

Developing the 01005 stencil printing process

The 01005-chip component package, ranging from 0.10 x 0.304mm (0.004" x 0.012") to 0.20 x 0.40mm (0.008" x 0.016") depending on the supplier and if the 01005 component is a resistor or a capacitor, is dramatically smaller than the previous smallest component package used for resistors and capacitors, the 0201 (0.60 x 0.30mm (0.0236" x 0.0118") component package.

The use of such a miniature component presents several significant challenges solder paste printing in the areas of solder paste selection, printed circuit board pad size determination and stencil aperture design. This paper will use the latest available information and our experience to explore the issues with the development and optimization of the 01005-assembly process.

*Joe Belmonte and
Srinivasa Aravamudhan
Speedline Technologies
Franklin, Massachusetts USA*

Keywords:
**Stencil printing, 01005
components, 01005
assembly.**

Introduction

It seems like just a short time ago (actually it was a short time ago!) that the 0201 component size was introduced, and the industry was scrambling to figure out how to assemble such amazingly small components. No one would ever conceive of using a smaller component, would they? Well, now the industry is considering how to assemble a significantly smaller component, the 01005-package size - even while many operations are still developing and optimizing the 0201 component assembly processes.

Over the last few months we have spent a good deal of time searching all our available sources to learn what we could about the 01005 assembly process. What we discovered is that there is not a great deal - actually very little - of publicly available details on assembling 01005 components. We suspect that some of the leading edge assemblers have done or are working on some in-house proprietary 01005 assembly process development work that, hopefully, will become public information over the next few years. Fortunately, we were able to get enough viable information to create a 'starting point' for the development of a 01005 assembly process, specifically with regards to solder paste printing.

Given the cost of and assembly problems associated with these extremely small 01005 components, they are being used very selectively at

the current time, primarily on leading edge portable products where size and/or weight are key selling features.

In researching this article, we made one attempt to purchase a small quantity of 01005 components. The price we were quoted was many hundreds times more than an 0402 component. Of course, at some point in time, prices will come down, but given the process issues, at least for the foreseeable future, they will be designed only into products that require the ultimate size reduction, and thus sales volumes of these components will keep prices high.

Component trends

Since the inception of surface mount technology, the trend, driven by the portable product industry (cell phones, laptop computers, PDAs, etc.), has been to introduce smaller and smaller passive and active components. We now have available the smallest passive component package ever: the 01005.

We certainly know that the cell phones have had a significant increase in features and functionality while still reducing or maintaining their size. One of the factors that allow this increase in features and functionality is the introduction of miniature components, including 0201 and 01005 chip packages.

For example, significant growth in the number of cell phones to be produced over the next few years is predicted, and these new 2.5- and 3rd-generation cell phones will

incorporate more and more functions and features, requiring more capability on the same or smaller printed circuit board size. The trend for miniature components will only increase.

The industry-wide forecast for the use of passive components packages confirms the trend toward increased use of small packages.

We will certainly not see these very small packages used in all products. The cost of the component and the cost of the assembly of these components will limit them to products where size and weight are primary concerns. Over time, as the cost of the miniature components decreases and the process of assembling them is optimized, we will see their use spread.

Features of 01005 components

Construction

Construction of the 01005 components is the same as other passive package sizes.

Size

01005 components are significantly smaller (75% smaller in mass) than the 0201 components that were introduced just a few years ago. Three different package sizes are considered 01005 packages, two for resistors and one for capacitors (*Figure 1*).

Area density

The primary advantage of using miniature components, such as the 01005 chip component, is printed circuit board real estate functionality;

The **Lead-Free** Profiling Leader

Only one manufacturer makes a RoHS Compliant thermal profiler - ECD. And there's only one M.O.L.E.®, the name more people associate with thermal profiling than any other brand. Trust your Lead-Free transition work and thermal process quality management to us.

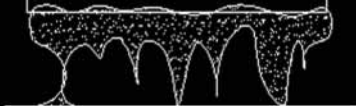
Whether you are seeking Automatic reflow oven Recipe Generation, Real-Time Data, Wave Reflow or Rework Machine Capability CpK or Continuous Oven Monitoring & Playback, ECD has a Lead-Free Compatible solution.

