



On the
Forefront

Dr. H.J. Zapfardt

The Doctor is In!

Why your area array could use a good colonoscopy.

Ed.: Filling in for the vacationing Phil Zarrow is Dr. H.J. Zapfardt, well-known (and antagonistic) assembly process guru. The good doctor has selected some letters from his mailbox.

We conscientiously profile every assembly we reflow, but have trouble keeping thermocouples attached. We solder with Sn63/Pb37, and use a high-temp alloy, Sn95/Pb5, for attaching the TC. Yet on every pass at least one TC comes loose.

I'll give you credit for not being one of those bozos who attempt to attach the thermocouples with Kapton tape. Those misguided souls think they are getting accurate readings of solder joint temperature. If process accuracy doesn't count for much, they are on the right track. And profiling takes on the role of busy work for an attempt at job security. (Did you know a KIC could be reprogrammed to play Frogger and a M.O.L.E. to emulate Donkey Kong?) You are omitting an important step. Since you use a populated and previously soldered PCBA, it contains Sn63/Pb37 or 2% Ag. When a TC is attached with high-temp solder, a mystery alloy is created. The existing Sn63/Pb37 is undermining the high-temp characteristics of the attaching solder. So, Sparky, use solderwick (or another solder-removal method) and remove as much of that eutectic stuff as possible. Then, using some liquid flux and a conservative amount of high-temp solder, attach that TC. Or use conductive epoxy, but get that old solder off. Let's face it, profiling is boring as hell (clever thermal analogy, huh?) and the last thing you need is a thermocouple popping off. I have profiled hundreds of boards using this method and have never lost a TC. I say "have profiled" because I don't waste my time doing that anymore; I get some flunky to do it, which leaves me more time to program my KIC to play Mario Brothers.

We are a contract assembler and have recently taken on a project with BGAs. Do we need an x-ray machine?

Welcome to the 21st century! Rest assured, those newfangled BGAs (and other area array packages) are no passing fad (like some said surface-mount would be). In fact, a number of ICs come packaged only as BGAs, so those of you with a QFP gullwing fetish are out of luck. They are here to stay; if your company plans to stick around, be conscientious about how you process the little buggers. That includes inspection. Does Superman work on your assembly line? I didn't think so. Looks like

purchasing x-ray equipment is in your future. You can find great bargains on used machines (but stay away from the one that orthopedist is hawking on EBay). A year or so ago, the latest generation of systems took a major step in capabilities and pricing. A heck of a lot of flexibility can be had for \$125,000 to \$140,000. But since no manufacturer has made the effort to buy me off, that's all I'm going to tell you. As good as it is, x-ray is not a complete "view." Check out devices that look under the component. One method is with an endoscope where a prism (attached to a high power microscope) is used opposite a light source. Adjusting the depth of field allows a very intricate view. Boroscope-type devices work sort of like a colonoscopy for your area array. While not in-process methodologies, they are useful in process assurance and failure analysis.

We wipe the underside of the stencil every eight prints. Is this adequate?

And I make it a point to change the oil in my car at least once a year, whether it needs it or not. Quite a few factors determine the frequency of stencil wiping. If you are getting skips (due to apertures blocked with semi-dry solder paste), the answer is an emphatic "no." Parameters include ambient relative humidity, temperature, the size of the smallest aperture and solder paste formulation. Your attempts to ration wipe cloths and solvent are going to cost you. Most practitioners with apertures for 0.020" pitch (or less) components wipe after every print. Considering that the printer will always outpace the pick-and-place machine, this is good, safe practice.

We have a designer who insists no-clean pastes and fluxes cannot be used in RF applications. Is this true?

Is his CAD station coal-fired or what? In the early days of no-clean there was some truth to this, but today it's ancient history. Keep him in design and off the manufacturing floor. Better yet, lock him in a closet, along with some back issues of the *Saturday Evening Post*. No-clean applications passed the 1 GHz barrier some time ago. I even encountered an application pushing 5 GHz using no-clean. So RF, by itself, is not necessarily a limiting factor. Is the product used in an extreme environment, particularly in temperature and humidity? Before converting any product wholesale to the no-clean process (and it is a process, not just a material), always undertake appropriate environmental stress, life and other reliability-related testing. ■

Dr. H.J. Zapfardt is a member of the National Academy of Arrogant Engineering and an inductee of GIT's International Hall of Fame. He is a sought-after advisor to industry and government, or basically anyone who will put up with him. A popular speaker on the SMT "rubber chicken circuit," he was recently cited as "a personality to avoid." He has written more than 300 articles and textbooks, including the acclaimed A Children's Garden of Flux. Dr. Zapfardt holds a doctorate in metaphysics from the University of Bonghit. He can be reached at groucho lives@aol.com.

