Solder Paste is Dumb

Could vendor collaboration get to the root cause of an intermittent soldering problem?

Published in CIRCUITS ASSEMBLY

Simply put, the ultimate function of a PCB assembly line is to create millions of solder joints without error. This task is complicated by the myriad materials that come together during assembly, and relies on the quality of each lead, pad and sphere to be soldered. When a soldering defect is discovered, it is common practice to presume the soldering materials are the cause, which seems logical, considering it is a solder defect. This assumption is often misplaced. This scenario plays out regularly, as illustrated in a recent case submitted to our failure analysis team for diagnosis.

In this case, the assembler had an intermittent solderability issue with a component. It brought the problem to its local representative's attention on several occasions. The issue was isolated to a single component and was repairable at the rework station. It was a nuisance but didn't interfere with production schedules. The situation was difficult to address; it was intermittent and subtle, but persistent. And each time the representative brought its solder supplier's field engineer for site visits, the solderability issue was not present. Solder companies detest these types of issues because they have a negative effect on customer satisfaction and product perception.

Often a production quality issue appears and disappears with no obvious cause. Depending on the severity of the defect, it can grind production to a halt or simply frustrate the engineering staff and create extra work, costing time and money. (The importance of a responsive supplier cannot be overstated. If your vendors do not promptly return emails or phone calls, promptly find one who does!)

There is a telltale clue in the description of the issue the customer was experiencing: All the components soldered perfectly, except for the one with the issue. Solder paste is dumb. It doesn't solder one component perfectly and not solder another on the same assembly in the same profile. It's not lying in the printer scheming to destroy your first-pass yield and defects per million opportunities (DPMO) numbers. Something else must be happening to cause this component to have poor solder results.

Diagnosis can be frustrating and tests inconclusive, which can lead to inaccurate assumptions. When this happens, check all the basics before going deeper. The checklist should include screen printer setup (level, rails parallel, correct tooling, clean, correct/damaged/plugged stencil, to name a few). Pick-and-place issues like dirty nozzles, contamination and placement settings should also be considered. The reflow oven gets a special mention. Assuming the printer and placement are performing as intended, the only place the outcome of an SMT line can be significantly impacted is in the oven reflow profile. With expert knowledge, residue can be relocated; solder voids can be eliminated and wetting dramatically improved. It is also where a process can be profoundly and adversely impacted. A malfunctioning heating element or a failing blower motor can cause subtle process inconsistencies. But in this case, everything checked out. It wasn't the process. Something was going on with the materials.

The PCB was an ENIG surface finish. Since there were no solderability issues with other components, attention turned to the single offending component. The issue was not traceable to internal handling; it would occur on components in properly stored and sealed packaging. The production environment was adequately controlled. The facility is in a southern climate, and temperature and humidity trend to the high side of normal during half the year, but the issue wasn't correlated to the conditions. The reflow profile and oven were optimized by an SMTA-certified engineer using the latest in profile data collection. Due to the thermal mass of the assembly, a fairly long, hot profile was necessary, but it was well within the limits of the materials in use. The component vendor's wetting balance tests indicated no defect present, and the issue was seen across multiple component lots.

OUT OF BALANCE. When testing SMT component solderability, wetting balance test results can be misleading. Wetting balance testers use a molten solder bath that is not representative of a three to four minute reflow profile. In other words, rapid application of high-temperature solder can break down oxides present much more effectively than solder paste in a long, hot profile. In fact, reflow profiles can exacerbate oxide formation before soldering occurs, worsening the condition. Wetting balance testers will reveal gross wetting defects, but may mask more subtle issues.

The indicators pointed to a component solderability issue, but all the evidence was circumstantial. A definitive test was needed to rule the component in or out of specification. Customer-supplied components were baked at 125°C for five minutes to simulate exposure to reflow profile temperatures, and again subjected to wetting balance testing. It was immediately evident the component was not soldering correctly after exposure to the elevated temperatures. The test was repeated on several lots of components, and the issue revealed itself in the same way it was presenting at the customer site, transient and unpredictable with no observable pattern. With this evidence, the assembler was able to provide the component vendor with irrefutable evidence the component was the root cause and a remedy needed.

It is not uncommon for multiple vendors to have conflicting advice for a shared client. It can seem they are responding to protect their interests. In this case, however, all the vendors had provided sound information to the best of their ability, with no clear solution. It took commitment, patience and creativity to resolve a smoldering issue. Solder paste may be dumb, but choosing vendors who can support their products and customer processes is very, very smart.

Published October 2018