Choose ECD for the most comprehensive thermal profiling product line in the business.

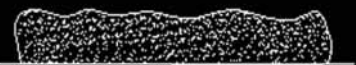


ECD, Inc. 4287-B S.E. International Way Portland, Oregon 97222-8825
www.ecd.com phone: +1 (503) 659-6100

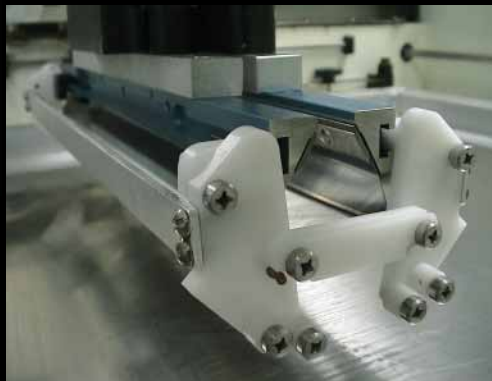
Paste Sticking
Problems?



Help is
on the way!



ADVANCES TO BENEFIT YOUR SMT PRINTER



New Integrated Wiping built into dual squeegees bring further value to the SMT process, by helping you maintain and preserve your precious solder paste and printing process.

Contact your nearest distributor, or TA direct, and we will help you be **Lead-Free Ready** and even more!

**TRANSITION
AUTOMATION**

Web: www.permalex.com Productronica: A4-154 sales@transitionautomation.com Fax (USA) : 978-670-2300

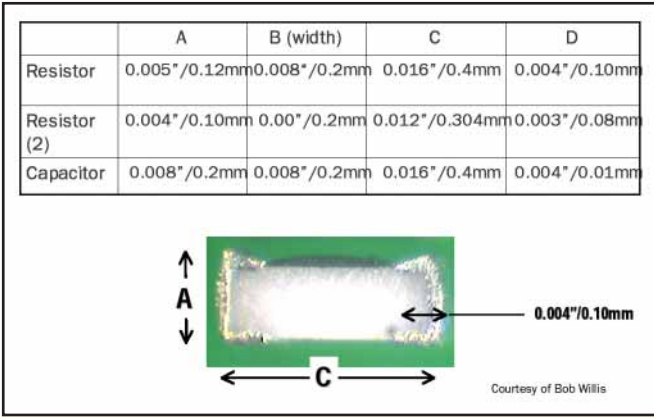


Figure 1. Notice that the metallization is only .10mm (.004") wide. (courtesy of Bob Willis)

that is, the opportunity to pack more 'stuff' into the same or smaller space. Where one hundred 0402 chip components require 5.9 x 10.9mm of space, 0201s require only 3.9 x 6.9mm- and 01005s need only 2.9 x 4.9mm! That's an 88% reduction in required real estate.

Packaging

There is nothing new in how the 01005 components will be packaged; they will come in the standard tape and reel format and use the same tape and reel size as other small chip components.

01005 packages and the solder paste printing process

Printed circuit board pad size

A number of 01005 printed circuit board pad sizes are being evaluated in experiments and tests. At this time, we know of no publicly available study that has identified the optimum 01005 printed circuit board pad size. Over the next several years, information on 01005 pad size experiments will become available.

Figures 2, 3, 4 and 5 show some of the 01005 printed circuit board pads being used in studies.

Stencil Design

The challenge is developing a solder paste printing process and stencil design that will satisfy the solder volume,

accuracy and repeatability requirements for both the smallest and the largest components on the board.

First, let us review the basics of stencil design.

Our primary concern in stencil design is transfer efficiency. How much of the solder paste that goes into the stencil aperture comes out and is printed onto the printed circuit board pad.

There are two opposing forces competing for the solder paste. One force is the adhesive force of the printed circuit board pad, to pull the paste out of the aperture, and the other force is the adhesive force of the aperture walls, to keep the solder paste in the stencil aperture. Only a well-designed stencil will insure that the solder paste will transfer or release from the stencil and consistently be printed onto the printed circuit board pad.

