

## “Pause” for Better Solder Paste Printing

*By Harry Trip  
COBAR Europe*

Quite often, NiAu plated PCB surfaces create serious printing problems on SMT fine-pitch assemblies. The paste tends to remain in the apertures and only traces of flux are visible on some pads. Why? One cause is the solder resist. Sometimes, a thick resist is used and the pad is just too far away from the aperture to be reached by the paste. Also, the NiAu surface structure provides less adhesion to paste than other finishes. It all depends on the PCB manufacturer's gold plating process.

To avoid the non-printed pad phenomenon, remember to stay within the design rules for aperture width. Too wide a pad is better than too narrow. Modern synthetic no-clean pastes provide high elasticity and therefore have a great anti-bridging tendency. This high elasticity doesn't help us, however, with fine-pitch NiAu printing. If the paste doesn't fully contact the pad, it tends to pull back into the aperture. Most modern printers provide a simple solution to the problem that many are not aware of: A programmable pause after each print stroke, before separation begins.

The length of this pause depends on paste type and can vary between only half a second to several seconds. This will give the paste extra time to sink down onto the paste surface so proper aperture release can take place. Obviously, this will slow-down the printing process; but luckily, most NiAu boards are used for high-tech purposes where printer output isn't the limiting factor.

### **Harry Trip**

Harry Trip is Product Applications Manager at Cobar Europe BV in Breda, Holland (The Netherlands). Contact him at [h.trip@cobar.com](mailto:h.trip@cobar.com); **Cobar Europe BV**, Aluminiumstraat 2, 4823 AL Breda, Holland, (PO Box 3251, 4800 DG Breda, Holland), Tel +31 (0)76 544 5566; Fax +31 (0)76 544 5577; Internet: [www.cobar.com](http://www.cobar.com)

