SMT ASSEMBLY CHALLENGES AND PROVEN SOLUTIONS FOR IMPROVING YIELDS

Robert Dervaes
FCT Assembly
Greeley, CO, USA
rdervaes@fctassembly.com

ABSTRACT

Surface mount technology (SMT) assemblies are getting more complex as advancements in the areas of printed circuit board (PCB) manufacturing and component design become more main stream. Most SMT manufacturing processes have to now be capable of building "hybrid" assemblies, which contain both previous generation technology and more cutting edge technological advancements. Increasing SMT assembly yields is a must, but it is getting more and more difficult to just maintain yields, let alone increase them, as new technologies continue to be introduced.

There are many variables in the SMT assembly process. Some, like SMT assembly equipment, process parameters, and personnel, are directly controlled by the SMT assembly company. Some, like PCB layout, PCB fabrication, component availability, component substitution, and solder paste stencil fabrication, are provided either by the end customer or through suppliers. There are many times where everything in the SMT assembly process is within specifications, but SMT defects still occur and yields are not as expected. What challenges can those in SMT assembly expect to face when building these more advanced assemblies? What can be done, if anything, to reduce SMT defects and improve process yields?

This paper will present the most common SMT yield challenges being confronted today, discuss their root cause, and then provide solutions that are proven, repeatable, and economical.

Key words: Root cause analysis, yield improvement, SMT defects, stencil, solder paste release, reflow

INTRODUCTION

100% yield is the holy grail of the SMT assembly world. Achieving this coveted prize is extremely difficult. Keeping it is even more difficult as the drive to pack "more power" into smaller devices is an important part of technological advancement.

The majority of electronics today utilize, to some extent, surface mount technology (SMT) components. Without SMT, the reduction of component sizes, and product sizes, would be extremely difficult. However, reducing the component sizes increases the complexity of putting them onto printed circuit boards (PCB). The smaller the size, the more difficult it can be to print solder paste onto the PCB, precisely place the component, and send it through the reflow oven with no problems.

There are a myriad of SMT defects (insufficient solder, bridging, solder balls, voids, etc.) that SMT assembly has to try and prevent or overcome. While building millions of the same product certainly provides more opportunities to improve yields, a 100% yield is never a guarantee. In some cases there is only one SMT assembly production run and only one shot at trying to get everything right.

Common "universal" SMT defects are poor solder paste release at print, bridging at print, insufficient solder volume at reflow, and bridging at reflow. Root causes of each of these common problems will be presented, along with proven, repeatable solutions to dramatically reduce, and many times eliminate, these particular SMT defects.

EXPERIMENTAL METHODOLOGY

Determining the root cause of one specific SMT defect, on one specific assembly, using specific SMT equipment can be achieved through a carefully controlled experiment. The results can help determine a solution for this one particular assembly, but this solution may or may not be applicable to any other assemblies without further testing.

Determining the root cause of a "universal" SMT defect, that is more assembly independent, involves testing a very wide array of assemblies over a much longer period than can be fit into one experiment. Variations in PCB layout, PCB copper weights and surface finishes, SMT components, SMT equipment, process parameters, environment, etc. have to be considered and evaluated to determine if defects are specific to one assembly, one customer, one SMT assembly line, etc., or would occur regardless of where a product is built.

For this paper, 1,000 different SMT assemblies were analyzed, over a 4 year period, to determine the root cause, and solution, of specific SMT defects. These assemblies were not tested in the same environment, but were real production assemblies, at hundreds of different SMT assembly customers, located all over the world. Many of the defect types were the same, but the SMT equipment and process variables were completely different and independent.

To determine the accuracy of the root cause analysis and the effectiveness of the solution, customers reported on how well the recommended solution worked at reducing, or eliminating the reported SMT defect(s). In the majority of cases, the feedback was as simple as an email from the customer indicating the SMT defect was completely eliminated. When the defect was dramatically reduced, but not eliminated, quantifiable data was sometimes provided. In all cases, the customer had the ability to determine whether or not the recommended solution had a measurable impact on improving production yields.

