of 0.2 mm by 0.4 mm. They are almost invisible, at least for the "naked" eye, and extremely light weight (0.04 mg). With those facts 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 the use of these components.

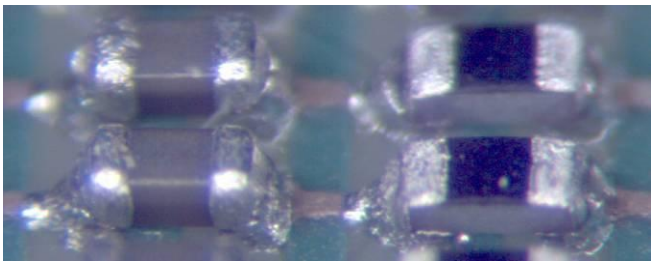


Figure 1 - 01005 Capacitors and Resistors

Now which process steps or topics need to be looked at closer? Of course, the first question to ask is if 01005s should be used or not. And, if so, what benefit do they offer. When these questions are answered, the design and layout should 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 particle like" components the correct and reliable pickup is the key to a smooth placement process. Reflowing 01005s seems to be a process step which is not really different to soldering bigger passive components.

PCB Design and Pad Layout
In SMT (Surface Mount Technology) typically NSMD (**N**on **S**older **M**ask **D**efined) pads are used. However, when it comes to small passive components some people tend to use SMD (**S**older **M**ask **D**efined) 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 compared to the other two.

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

When looking into these processes we found out that all 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. Relative to the PCB-manufacturing, the solder paste printing, or component placement we discovered the following:

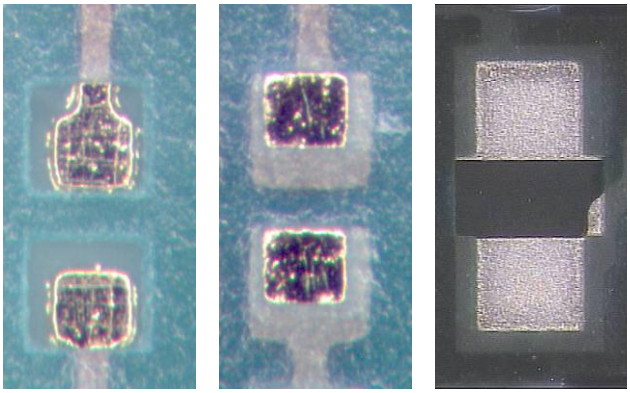


Figure 2 – NSMD-Pad SMD-Pad Semi-SMD-Pad

- 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.
- For the solder paste printing it is important that the remaining stencil material 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 is also stabilizing the bottom layer of the solder balls of the printed solder paste layer. This results in a better defined print.

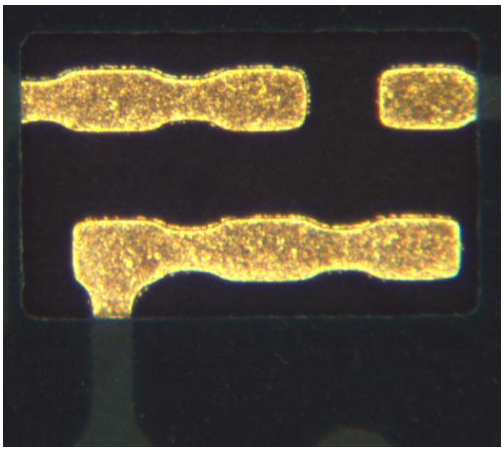


Figure 3 - Layout without solder mask between pads

The solder mask, especially for such small components, has an important function if it is well designed. It has to reliably cover wetable 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 in case of such a layout it will cause mis-alignment. 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 effect is if the solderable pad area is well defined by the pad design. This leads to the so called "self-alignment" of the components which can compensate for minor shifts in the paste print and/or component placement.

In figure 4 you can see the accurate placement of three 01005 components with 120 µm gap. However, the solder paste is too much due to a relatively thick stencil with 80 µm, and consequently the solder paste depot on the pad is smashed a little.

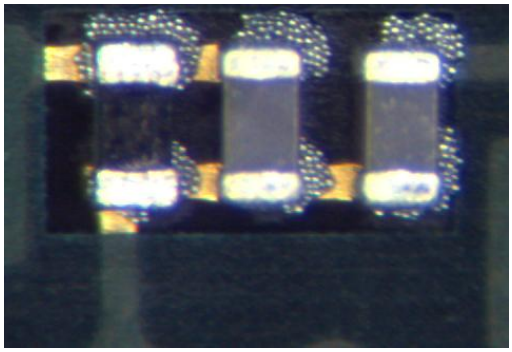


Figure 4 - 01005 placed in paste, correctly aligned

In figure 5 you can see what has 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.

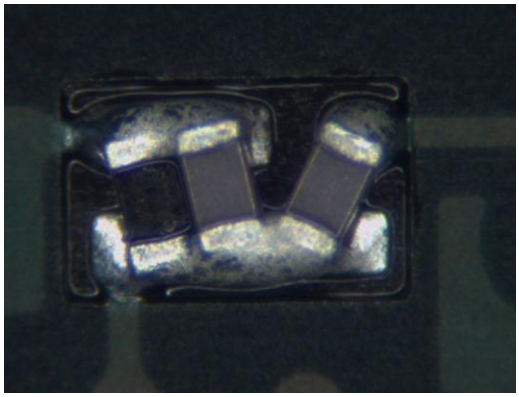


Figure 5 - Component "pile" after reflow

Figure 6 shows the effect when the solder mask opening is too large and portions of the tracks are uncovered. You can also see that 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 above examples prove that a good PCB design is essential for quality. Especially for miniaturized components design for manufacturability is very important.

Solder Paste Application

The principle of solder paste printing via stencil is that the paste which is first filled in the opening (by the squeegee) has in the moment of depositing less adhesion to the sidewalls of the opening than to the surface of the pad. If the adhesion to the side walls exceeds the one to the pad, no print is possible. Since the adhesion to the side walls 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 times 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 manufacturing. Therefore, it is imperative that you choose a solder paste with good printing properties and a fine enough powder size.

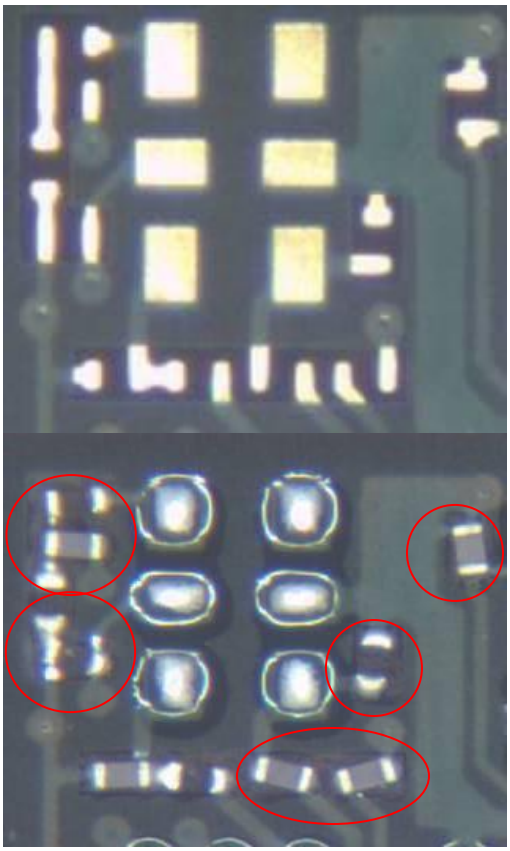


Figure 6 - Impact of large openings in the solder mask

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 then cutting the openings with a laser.

The solder paste powder type can be selected with the common rule of thumb in mind 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). Precondition for good component pick up is:

- 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

Figure 8 - Nozzle at Pickup Position

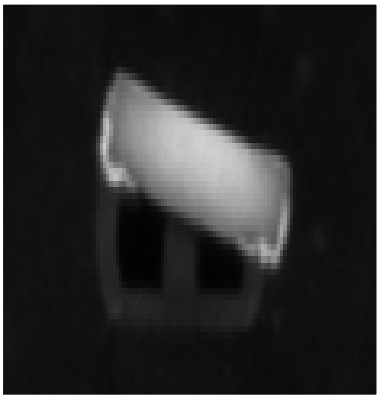


Figure 7 - Bill board component at the nozzle

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 precise 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. (See 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 depot will be smashed as well. The force control is essential to avoid any mechanical damage to the component which is usually very sensitive and may break if forces bigger than approx. 2 N 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.

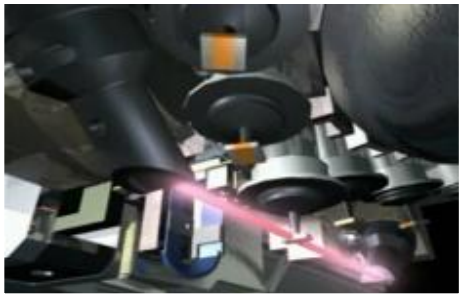


Figure 9 - Component sensor in action

Reflow

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 oxidize more and more easily than traditional Type 3 solder pastes. Although one should not depend on the self-centering effect during soldering, reflow soldering in nitrogen improves this effect because it generally improves the solder paste's wetting characteristics.