Two stencil design rules must be considered when designing a stencil: the Aspect Ratio Rule and the Area Ratio Rule.

Aspect Ratio is the ratio between the width of the stencil aperture and the thickness of the stencil. This ratio should be 1.5 or greater (Figure 6) to create a stencil that will print reliably all day every day with a minimum of stencil wiping. An Aspect ratio of 1.5 or greater will

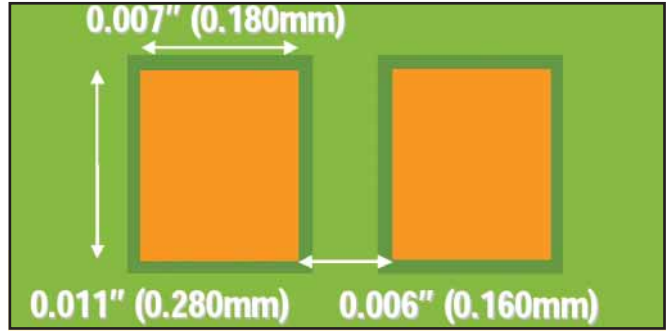


Figure 2. (courtesy of Bob Willis)

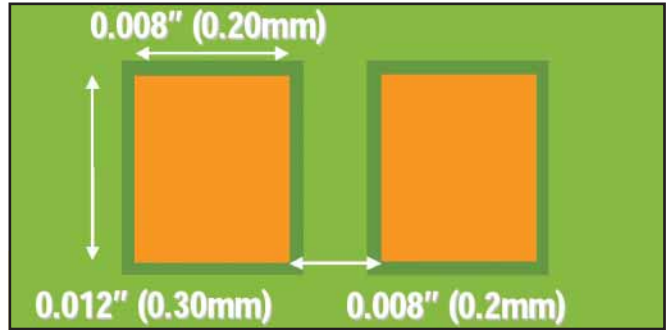


Figure 3. (courtesy of Bob Willis)

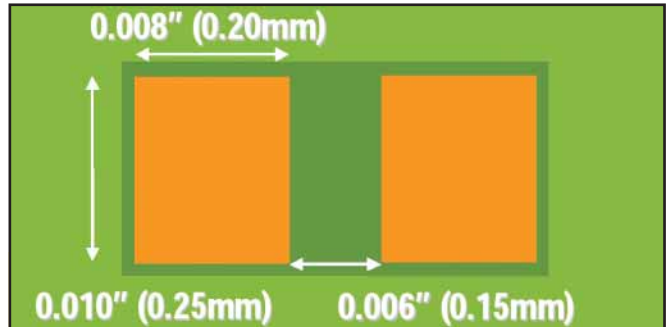


Figure 4. (courtesy of Bob Willis)

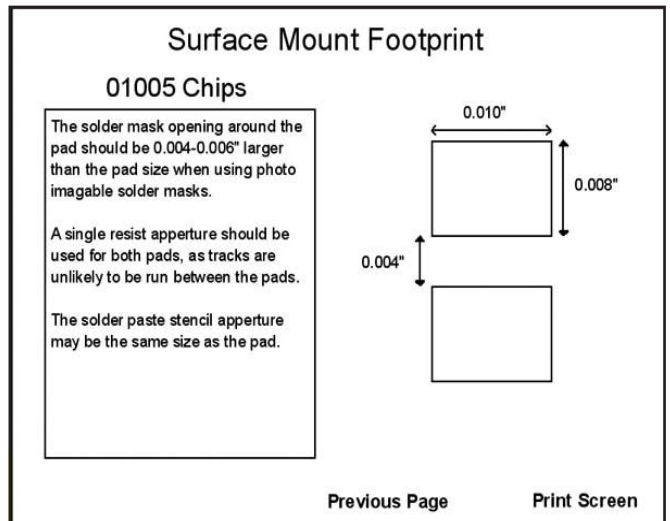


Figure 5. (courtesy of Bob Willis)