The sections that follow are the root causes of common SMT defects that have been validated by hundreds of independent customers on 1,000 different assemblies. The defects can be dramatically reduced, eliminated, or prevented using the recommended solutions.

RESULTS

Poor Solder Paste Release

Root Cause

The two methods of determining solder paste release at stencil print stencil are the *aspect ratio* and *surface area ratio*. The first, *aspect ratio* (Figure 1), compares the smallest dimension of the stencil aperture (w) to the thickness of the stencil foil (t). Typically, the lowest acceptable aspect ratio is 1.5. While aspect ratio is a valid determination of potential stencil performance, it is limited to simple shapes like squares, rectangles, and circles.

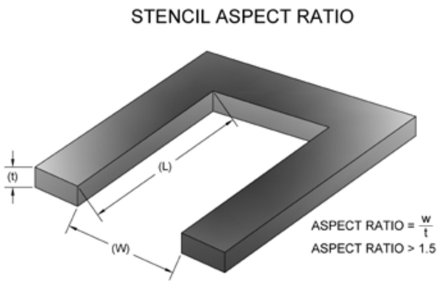

Figure 1: Aspect Ratio

A more accurate, and more detailed, method of determining potential stencil print performance is the *surface area ratio* (Figure 2). It can be used on any stencil aperture, regardless of the shape, and compares the surface area of the stencil aperture (LxW) to the surface area of the stencil aperture walls ((2L)+(2W))x(t). Typically, the lowest acceptable surface area ratio is 0.66.

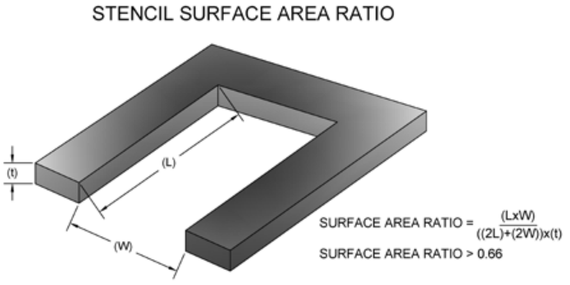

Figure 2: Surface Area Ratio

The current stencil *aspect ratio* and stencil *surface area ratio* only consider the stencil thickness and stencil aperture size when predicting solder paste release. The printed circuit board (PCB) is not considered as a factor, but it is the adhesion of the solder paste to the SMT pad that pulls the solder paste out of the stencil. The adhesion strength of the solder paste to the SMT pad is determined by the size of the SMT pad, and the corresponding surface area. Differences in copper weights and surface finishes will produce differences in SMT pad sizes and these differences are becoming critical as miniature components become more main stream.

Printed circuit boards typically have a pre-finished copper weight of 0.38 oz, 0.5 oz, 1.0 oz, or 2.0 oz on the outer layers. The PCB manufacturing process consists of etching the copper to produce the circuitry (traces, SMT pads, via pads, etc.). The copper etching process will produce an SMT pad cross section (Figure 3) where the top of the SMT pad will be smaller than the bottom.

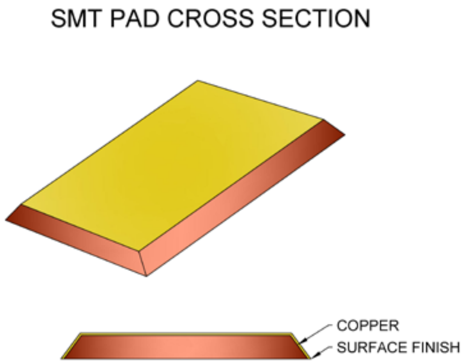

Figure 3: SMT Pad Cross Section

In most cases, the bottom of the SMT pad will match the size in the electronic PCB files, due to impedance requirements and IPC requirements of where measurements are to be taken during PCB manufacturing. The top of the SMT pad will be smaller and the difference in size, from bottom-to-top, is determined by the thickness of the copper, i.e. copper weight. The smaller top size must be used in determining the stencil *surface area ratio* since the smaller size has less surface area for generating solder paste adhesion.