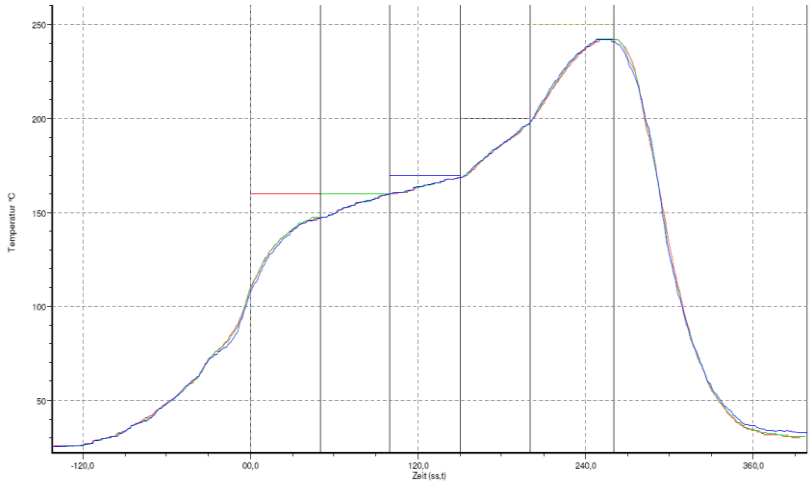


Figure 10 - Standard reflow profile for lead free

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.

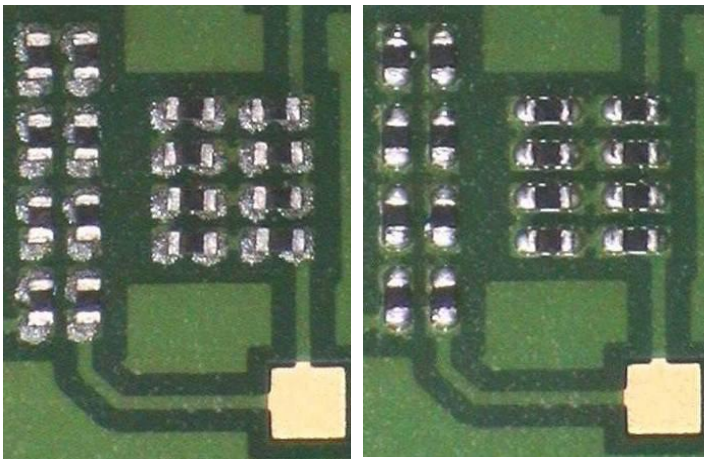


Figure 11- "Before" and "After" Reflow - Self-alignment

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-centering-effect has limitations. The solderability of the pad, the component terminations and the solder paste has to be good, and the pad, the printed solder paste depot and the termination of the component have to overlap enough in order to avoid tombstoning. If all conditions are fulfilled the self-centering works best and most reliable for a shift of up to 50 µm. Since the self-centering effect is not a process element and cannot be controlled, it should not enter in your process planning.

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 can not be seen with the naked eye.

Mass production rework of 01005s is not a choice. 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 it was not detected before reflow.

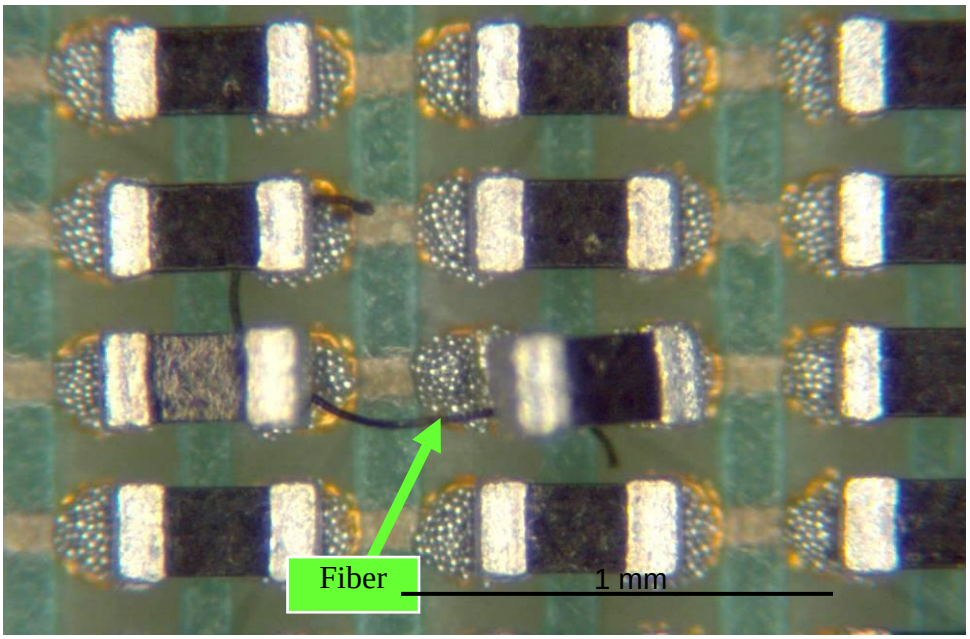


Figure 12 - Fiber across pads

Conclusions
As a general statement it can be said, that the use of 01005 components makes 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 be done under consideration of 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 miniaturization is the key driver, like in submodules, sensors and medical devices like hearing aids. In those applications it is no problem, that the electrical performance of those components is also limited due to the 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. Mechanically the small size has the advantage that the mismatch of the thermal expansion coefficient does no harm any more when thermal stress is applied.

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

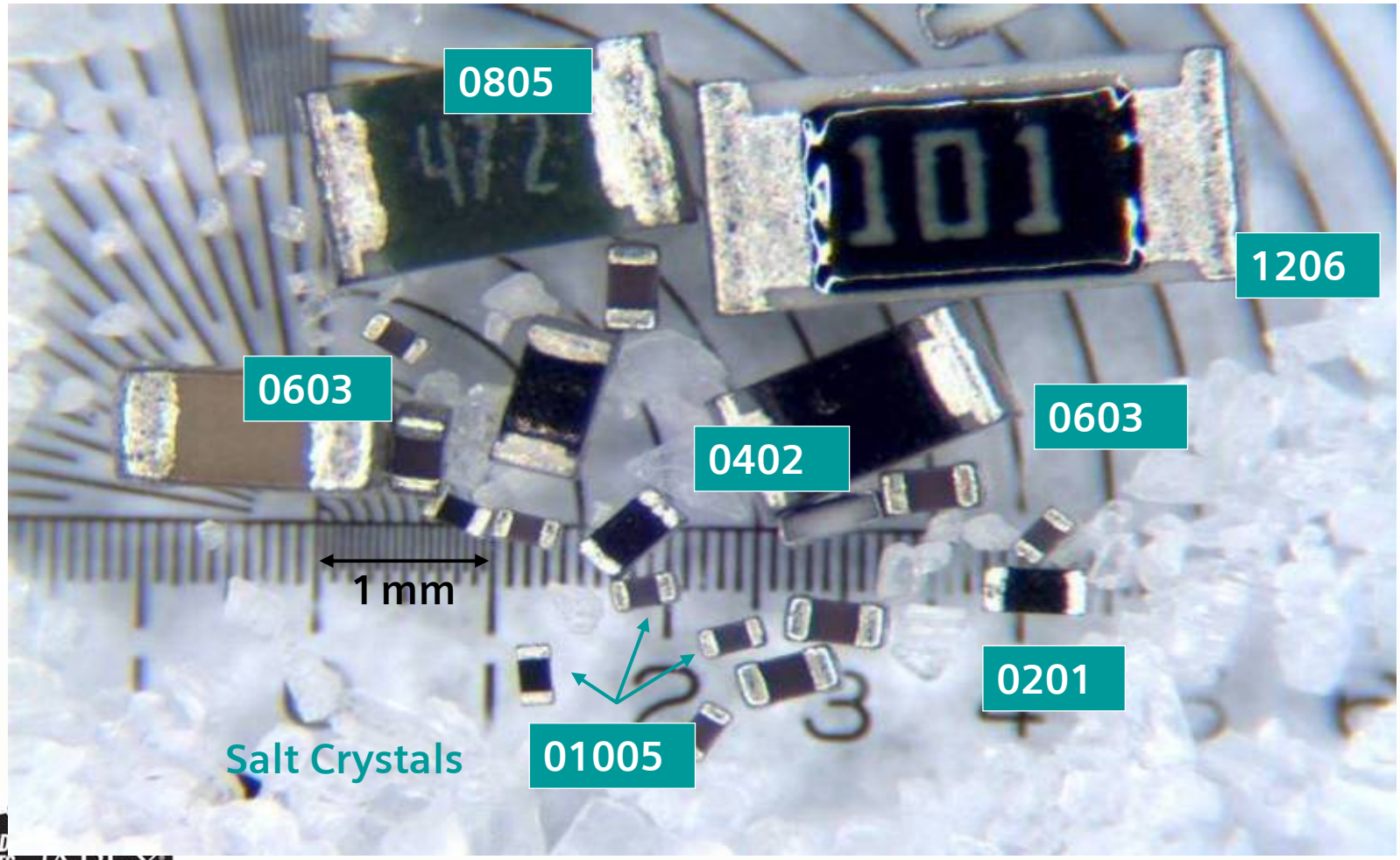
So far the most difficult process step has been the solder paste application for all who tried to develop and establish the 01005 process in production. But this is not surprising, since this process step has always contributed directly or indirectly to the most defects detected after reflow on Surface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

Never the less, 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.

