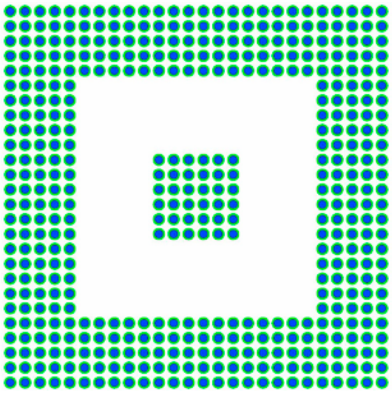
Problem Description:

MicroBGA component U1 is having release problems at print and subsequent insufficient solder at reflow. Filter U29 is having some issues with unsoldered leads at reflow

Top
MicroBGA 456-pin: U1
Filter 12-pin: U29

Root Cause:

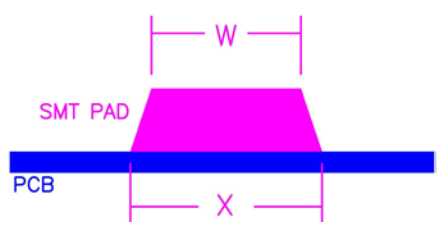


µBGA 456-pin
(blue: SMT pad, green: soldermask)

µBGA 456-pin:
SMT pad size in the Gerber data: 0.0102” round
Expected size of the SMT pad on the physical PCB: 0.0087” (with 0.38 oz Cu, the size at the top of the SMT will be approximately 0.0015” smaller than the size in the Gerber data)

Actual surface area ratio with 0.004” foil and a 0.0087” round SMT pad size: **0.54**

On the physical PCB, the actual sizes of these land pads will be smaller than what is specified in the board files. The etching process at the board manufacturer produces a trapezoidal etch (see figure to the right) where the top of the land pad is smaller than the bottom. The board manufacturers take their measurements at the bottom/base of the copper because this dimension is critical when controlled impedance designs are produced. For the assembly house, the top of the land is critical since the adhesion of the solder paste to the top of the land pad is what pulls the solder paste out of the stencil. The typical size reduction is 0.0015” for 0.38 oz Cu.



When calculating the surface area ratio, one needs to use the horizontal surface area of the physical SMT pad that will be in contact with the solder paste and not the horizontal surface area of the stencil aperture. In this case, the stencil aperture is a 0.011” square with rounded corners. The surface area ratio with a 0.011” square and 0.004” stencil foil is 0.69. Using the SMT pad size of 0.0102” round and assuming no reduction due to etching, the surface area ratio on 0.004” foil drops to 0.64. The surface area ratio limit for laser-cut standard stainless steel is 0.66. If alignment between the stencil apertures and the PCB were perfect, a standard stainless steel, laser-cut stencil might work. However, all of the SMT pads on the PCB will never line up perfectly with the stencil apertures. Any misalignment between the stencil and PCB and less of the solder paste contacts the SMT pad. This reduces the surface area ratio even further.

Since the size at the top of the SMT pad, in 99% of cases, will not match the size in the electronic board files, the actual surface area ratio will be lower than 0.69 and 0.64. Using an expected SMT pad size of 0.0087” and perfect alignment, the actual surface area ratio is 0.54. This is too low for a laser-cut, standard stainless steel stencil, even with perfect alignment, and is the cause of the poor solder paste release.

Filter 12-pin:
For this component, the customer has communicated that the cause of the unsoldered leads is due to varying heights in the gull wing leads. Some sit too high off of the land pad surface and little-to-no solder touches the leads.

Recommendation:

µBGA 456-pin: Keep the 0.004” step thickness, but switch the stencil material to UltraSlic™. The minimum surface area ratios for stencil foil materials is as follows:

Standard Stainless Steel:	0.66
Fine Line Stencil Slic™:	0.55
Electroformed:	0.50
Fine Line Stencil UltraSlic™:	0.45

The minimum surface area ratio of 0.45 for UltraSlic™ material is low enough to provide excellent solder paste release on U1 and it provides a little cushion to allow for misalignment between the stencil and SMT pads. The UltraSlic™ material is stainless steel, but is specially formulated for extremely low surface area ratios. It can be used in place of a standard stainless steel, laser-cut stencil with no required process changes on the assembly line.

Filter 12-pin: The best option is to use a foil thickness of 0.006” at U29. However, the component density around U29 is high and printing the surrounding components would be difficult. There are also 0402 components in between the leads of U29. Both of these prevent a 0.006” foil thickness option at U29.

The second best option is to make the entire stencil 0.004” UltraSlic™. This would solve the release issues with U1 without a step. Since the reduction from 0.005” foil to 0.004” foil creates a 20% reduction in solder paste volume, all of the apertures, with the exception of U1, would be increased by 20% volume. For leaded devices, the volume increase is applied to the aperture length and not the width. For chips, the volume increase is applied to the 3 outer sides and not the inside edge. For leadless devices, the volume increase is applied to the length and only on the “toe” end. Each aperture is manually verified that it is not printing solder paste onto exposed vias our any other solderable surface. However, additional height is needed at U29 and switching to 0.004” foil would only make the problem worse.

The most viable option is to increase the solder volume of the U29 apertures and keep the 0.005” foil thickness. Given the SMT land pad size and the foot size of the gull wing lead, the volume of the U29 stencil apertures will be increased by 20%.

The recommendation is to keep the 0.005” foil thickness and the 0.004” step for U1. However, the recommended stencil material is UltraSlic™. For U29, the volume will be increased by 20%.

Another recommendation is to use “U-shaped” apertures for the chip components rather than standard homeplates. From a physics standpoint, the U-Shape is much better at preventing solder balls and tombstoning. Two checkplots are submitted. One has the standard homeplates supplied by the customer and the other has the U-shaped homeplates. Either will work, but the U-shaped homeplates have shown significant improvement in eliminating solder balls and preventing tombstoning.