Solution

To accurately determine solder paste release at print, and fix/prevent potential solder paste release problems, the PCB configuration (copper weight and surface finish) must be a part of the *surface area ratio* formula. The result is a *modified surface area ratio* formula that compares the surface area of the SMT pad (at the top) and the surface area of the stencil aperture walls (Figure 4).

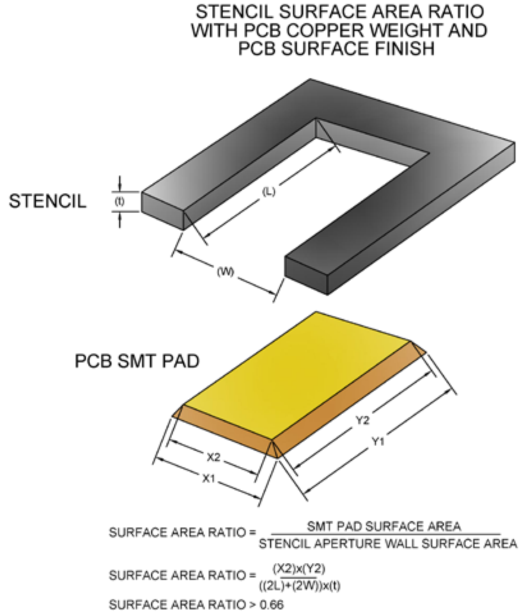

Figure 4: Modified Surface Area Ratio

This modified comparison takes into account changes in PCB SMT pad sizes due to variations in copper weights and surface finishes. Heavier copper weights will produce larger size differences from bottom-to-top. Flat surface finishes, like ENIG, OSP, immersion Ag, and immersion Sn, allow the solder paste to stick to the entire top surface of the SMT pad (assuming PCB-to-stencil alignment is accurate) for maximum adhesion strength. Non-flat surface finishes, like HASL and HAL, have a more domed/irregular surface and the semi-rounded surface makes it more difficult for the solder paste to stick to the entire surface. This will reduce the adhesion strength between the solder paste and SMT pad. Typical SMT pad size reductions, due to copper weight and surface finish, are shown in Table 1.

Copper Weight (oz)	Copper Thickness (µm)	Size Reduction (ENIG, OSP, Ag, Sn), inches	Size Reduction (HASL & HAL), inches
0.38	13	0.0015	0.0035
0.50	17.5	0.002	0.004
1.00	35	0.003	0.005
2.00	70	0.004	0.006

Table 1: SMT Pad Size Reductions

Utilizing the SMT pad size reductions in Table 1 will provide a much more realistic representation of what SMT pad sizes to expect on physical PCBs. This data, along with the *modified surface area ratio*, provide a much more accurate prediction of solder paste release at print. Results can confirm current problems with solder paste release, as well as prevent future problems. It is reliant upon SMT pad sizes in the PCB files, copper weight, and surface finish, but can be applied to any assembly, regardless of where it is built.

Bridging at Print

Root Cause

As discussed in the previous section, PCB copper weights and surface finishes can have a major impact on solder paste release. However, these changes in SMT pad sizes are not limited to affecting just solder paste release. They can also have an impact on another common SMT defect – bridging.

The solder paste print process relies heavily on a good seal, or gasket, between the PCB and stencil. This seal is very much dependent on the size of the SMT pad and the size of the stencil aperture. While the size of the stencil aperture does not have to be smaller than the SMT pad to create a good seal, stencil apertures that are larger than the SMT pad make it more difficult to prevent solder paste from squeezing out between the PCB and stencil. Large SMT pad size reductions, due to heavy copper weights and/or non-flat surface finishes, can easily degrade this seal and allow solder paste to squeeze out during print (Figure 5). In some cases, enough to cause bridging at print. In others, not enough to cause bridging at print, but enough extra solder paste volume to cause bridging at SMT reflow.