01005 Assembly Process

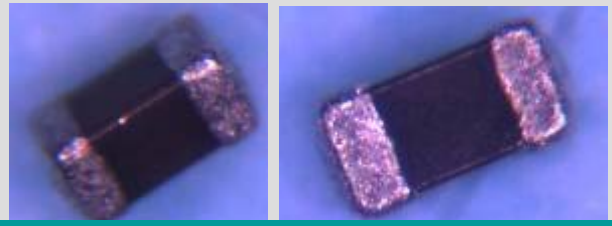
From the Board Design to the Reflow Process

Size Comparison of passive Chip Components

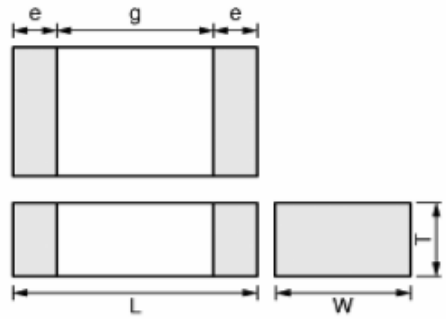


Typical Appearence/ Dimensions of 01005 Components

Capacitor

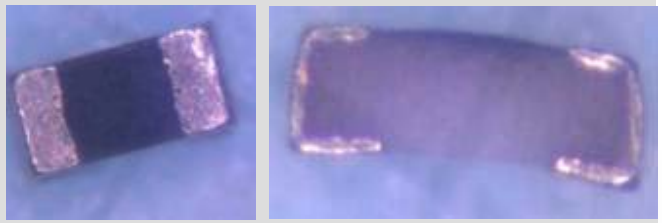


Dimensions (mm)				
L	W	T	e	g min.
0.4 ±0.02	0.2 ± 0.02	0.2 ± 0.02	0.07 to 0.14	0.13

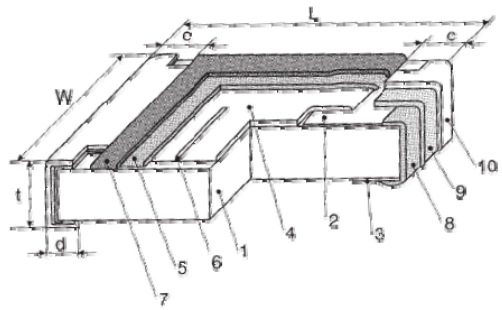


Source: Murata

Resistor

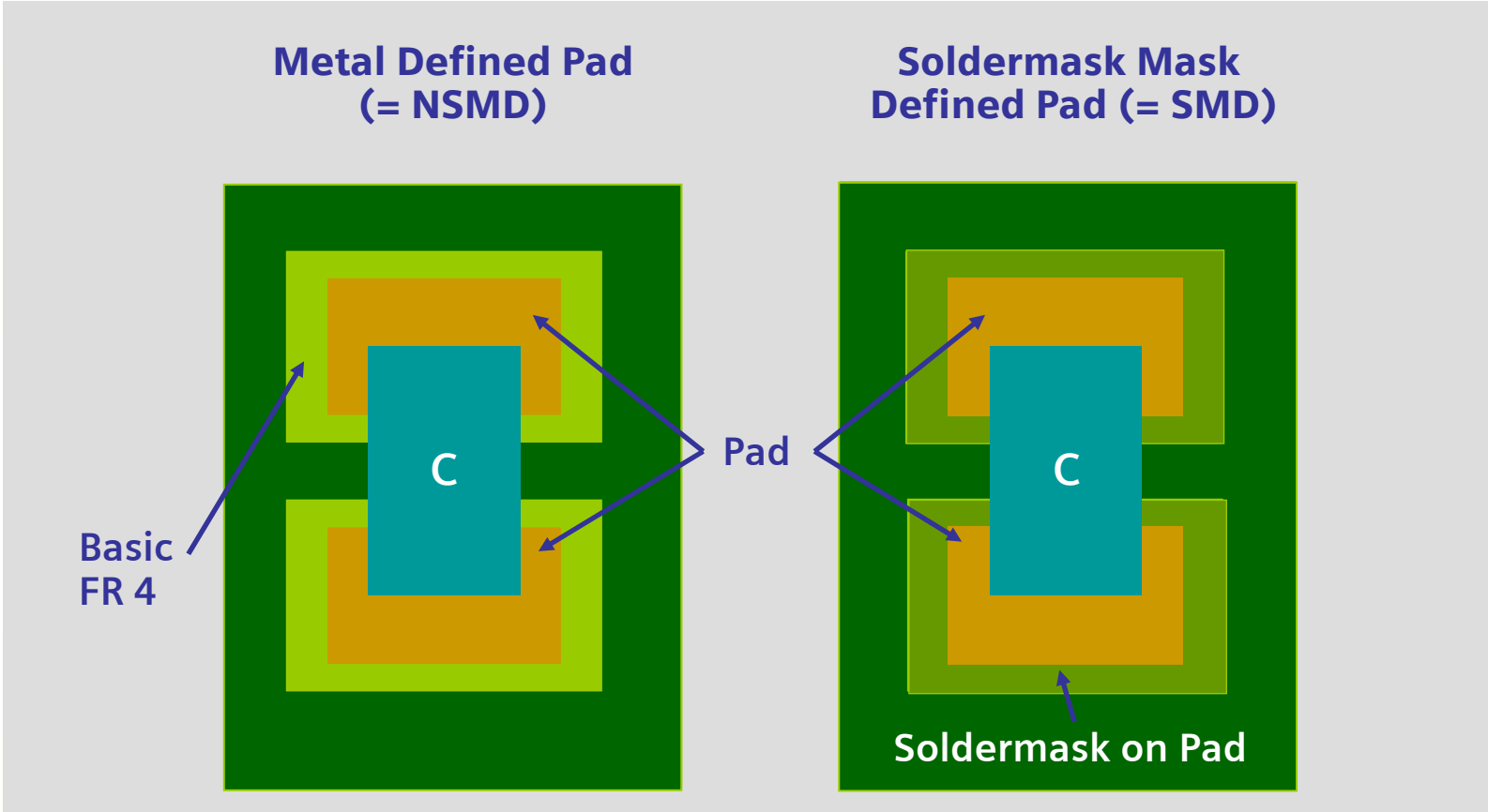


Dimensions (mm)				
L	W	c	d	t
0.4 ± 0.02	0.2 ± 0.02	0.1 ± 0.03	0.1 ± 0.03	0.12 ± 0.02

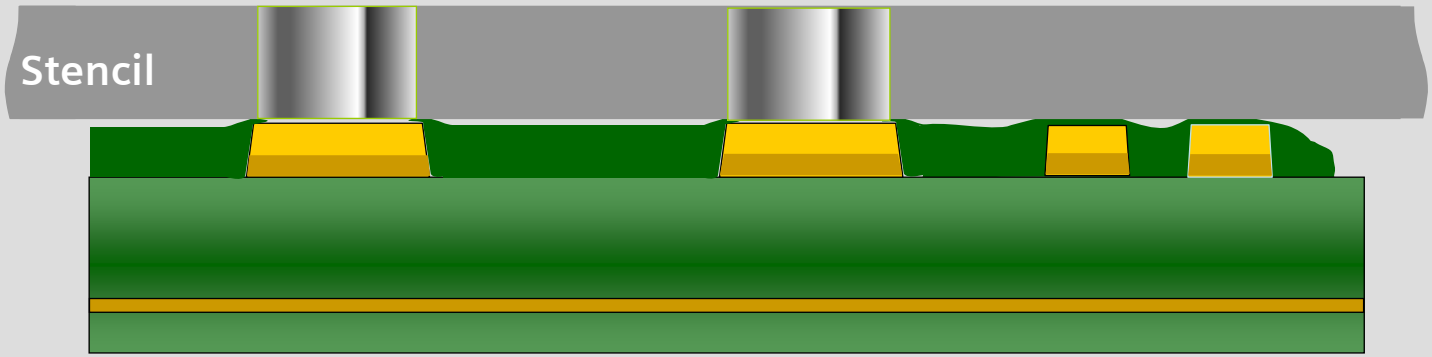


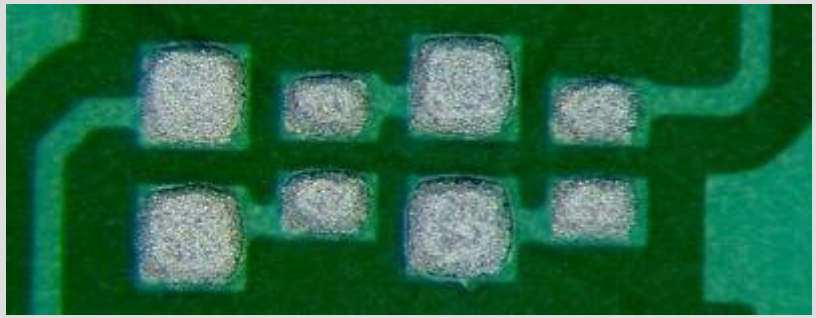
Source: Rohm

Definition of Pad Design Variations (NSMD and SMD)

