

October 8, 2007

## Method for cleaning boards after misprint

What is the correct process for cleaning bare boards after solder paste mis-print?

- We are using no-clean solder paste both leaded and lead-free.
- Our operators are unable to clean the bare boards thoroughly enough to avoid extra solder or solder balls on the completed assemblies.
- We are using IPA to clean the bare boards.
- We have a combination of FR-4 HASL and Immersion Silver lead-free boards.

E. R.

This "Ask the Experts" page has been viewed 366 times.

### Ask the Experts Comments



October 8, 2007

Hand wiping is not effective for removing uncured solder paste due to solder ball smearing over the surface of the circuit assembly. Tiny solder balls may end up in via's or other small spaces and lead to shorts.

Solder paste and particulate removal is often more difficult than dissolving uncured flux residue. The most reliable removal method for is a process that integrates mechanical and chemical driving forces. Solder balls are held in place by the flux composition. Developing an integrated cleaning process, releases the solder balls and allows removal during the wash and rinse cycles.

For additional information, [IPC-7526 Stencil and Misprinted Board Cleaning Handbook](#) may be downloaded free of charge.

*Mike Bixenman, CTO*

**Kyzen Corp.**

[mikeb@kyzen.com](mailto:mikeb@kyzen.com)

Mike Bixenman is the CTO for Kyzen Corp. Kyzen Corp. is a leading provider of engineered cleaning fluids for high technology manufacturing environments.

This is a good opportunity to let everyone know that the IPC has recently published the Stencil and Misprinted Board Cleaning Handbook (IPC-7526). This and other related articles are available on the [Smartsonic web site](#) as a free download.

*Bill Schreiber, President*  
**Smart Sonic Stencil Cleaning Systems**  
[bill@smartsonic.com](mailto:bill@smartsonic.com)

Mr. Schreiber developed the original ultrasonic stencil cleaning process in 1989. Obtained the only EPA Verification for specific parameters of Environmental Safety, User Safety and Cleaning Efficiency for a stencil cleaning process. Introduced 440-R SMT Detergent in 1990 for cleaning all types of solder paste, flux residues and SMD adhesives.



Manual cleaning of the misprinted board through either wiping or scrapping results in entrapment of the solder balls in the vias or other board apertures. To ensure proper, reproducible cleaning results and last but not least complete removal of all solder paste from the misprinted boards, we highly recommend using a automated cleaning process supported by either an ultrasonic or a spray-in-air agitation using an alternative cleaning agent to IPA.

The limited solvency of IPA on the flux residues was overcome with alternative aqueous based cleaning agents that do not only ensure the proper removal of the solder paste but also provide excellent compatibility with board finishing ranging from immersion silver to OSPs and are in full compliance with the environmental regulations.

Cleaning trials performed at independent technical centers that have real cleaning replicating capability would certainly be the most effective way to pin point which cleaning agent and equipment would be the most appropriate for your current application.

*Umut Tosun, Application Technology Manager*  
**Zestron America**

Mr. Tosun has published numerous technical articles in the leading electronic manufacturing magazines. As an active member of the SMTA and IPC organizations, Mr. Tosun has presented a variety of technical papers and studies on topics such as "Lead-

Misprinted circuit boards should have excess solder paste manually removed from the surface with a suitable cleaner recommended by the solder paste supplier and a lint free wipe. These boards should then be cleaned in an automated cleaner to further remove any residual solder paste in vias, through hole barrels or lodged along the edge of the solder mask.



It might also be a good idea to mark the edge of a misprinted boards with a Sharpe as an identifying mark. This way, boards processed in this manner are easily recognized at the end of the production line should further disposition be necessary.

*Mike Burgess, Vice President*  
**Photo Stencil**  
[mburgess@photostencil.com](mailto:mburgess@photostencil.com)

Mr. Burgess has 20 years of experience in the electronics industry as a user and supplier. At Photo Stencil, he is responsible for product management which includes, the patented AMTX electroformed stencils, laser cut Stencils and SMT assembly products. Michael previously worked for Cookson Electronics in various technical positions for 14 years.

The best method for cleaning misprinted boards is to immerse the boards in some sort of an ultrasonic dip-tank and clean them with a nonflammable solvent. Such machines are made by JNJ Corp., Smart Sonic, and others. Lockheed Martin even made one, a number of years ago, called a JetClean machine. Shop around; you can get excellent values on these machines.

Do not use alcohol as the solvent in these machines; alcohol is a weak cleaner that quickly saturates with paste and will stop cleaning quickly, so you're constantly draining and cleaning the machine. A better choice is a nonflammable, water-based solvent which has been engineered specifically for this application...

Cleaning misprints manually, which many companies do, is sub-optimal. Solder paste gets into vias, apertures and/or on the ends of the boards, and unless the techs are extremely conscientious, some residues always remain. Most companies then blow-dry the board with compressed air, which often blows microscopic bits of oil onto the board. This cleaning process also wastes a lot of labor, ties up inventory and consumes quantities of solvents and paper wipes. Overall, this is just a bad idea.

October 8, 2007

Page 4 of 4

One other thought – if you're getting a lot of misprints (more than 2-3 per hour at normal production speeds) then the problem isn't the cleaning, it's the root cause of what's causing the misprints. More often than not, it's ineffective cleaning from the stencil wiping paper. Better paper will give better wiping, which can dramatically reduce misprints and defects at functional test.

*Mike Jones, Vice President*

**Micro Care**

Mr. Jones is an electronics cleaning and stencil printing specialist. Averaging over one hundred days a year on the road, Mike visits SMT production sites and circuit board repair facilities in every corner of the globe, helping engineers and technicians work through the complex trade-offs today's demanding electronics require.