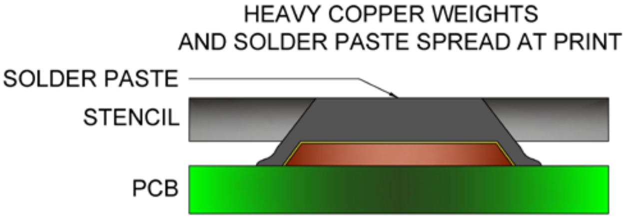

Figure 5: Solder Paste Spread

Solution

The majority of solder paste stencil design rules do not consider the weight of the copper, nor the PCB surface finish, when determining the width reduction for stencil apertures. The stencil aperture width is reduced by a standard amount, regardless of the PCB configuration, and is typically tied to component pitch, component type, and/or the SMT land pad size in the electronic PCB files.

Heavier copper weights and non-flat PCB surface finishes require a more aggressive width reduction to stencil apertures. This minimizes the opportunity for solder paste squeezing out between the PCB and stencil. Recommended stencil aperture adjustments, based on PCB copper weight and surface finish, are shown in Table 2.

Copper Weight (oz)	Copper Thickness (µm)	Stencil Width Reduction (ENIG, OSP, Ag, Sn), inches	Stencil Width Reduction (HASL & HAL), inches
0.38	13	0.001	0.002
0.50	17.5	0.001	0.002
1.00	35	0.002	0.003
2.00	70	0.003	0.004

Table 2: Stencil Aperture Width Reductions

Insufficient Solder Volume at SMT Reflow

Root Cause

Insufficient solder volume is a very common SMT defect that is typically caught at the end of the SMT assembly process during automated optical inspection (AOI) or visual inspection. In some cases, the potential for insufficient solder volume is caught before production during a design for manufacturing (DFM) review. However, a DFM analysis is not always available, nor accessible. When insufficient volume does occur, it is most commonly associated with leadless packages (QFN, DFN, RNET, etc.).

Sufficient solder paste volume with leadless packages is provided when the PCB land pad length is ~110% of the leadless termination length (Figure 6). However, inspection and rework of leadless components is extremely difficult when almost 100% of the land pad is underneath the component. For this reason, the majority of leadless land pad designs will lengthen the PCB land pad well beyond the component termination.

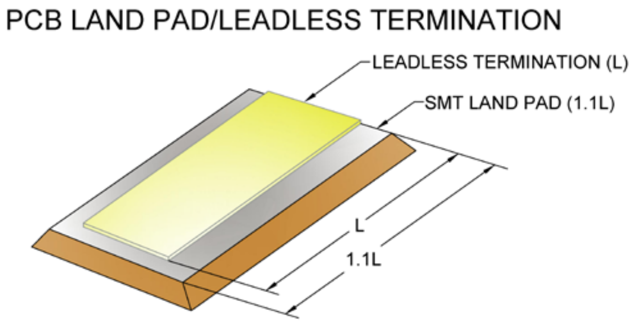

Figure 6: Leadless Package Land Pad Size

While in the reflow oven, the leadless package (Figure 7) design tends to obstruct more of the convection air flow and IR, compared to a gull wing style component. Unless the PCB has extremely heavy copper weights, the leadless termination and PCB land pad temperatures will increase fairly uniformly, and be close together at liquidus. This will produce a uniform wetting of the solder across the surfaces.

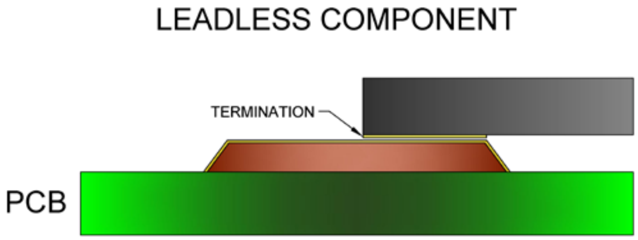

Figure 7: Leadless Component

Increasing the land pad length beyond 110% of the termination length increases the surface area that the solder has to cover. While this helps with inspection and rework, it can also force rework unless solder paste volume is increased at stencil print.