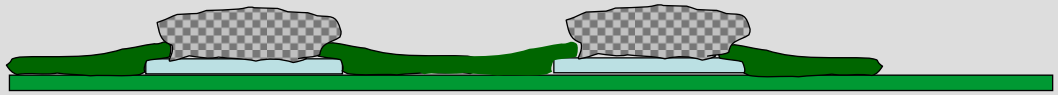


Stencil sitting on Top of the Solder Mask (SMD-Pad)



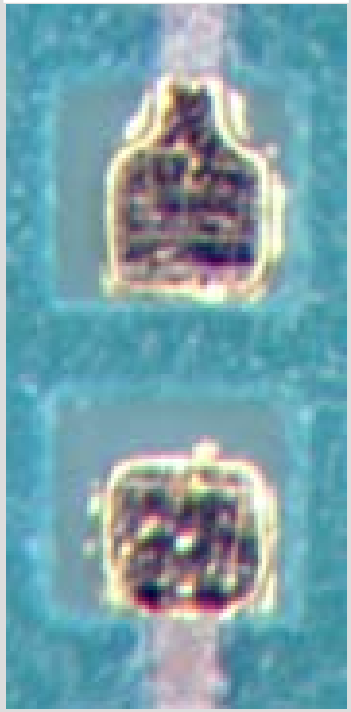


Real Paste Print

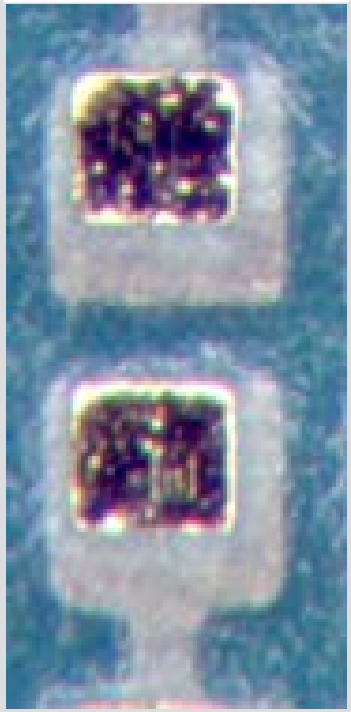


Comparison NSMD / SMD / Semi-SMD-Pads

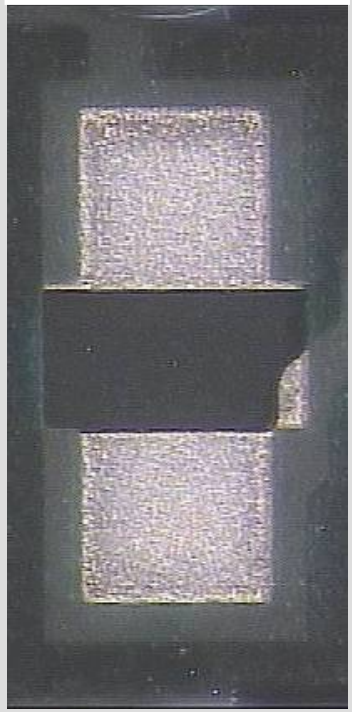
NSMD-Pad



SMD-Pad



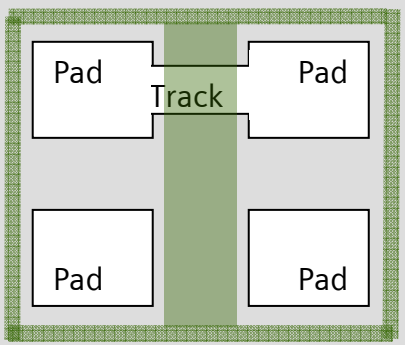
Semi-SMD-Pad

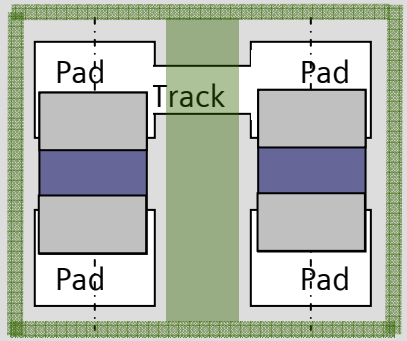


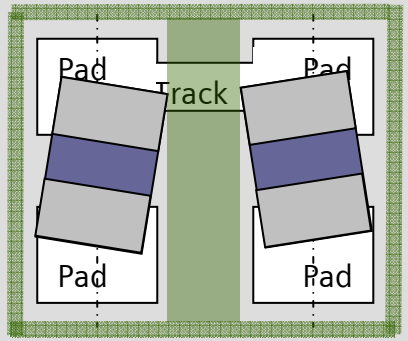
Obviously problems will occur when the solder mask is mis-registered (typical tolerance up to +/- 50 µm)

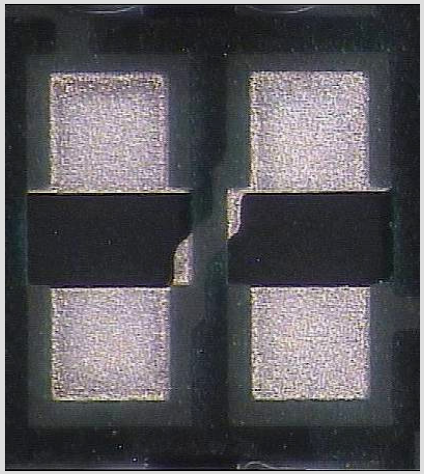
Advantage of a Semi-SMD Pad:

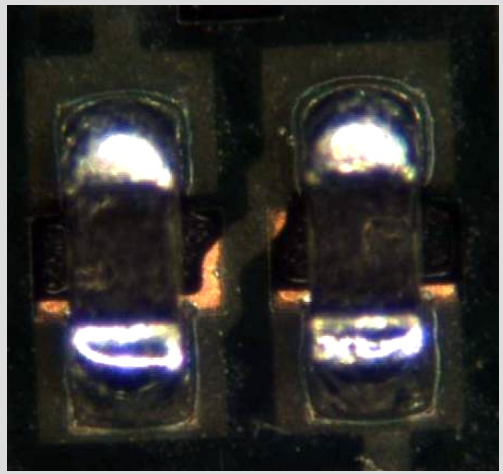
Avoid Component Shifts During Reflow



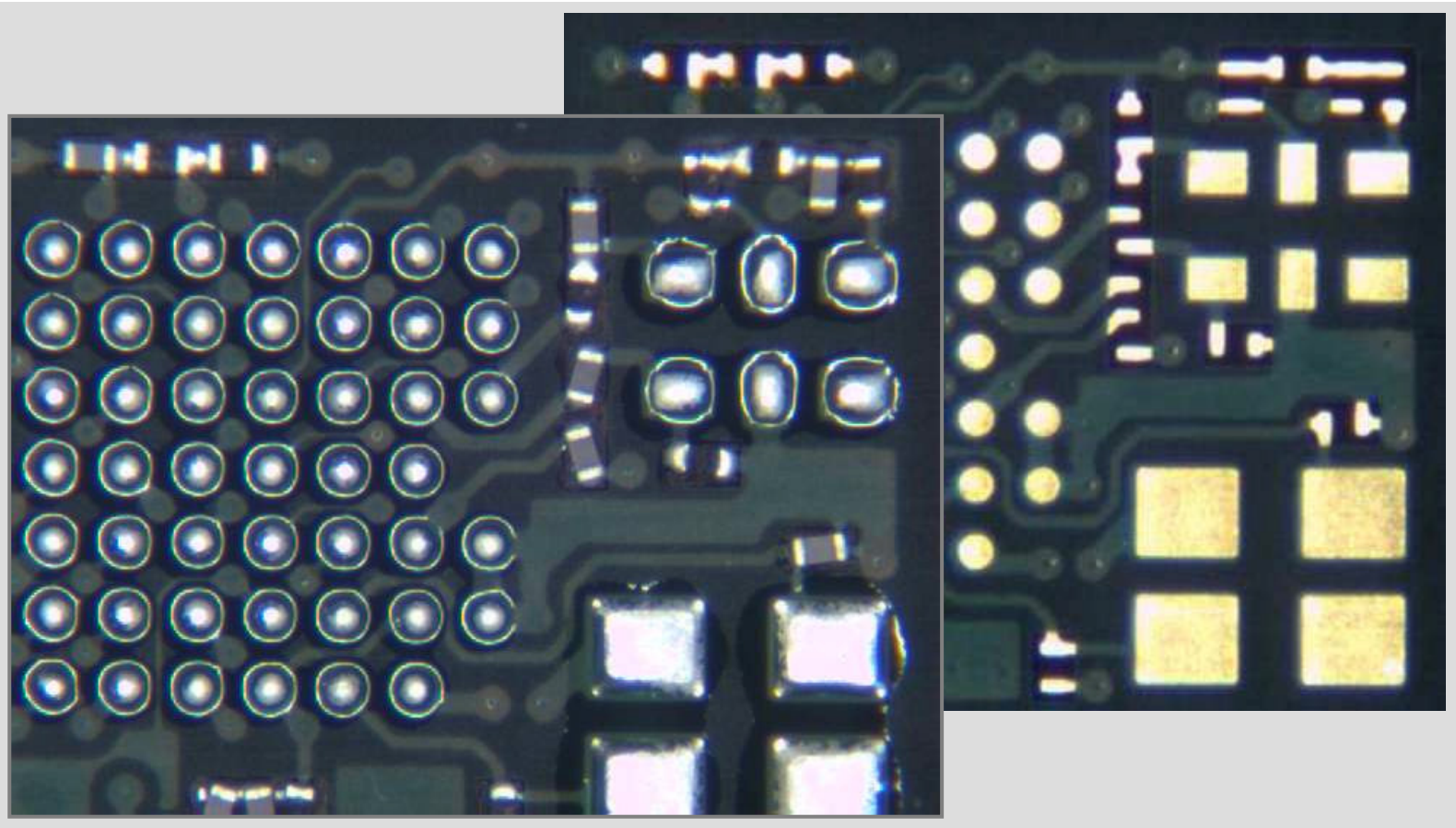




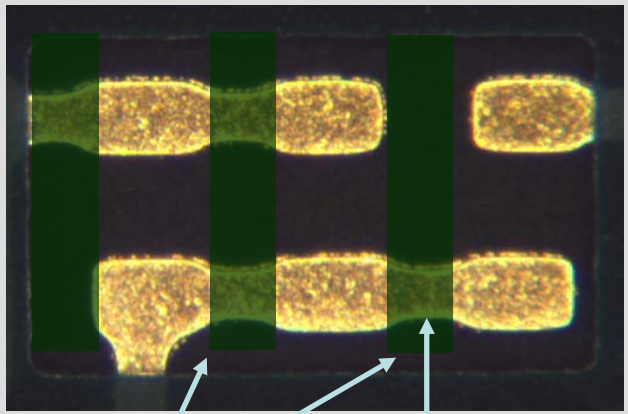


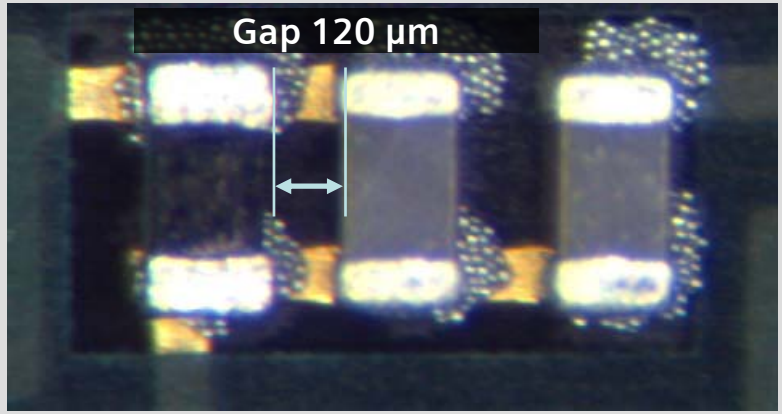


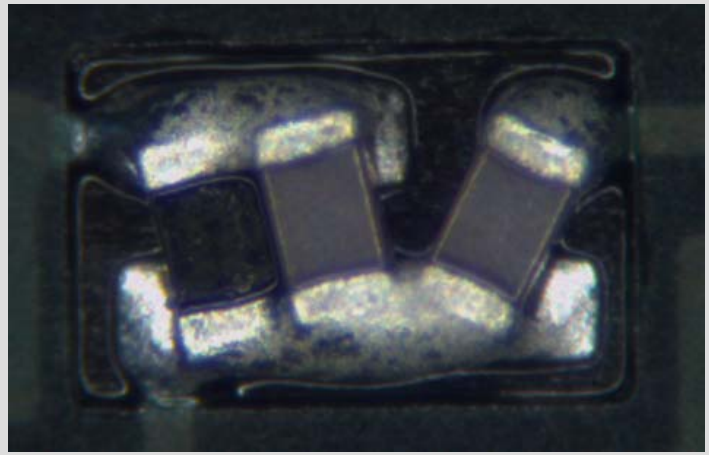
Impact of Layout on Component Positions, due to uncovered tracks



Example: Bad Design / Layout







No solder mask to separate connected Pads

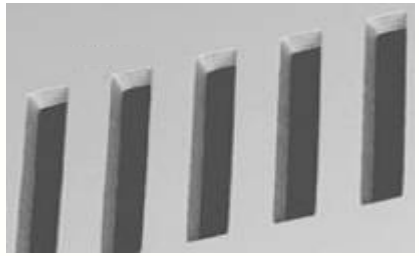
No sufficient width reduction of tracks, to avoid solder thieving

Excessive Solder Paste (80 µm)

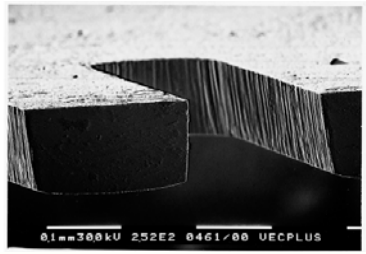
Stencil Types for 01005

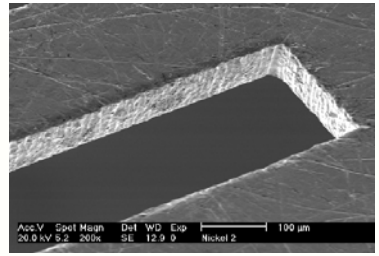
Stainless Steel
Laser cut

Nickel
electroformed



Nickel – Laser cut





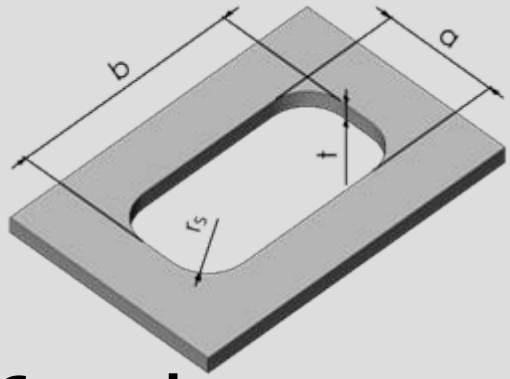
	+	–
Stainless Steel Laser Cut	Less expensive Precise Standard	Least good Solder Paste release
Nickel Laser Cut	Better Solder Paste release, Precise	Only available for special frame type
Nickel Electroformed	Best Solder Paste release	Lithographic Process

Stencil Design Basics

$$\text{Aspect Ratio} = \frac{\text{smallest Aperture width (a) oder Diameter}}{\text{Stencil thickness } t}$$

$$\text{Area Ratio} = \frac{\text{Length} \times \text{Width}}{2 \times(\text{Length} \times t) + 2 \times(\text{Width} \times t)}$$