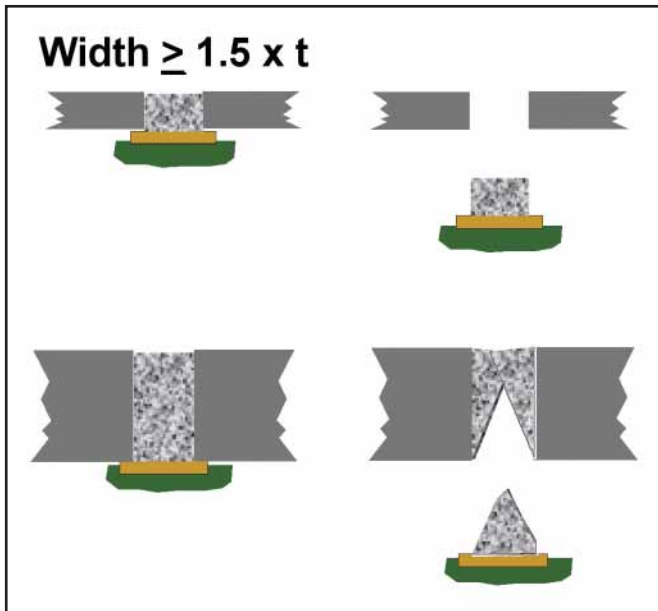


Figure 6. The top figure shows a stencil that complies to the Aspect Ratio Rule resulting in the majority of the solder paste being printed onto the pad. The bottom figure shows a stencil that violates the Aspect Ratio Rule resulting in the majority of solder paste remaining in the stencil aperture (courtesy of Bob Willis)

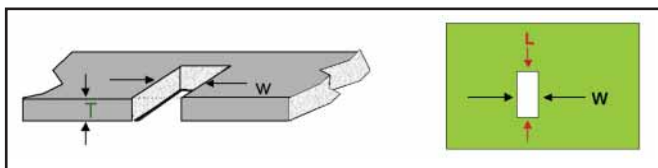


Figure 7. Shows a stencil that violates the Area Ratio Rule resulting in the majority of the solder paste remaining in the stencil aperture. (courtesy of Bob Willis)

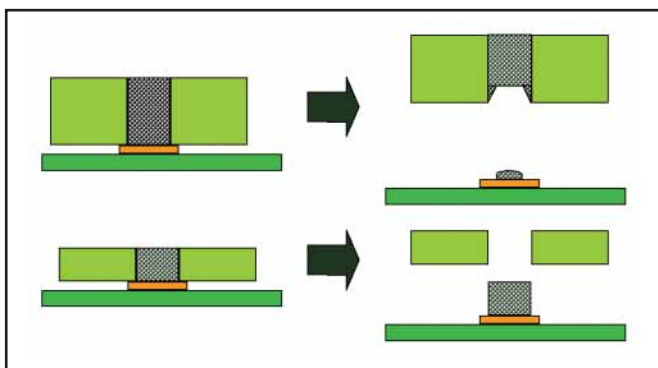


Figure 8. shows a stencil that complies to the Area Ratio Rule resulting in the majority of the solder paste being printed on the printed circuit board pad. (courtesy of Bob Willis)

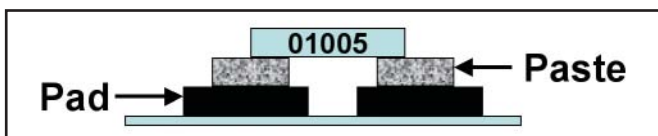


Figure 9. (courtesy of Bob Willis)