Solution

The required volume increase is based on the size difference of the leadless termination and PCB land pad and is applied to the stencil. Referring to Figure 8, the volume increase is calculated as follows:

$$\text{Volume increase (\%)} = 50 * \left(1 - \left(\frac{L}{P}\right)\right)$$

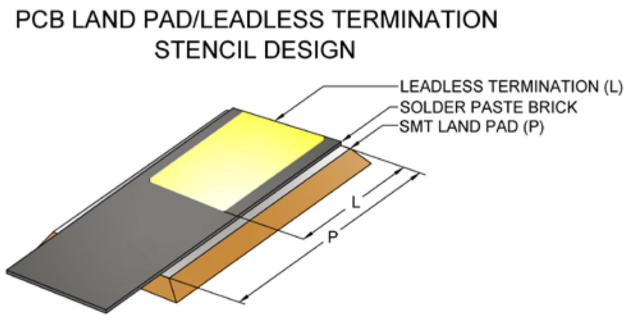

Figure 8: Leadless Package Stencil Design

The additional solder paste volume should always be printed to the "toe" side for leadless components (extending the solder paste brick further underneath the leadless package should be avoided, due to the bridging potential) and increasing the stencil aperture width should also be avoided. Extending solder paste beyond the SMT land pad, up to 0.040", is not a problem with leaded or lead-free solders (both coalesce and pull back onto the SMT land pad extremely well). However, it is extremely rare that an overprint gets even close to 0.040" with leadless components (an overprint to this extent is typically reserved for paste-in-hole applications). The majority of the time the extension is somewhere between 0.005" and 0.010".

In addition to increasing the stencil aperture size, the stencil foil thickness is also very important. For the majority of leadless components, a 0.005" foil thickness is required. If the foil thickness has to be reduced to accommodate other SMT components, the stencil aperture volume for the leadless components should be increased accordingly. Solder volumes are critical and it does not take much of a volume reduction to start causing yield problems.

Bridging at SMT Reflow

Root Cause

Bridging at solder paste print was presented earlier in this paper. Its root cause was solder paste squeezing out between the PCB and stencil at print, depositing too much additional solder paste volume. In some cases, the bridging was obvious after print. In others, bridging was not observed at print, but only after SMT reflow. Regardless, the SMT defect was generated at the solder paste print process.

There are times when bridging at SMT reflow print is not linked to any problems at solder paste print. Sometimes the root cause can be traced to PCB fabrication problems, placement pressure at component placement, reflow oven settings, etc. In many of these cases, PCB layout and/or component substitutions are the root cause of the SMT defect and bridging would occur regardless of PCB quality, solder paste print quality, component placement, or SMT reflow oven settings. When everything in the SMT assembly process is optimized and bridging does occur at SMT reflow, it is most commonly associated with gull wing/leaded packages (QFP, SOIC, etc.)

Whereas leadless packages tend to obstruct the convection and IR heating in the SMT reflow oven, gull wing/leaded packages have the component leads (Figure 9) completely exposed to the convection and IR heating. Leadless packages will have fairly uniform heating, and more uniform temperatures, between the PCB pads and component terminations. Gull wing leads, because of their exposure, are much more susceptible to temperature differences between the gull wing leads and PCB pads.

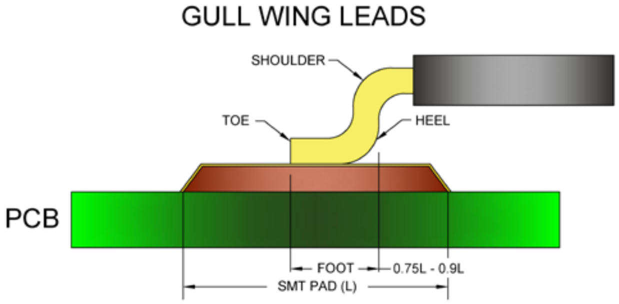

Figure 9: Gull Wing Leads

When the gull wing foot is 75% - 90% the length of the SMT pad (L), temperatures between the two will be closer together, at liquidus, and there is adequate surface area on the gull wing lead to wet the available solder. When the gull wing foot drops below 75% the length of the SMT pad (L), both the thermal mass and surface area of the gull wing lead are reduced. A lower thermal mass allows the gull wing lead to heat faster and creates a larger temperature discrepancy between the gull wing lead and PCB pad. The larger the temperature difference at liquidus, the less symmetric the wetting of the solder. Liquidus solder is more attracted to higher temperatures. When there is a larger temperature difference, liquidus solder will be more attracted to the higher temperature. The solder will wet the surface of the PCB pad, but more of the volume of the liquidus solder will start to pool at the "toe" and "heel" due to the higher temperatures of the gull wing lead. The gull wing lead has a limited amount of surface area to wet the solder. If too much solder pools at the "toe" and "heel," the excess will spill off of the PCB pad. In the case of fine pitch components, adjacent pads can bridge.

