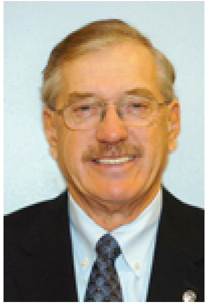

By: Dr. Bill Coleman

The Small Component Printing Challenge

Small components are now or will soon be available that present challenges to the solder paste printing and SMT assembly process. These challenges exist for (1) tools: including stencils, squeegee blades, and under board support: (2) materials: including solder paste and the PCB; and (3) equipment: including stencil printing equipment and pick and place equipment. Although not all-inclusive I like to consider the challenges in two categories: (1) positional accuracy and (2) the solder paste printing process.

But first, what are some of these very small components and why are they troublesome?

- .3 mm pitch CSP devices
- .3 mm pitch QFN devices
- 01005 passive devices

Positional accuracy is a challenge for PCB, the stencil, the pick and place, as well as the stencil printing equipment. The devices above typically have pad sizes on the PCB in the 125-micron (5 mil) to 200-micron (8 mil) range. The positional tolerances add up, each of the above mentioned sources have an effect. With a 125-micron (5mil) pad width it is not difficult to imagine printing a solder brick up to ½ pad off center. This can definitely cause shorts, opens, and tombstoning. The problem is less severe for small PCB where the small devices are relatively close to each other. The problem becomes more severe for larger boards where the small components are spaced far apart. What are some solutions for the positional accuracy issue? In my opinion PCB suppliers and stencil suppliers will need to supply positional accuracy data with the delivered product. After reviewing the positional data it may be necessary for the SMT process engineer to go back to the stencil supplier and have him scale the Gerber data to match the PCB.

The solder paste printing process is a challenge because the small components coexist on the PCB with more conventional SMT devices like .5 mm QFP's and 0603 and 0805 passive devices. These larger devices require more solder volume to achieve good solder fillets compared to the smaller components. Overprinting (making the stencil aperture larger then the PCB pad) is an option but has limitations due to lead density. To achieve acceptable fillets (sufficient solder paste volume) for the normal SMT devices normally requires a stencil thickness of at least 100-micron (4 mil). However, a stencil of this thickness presents a printing challenge for the small devices with apertures in the range of 125-micron (5 mil) to 200-micron (8 mil). Stencil printing is a two-phase process. First, the solder paste is forced into the stencil aperture as a squeegee blade wiped the solder paste across the surface of the stencil. Second, the solder paste is transferred from the stencil aperture to the PCB pad as the PCB is separated from the stencil. As I have mentioned previously, I like to think of this paste transfer process as a tug of war: the aperture walls are trying to hold back the solder paste within the aperture while the pad on the PCB is trying to pull the paste out of the aperture. The ratio of the inside aperture wall area to the area of the pad beneath the aperture opening determines who will win this tug of war. Over the years the stencil solder paste printing industry has established some guidelines for this area ratio (area of pad under the opening of the aperture / area of the aperture walls. A ratio of .66 was established standard stencils and recently in IPC 7525 rev A a ratio of .5 was established for certain special stencils with smooth aperture walls. However the area ratio for small component using a 100-micron (4 mil) thick stencil is in the range of .4 to .5 well below the recommended range. Here-in lies the challenge. Recently at SMTAI 2009 6 technical papers were presented in two sessions dealing with this challenge. Session 32 at APEX 2010 has 3 papers also dealing with this subject. I believe there are two options to resolve the printing challenges: Step Stencils or reduction of the acceptable area ratio down to the .4 range. The step stencil is a brute force solution and has definite limitations, namely the spacing between apertures that lie in the step to apertures that lie out of the step. Using a two-print step stencil, aperture spacing (1st print aperture to 2nd print aperture) as low as .4 mm (16 mil) has been reported[1]. Work continues in several areas to reduce the acceptable area ratio to .4. Solder paste materials, special squeegee blade systems, and stencils with special coatings on the aperture walls all have potential to contribute to the final solution.

1. "Stencil Design when 01005 and .3 mm pitch uBGA's coexist with RF Shields", William E. Coleman, S23 Paste and Printing II, APEX 2009.