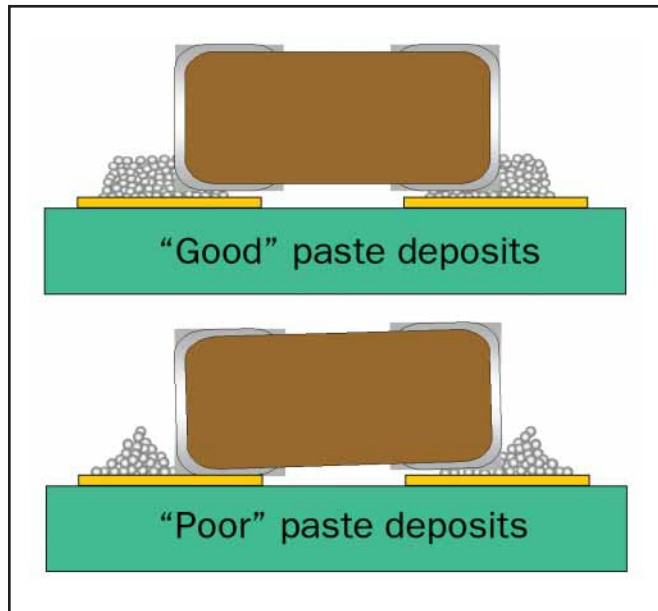


Figure 10. (courtesy of Bob Willis)

allow proper transfer efficiency of the solder paste onto the printed circuit board pad.

Until the introduction of miniature components we were only concerned with the Aspect Ratio Rule.

The stencil aperture for miniature components creates a situation where the area of the walls of the aperture can be very close to the area of the stencil opening. This situation creates a situation that the Aspect Ratio Rule cannot properly calculate. We must use the Area Ratio Rule. The Area Ratio Rule considers the ratio between the area of the opening of the aperture and the area of the aperture walls (Figures 7 and 8). This ratio must be .6 or greater to create a stencil that will print reliably all day every day.

With such small volume of solder paste that will be printed for 01005 components, it is essential that as much of the solder paste as possible be released from the stencil aperture and printed onto the component pad.

Equally as important is getting an equal amount of solder paste on each pad of the 01005 component. If we print unequal amounts of solder paste we will experience a

number of problems, including tombstoning. The key consideration is consistently printing the correct amount of solder paste equally onto each pad of the 01005 component (Figures 9 and 10).

Using a thin stencil, we can satisfy the Area Ratio Rule for the 01005 stencil aperture. However, will this thin stencil provide sufficient solder paste volume for the larger components on the board? We certainly can consider stepped or multi-thickness stencils. Stepped stencils have been in use for many years but they do have their limitations and problems, such as clearance area around the step, how much you can actually step the stencil, squeegee blade selection, etc. Stepped stencils may or may not be a solution to printing small and large volumes of solder paste on the same assembly. As we discussed, it all depends on the limits of the solder volume requirements of what you are printing.

In a study by Flextronics where three different 01005 printed circuit boards were evaluated, they had to use 76µm (0.003”) thick stencils to achieve an acceptable area ratio (equal to or greater than

0.6) for two of the three considered pad designs. The third considered pad design did not achieve an acceptable area ratio even with a 76µm thick stencil.

Will the 76µm thick stencil provide sufficient solder paste for the larger components on the same board? Will a stepped stencil work for this assembly? Each product will have to be evaluated independently to determine its correct stencil design.

One alternative to consider is the two-stencil/two-printer process. The first stencil would be thin, to comply with the area ratio rule for the miniature components. The second stencil would be thicker, to provide sufficient solder paste volume for the larger components. The second stencil would have material removed from the bottom (board side) to clear the solder paste already printed with the first/thin stencil. This dual-stencil printing process, adopted to satisfy the requirements of pin-in-paste, where a large volume of solder paste is required for through-hole components but much less for fine pitch SMT components, has been in use by some operations for a number of years. With well-designed stencils and a well-designed and well-controlled process, the dual-stencil process works very well. The obvious disadvantages are the cost of two stencils and two solder paste-printing machines.

Other factors impacting solder paste printing

A number of other factors impact the 01005 solder paste printing process. We must consider stencil fabrication (laser cut, chemical etch, electro fab, etc.) solder paste selection, printed circuit board fabrication and finish (OSP, Nickel Gold, Immersion Silver, etc.), printed circuit board support during printing, operating parameters

(squeegee speed, squeegee pressure, etc.), squeegee selection, operator training, etc.