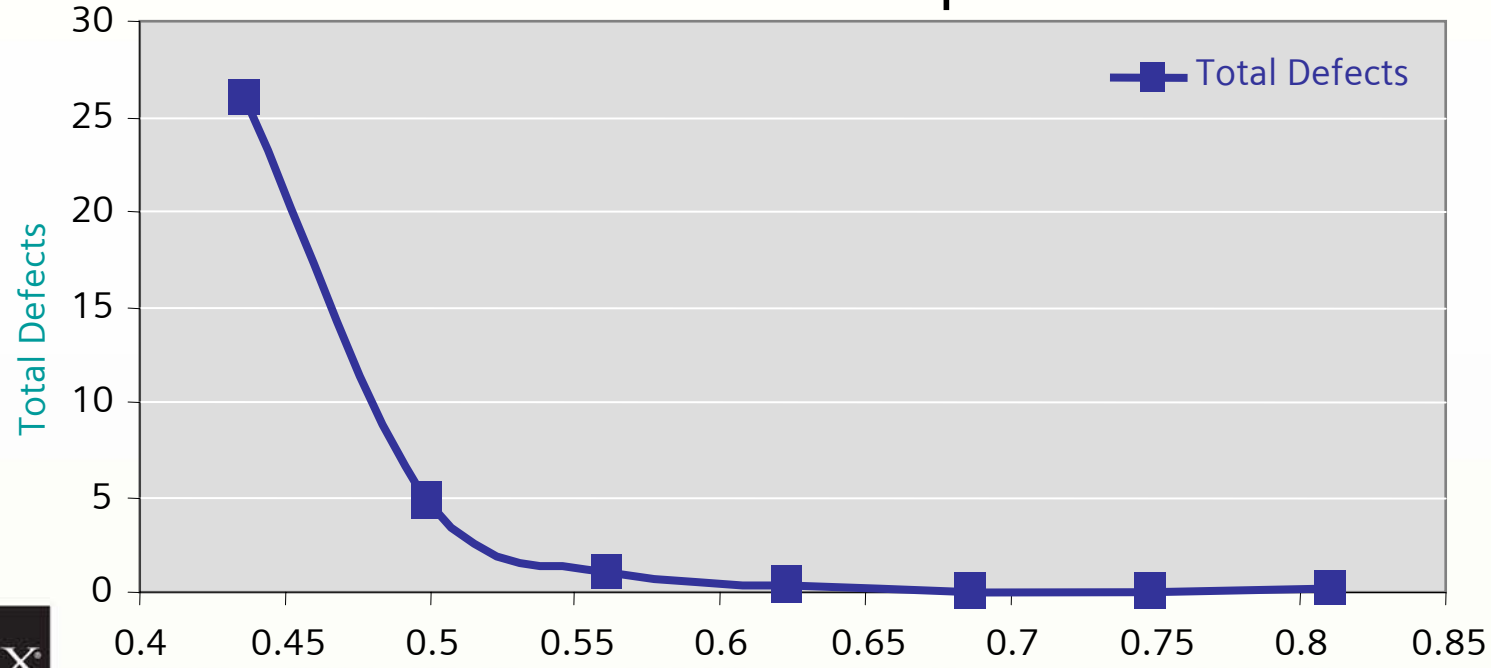
t = Stencil Thickness



An area ratio of at least 0.66 and an aspect ratio of greater than 1.5 should be used to ensure quality paste release!

Number of Defects as a Function of the Stencils Area Ratio

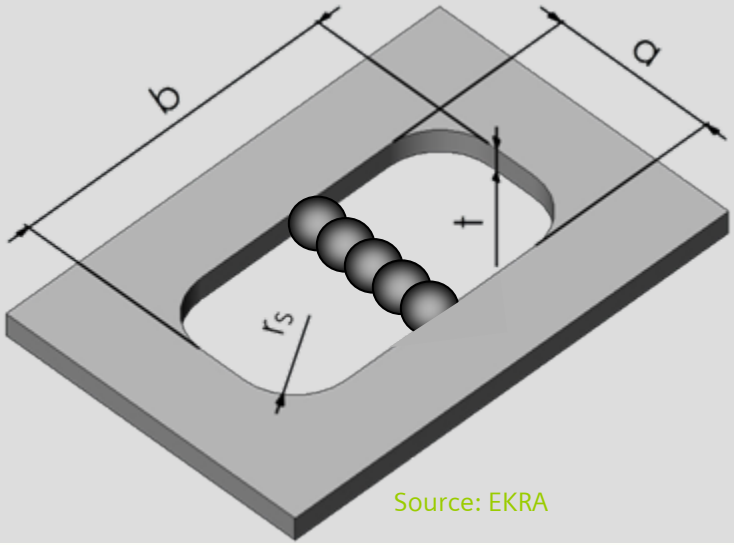
- The number of defects increases as the area ratio decreases
- The main defect for the smaller area ratios is insufficient or lack of solder on the pad



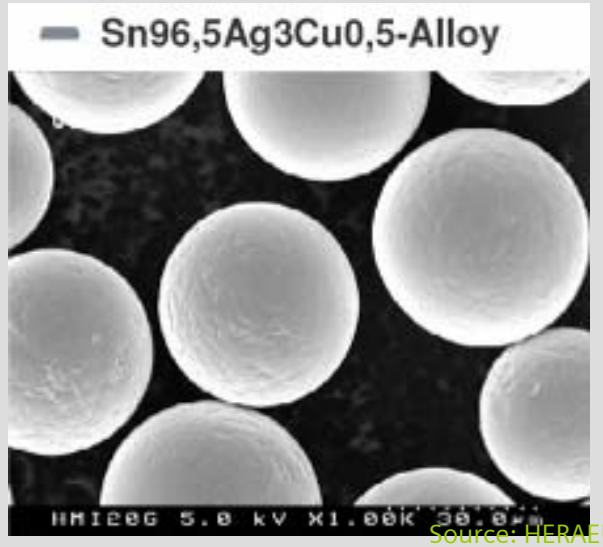
How to select the Solder Paste Type

Paste Selection

In the smallest dimension (a) of an opening at least 5 solder balls should fit next to each other in a line