This scenario can be created when a PCB layout is far from optimal, but it can also be created due to component substitution. Alternate components provide the same function, but all mechanical dimensions, primarily lead dimensions, of the alternate are not guaranteed to match the original. Minimal differences in gull wing foot lengths are drop in, but more substantial differences can easily create solder bridging problems at SMT reflow. The solution is to reduce the solder paste volume.

Solution

The required volume decrease is based on the size difference of the gull wing lead and PCB land pad and is applied to the stencil. Referring to Figure 10, the volume decrease is calculated as follows:

$$\text{Volume decrease (\%)} = 57 * \left(1 - \left(\frac{L}{P}\right)\right)$$

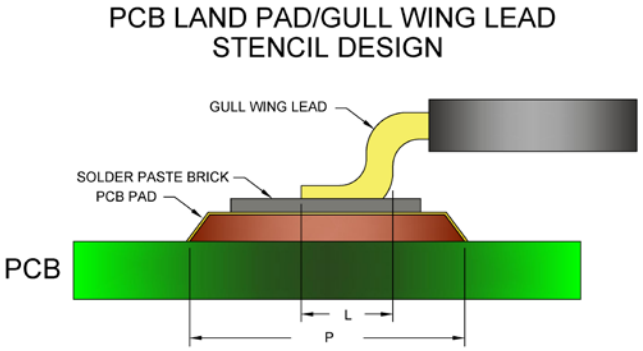

Figure 10: Gull Wing Component Stencil Design

The reduction in solder paste volume should always be centered on the gull wing foot and not the PCB pad. If centering the new stencil aperture on the gull wing foot forces solder paste off of the PCB pad, the stencil aperture should be shifted back onto the PCB pad. This is not due to the danger of solder balls or bridging, but is due to maintaining the highest surface area ratio possible. Sometimes the gull wing foot is extremely short and volume reductions can be as high as 50%. A reduction of this magnitude can reduce the surface area ratio substantially.

For the majority of assemblies, these volume reductions will dramatically reduce, and frequently eliminate, bridging on gull wing components. However, care must be taken when the PCB surface finish is OSP and the solder is lead-free. Lead-free solder does not wet as well as leaded solder and substantial volume reductions can leave some of the OSP exposed after reflow. Exposed OSP can create long-term, reliability issues. In these cases, this recommended solution may not apply.

CONCLUSIONS

Some SMT defects are isolated to a specific assembly, on a specific assembly line, or at a specific location. However, some SMT defects, like poor solder paste release, bridging at print, insufficient solder volume at SMT reflow, and bridging at SMT reflow occur everywhere and are not isolated to a unique set of variables.

The common root causes for these “universal” SMT defects, and their corresponding solutions, have been validated over a 4 year period using 1,000 different assemblies, at hundreds of customer locations located throughout the world. Clearly shown is that PCB copper weights and surface finishes are very important to SMT process yields and different PCB configurations, and their effects, must be considered.

FUTURE WORK

Validation of the “universal” SMT defects presented in this paper continues. Validation of other “universal” SMT defects, i.e. tombstones, solder balls, and voids, also continues and the results will be presented at a later date.

REFERENCES

[1] R. Dervaes, FCT Assembly, “Successful Stencil Printing: Performance is on the Surface”, 2013.
[2] R. Dervaes, FCT Assembly, “Improving SMT Assembly Yields: Eliminating Insufficient Solder Volume with Leadless SMDs”, 2013
[3] R. Dervaes, FCT Assembly, “Solder Bridging, PCB Copper Weight and PCB Surface Finish: A Cause and Effect Relationship”, 2013