All of these factors are discussed in a number of papers, article and books. It is vital that you perform the formal process development work to insure you are optimizing all of the factors that contribute to the performance of the solder paste printing process.

01005 assembly defects

As we have discussed, a number of factors can contribute to 01005 assembly defects. Some are common to all components, and some are unique or more prevalent because of the size of the 01005 components. We know of no high-volume data available on 01005 placement defect rates, but some process development and experimentation work has identified some areas of potential problems.

Figure 11 illustrates several soldering problems, including too much solder, insufficient solder, and tombstoning, that can be caused by improper solder paste printing, improper 01005 placement, improper reflow profiling, etc.

Introduction of lead-free materials

At the same time we are introducing miniature components, we are introducing lead-free materials. Either one is a challenge in itself. Together they dramatically reduce the process tolerance or, as it is commonly called, the 'process window'. We know that lead-free solders do not spread as well, which can lead to the pad finish being exposed after reflow. To counteract this, some companies have increased their stencil apertures. IPC610D, however, suggests that incomplete coverage is merely a process indicator, not a defect.

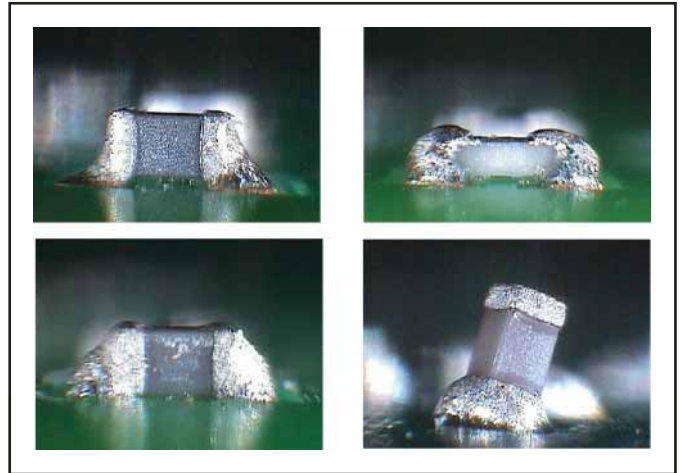


Figure 11. (courtesy of Bob Willis)

Conclusion

It is evident that the 01005 components will be introduced in specific size- and/or weight-driven products over the next few years. When will your company's or your customers' products introduce 01005 components? It is essential that you understand your company's and/or your customers' product design roadmap and when/if they will incorporate 01005 components. It is then time to 'roll up your sleeves' and start the formal, detailed process development work to fully understand the impact of 01005 components on your stencil printing process. Is your current soldering paste-printing equipment capable of the required accuracy? If so, what modifications may be required? What preventative maintenance and calibration schedule is required to keep your equipment at optimum performance?

You will also have to address training and process discipline issue. Everyone must know what to do and do it on all shifts, every day. All operators, technicians, engineers, etc., must be well trained and well disciplined in all they do to insure optimum process performance.

Finally, work with your equipment and material suppliers to understand how

they are addressing the 01005 component assembly issues, as they are an excellent source of information.

References and acknowledgements

- Bob Willis, for all the information, pictures, and graphics he so kindly provided, www.bobwillis.co.uk
- "Board Design and Assembly Process Evaluation for 0201 Components on PCBs", by: Mei Wang, Dr. Dongkai Shangguan, David Geiger, Kazu Nakajima, CC. Ho, and Sammy Yi, Flextronics, San Jose CA... IPC SMEMA Council APEX, 2002
- "PCB Design and Assembly Process Study of 01005 Size Passive Components Using Lead Free Solder", by: Fredrik Mattsson, Flextronics, Linkoping, Sweden.... David Geiger and Dongkai Shangguan, Ph.D., Flextronics, San Jose, CA, USA... Todd Castello, Flextronics, Youngsville, NS, USA... SMTAI Show, 2004
- "SMART Group Lead-Free Experience 3" www.smartgroup.org/ev_nepcon2005b.asp