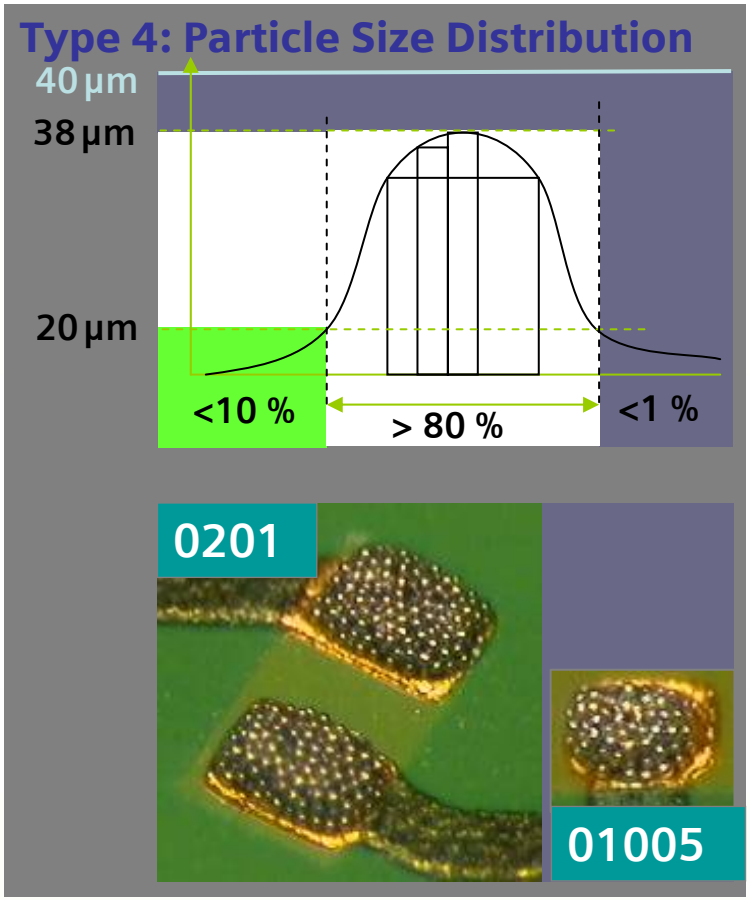


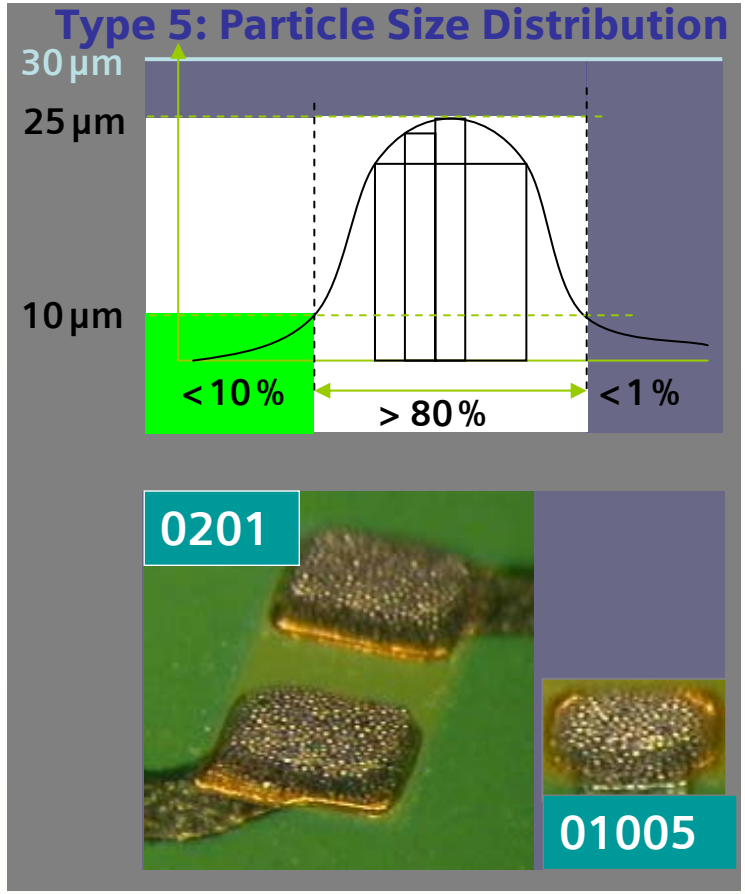
Source: EKRA



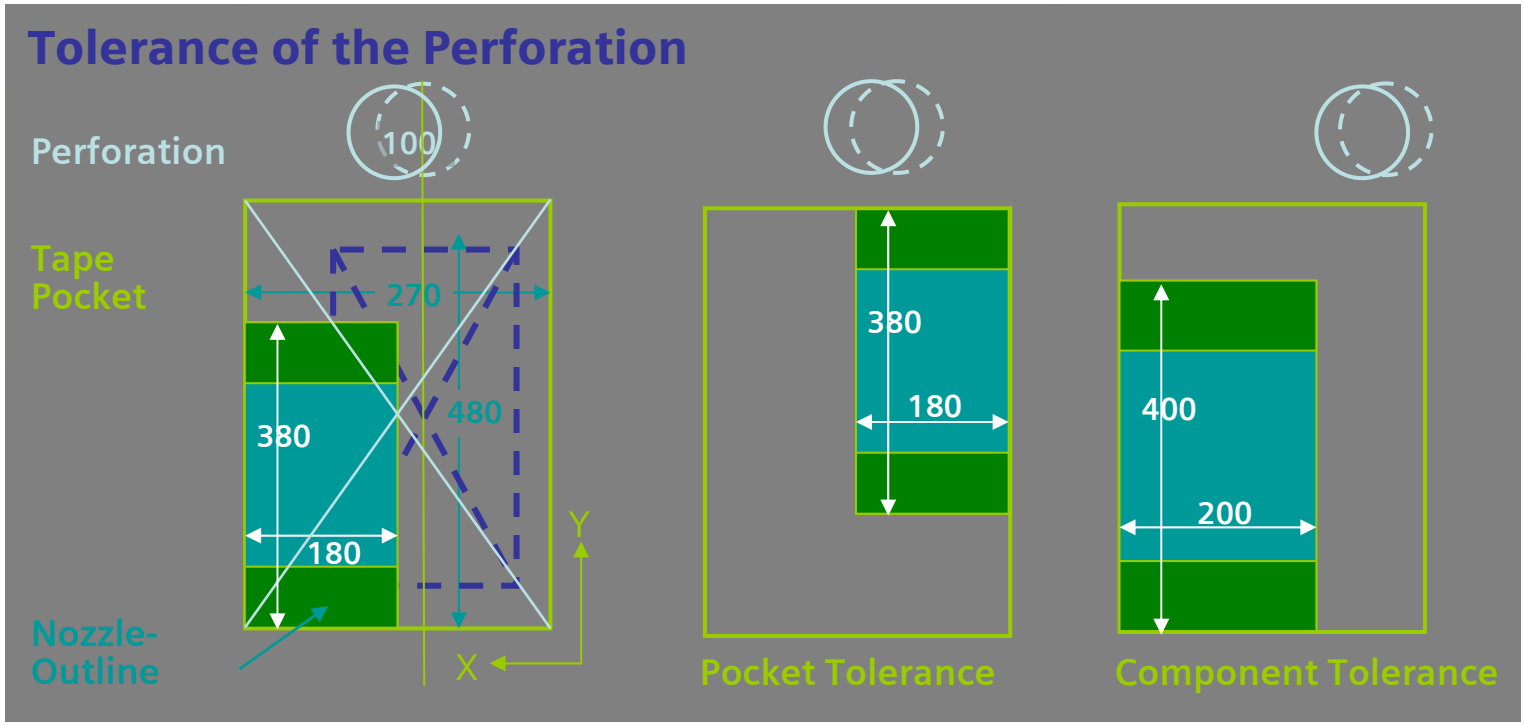
Source: HERAEUS

Comparison of Solderpaste Types



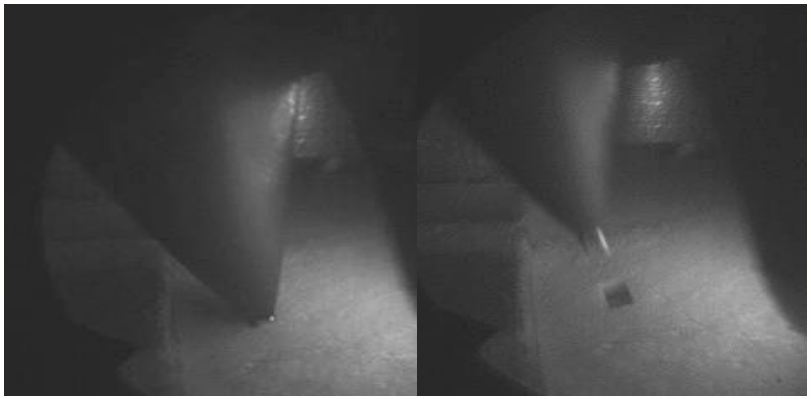


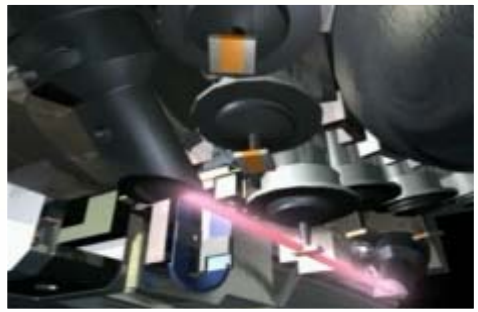
Placement: Tolerances of Components and Tapes

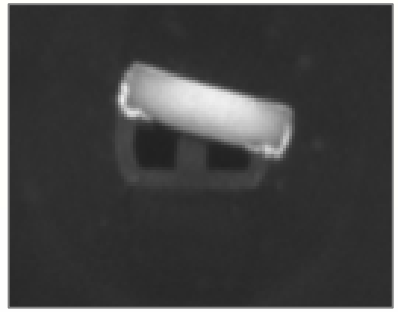


–Worst-Case: All Tolerances accumulate.
Result: Overlapp between Component/Nozzle (X:) 10 µm (Y:) 300 µm instead of 100 % only 5 %.

Placement: Pickup, Presence Check of Components, Vision







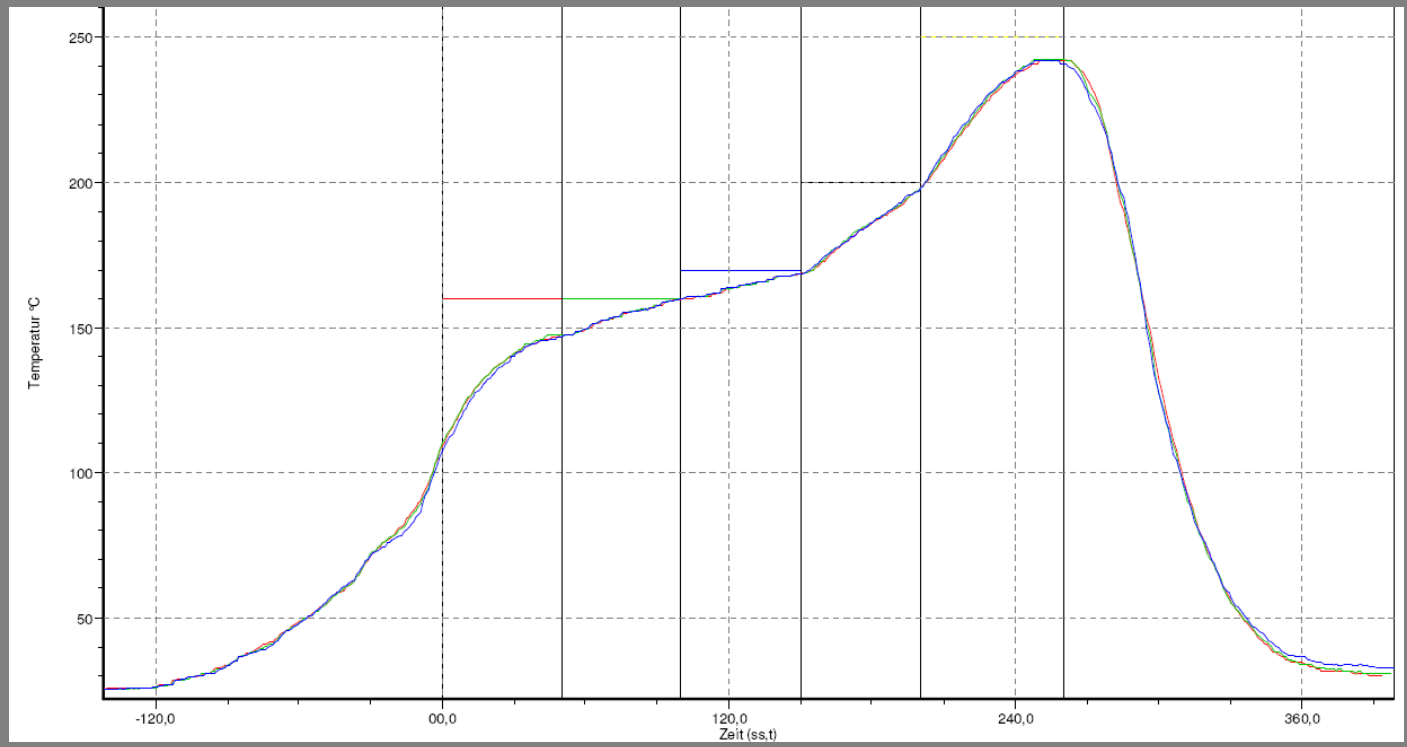
<u>Pickup of the component</u>
- component quality
- tape Quality
- feeder accuracy
- P&P machine accuracy

