

Component dimensions can be a guide for the right stencil design!

By: [Design for Success](#)

SMT defects can sometimes be so complicated that none of the changes made to the production line fix them, and may end up having your re-work station fully booked fixing every single board! But the solution sometimes is not within the production line settings, another item may fix the defect for good; which is the well-designed stencil! And by well-designed we mean to perform an analysis for the defect in order to determine the right aperture size, and not to just apply a blind design!

One of the defects we worked on was bridging at a SOIC, customer tried to change the printer settings and reflow profile but both changes didn't help, then they had their stencil house re-cut a new stencil with extra width and length reduction, but they had bridging again! That's what we call it a blind design!

When we analyzed the defect, we found the problem was due to size mismatch between the SMT land pads and component dimensions, as shown in Figure1, which has two issues; the size of the SMT pad versus lead feet, and the center point of the pads and feet, they had the SMT pads shifted outward!



Figure1: PCB Pad (Magenta), Component (Blue)

The first stencil caused so much paste released at the toe end of the lead which caused bridging at reflow, and the second stencil did the same!! Although extra reduction was applied but it didn't address the cause of the defect, and hence it didn't help! Figure2 shows the design of the first and second stencils compared to the component feet; it's clear that both stencils release too much paste at the toe ends!



Figure2: PCB Pad (Magenta), Green (First Stencil)
Red (Second Stencil), Component (Blue)

Such defect can almost be never fixed through stencil design without checking the component dimensions and comparing that to the SMT pad design in order to determine the root cause of the defect and apply the right reduction to the stencil aperture! For this defect, the component dimensions were the guide to design the stencil as shown in Figure3, where the apertures were centered to the component feet and not the SMT pads to ensure uniform wetting at both toe and heel edges of the leads (This was for a tin lead solder paste, different consideration be taken for lead free as it doesn't wet the SMT pads as good!)



Figure3: PCB Pad (Magenta), Yellow (New Stencil)
Component (Blue)

Stencils can truly help fixing almost all the defects happen at production line, but only when designed right after a thorough analysis, and one of the aspects to look at is the component dimensions!