<u>Verify presence</u>
- high resolution
- check at the right time / location

<u>Vision</u>
- high resolution
- smart algorithm
- high speed

Reflow Profile

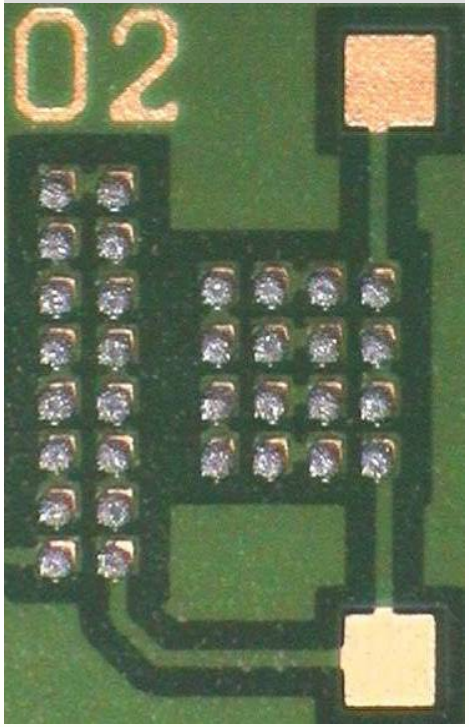
Example:



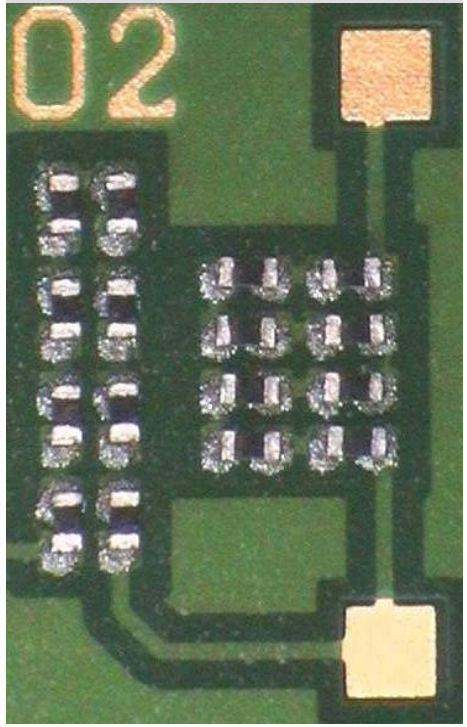
Lead Free, Type 5, Nitrogen

01005 Self - Alignment – Study

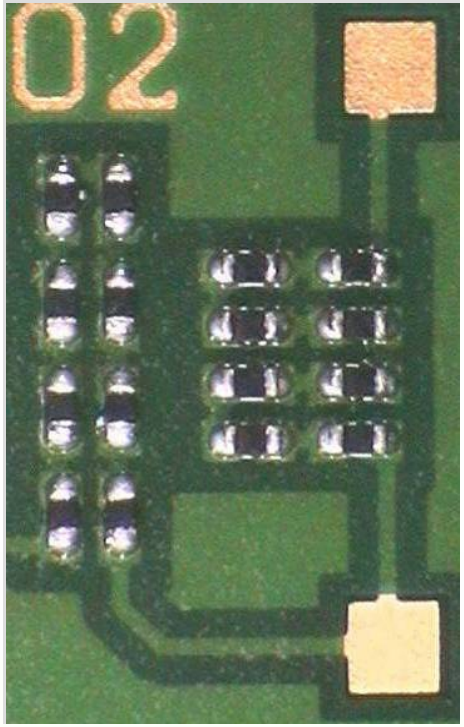
Paste printed with 50 µm offset



01005 placed with no offset



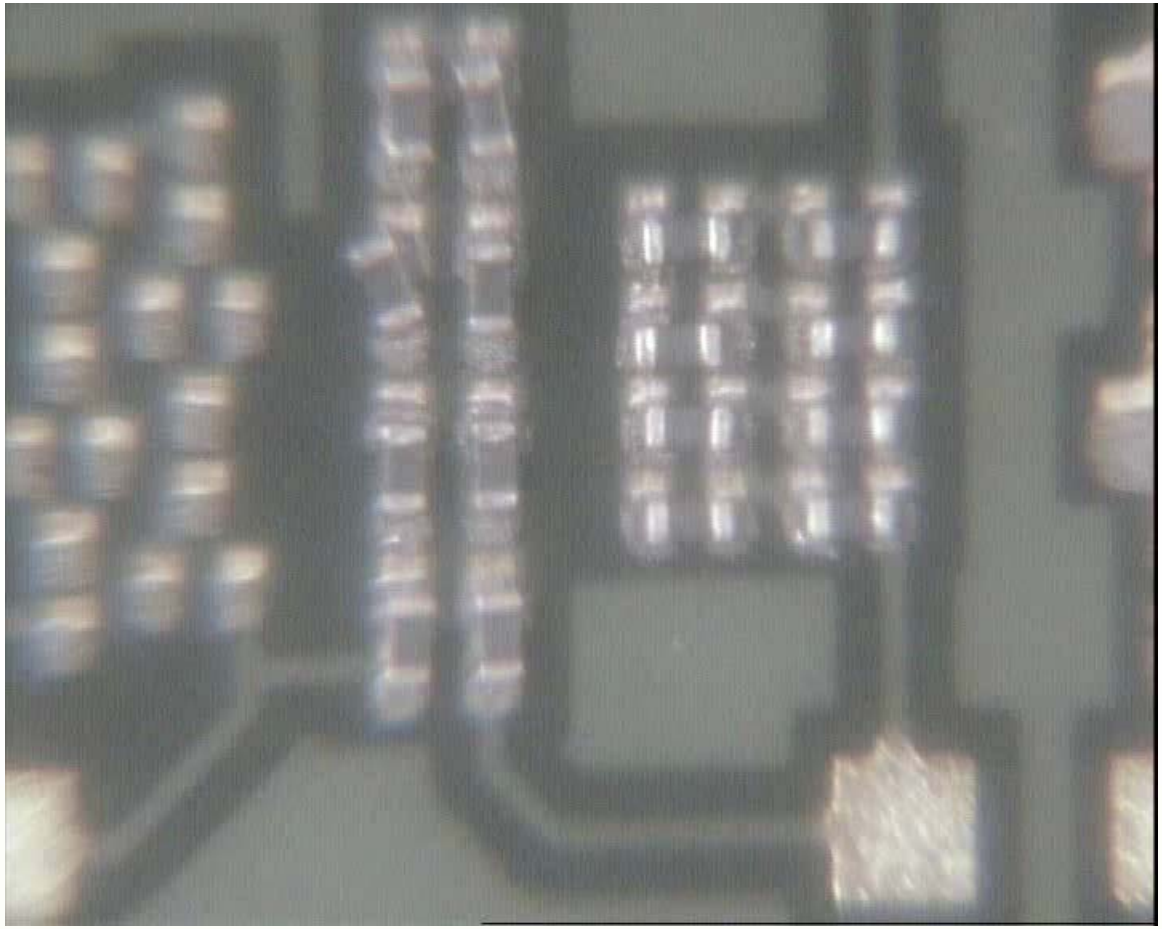
01005 self-aligned after reflow



Pad size 200 x 180 µm², 75 µm paste

Paste printed with offset, Placement centered to pads

Self-Alignment / Leadfree Solder



Summary

- Shrinking dimensions from 0201 to 01005 challenge us in all process steps
- Board and pad design needs to be done carefully, always with all the process steps in mind
- Narrower window for the component mix on a product
- Solder paste application seems to be a big challenge
- The accuracy and the „supporting features“ of a placement machine are getting more crucial to the Yield of a line
- Using 01005 in products makes them more expensive
- 01005 will be limited to certain applications

Thank you for your attention

SIEMENS Industry Sector, DT EA CRM PRD,

Dipl.-Ing. Norbert Heilmann

e-mail: norbert.heilmann@siemens.com

internet: www.siplace.com