circuit insight

Sep 10, 2015

Cleaning Misprinted Circuit Boards

Does wiping off the solder paste from a misprinted circuit board before machine cleaning help to improve the cleaning process?

Ask the Experts

View the Expert comments below.

Expert Panel Contributors

- Bill Schreiber, President, Smart Sonic Corporation
- John Vivari, Application Engineering Supervisor, Nordson EFD
- Mike Scimeca, President, FCT Assembly
- Jack Crawford, Director - Certification & Assembly Technology, IPC
- Craig Hood, Director of Intl Sales and Global Marketing, Petroferm Inc.
- Karthik Vijay, Technical Manager - Europe, Indium Corp.
- Christopher Perry, Worldwide Sales & Marketing Manager, EMC Global Technologies, Inc.
- Mike Jones, Vice President, Micro Care

Expert's Panel Responses

At Smart Sonic, we do not recommend wiping or pre-treating the misprint in any way prior to cleaning in a Smart Sonic ultrasonic stencil cleaner and for a couple of reasons;

1. Wiping will push the solder balls into vias or other areas where they do not belong and
2. Wiping with alcohol or other similar "chem-wipes" can chemically react with the flux and make the resulting compound more difficult to clean.

There are several published articles available for download on our web site, click on the "Recommended Reading" button.

You may also want to read our "Frequently Asked Questions" section.

Bill Schreiber
President
Smart Sonic Corporation
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.

Not in my experience. Wiping the board forces paste into vias along with solvent and whatever else was on the board surface.

It is significantly more difficult for a washing system to clean the inside of a via than the flat surfaces of a PCB. Material that remains trapped in vias can then spatter during later reflow operations, resulting in a solder ball problem.

John Vivari
Application Engineering Supervisor
Nordson EFD
Mr. Vivari has more than 15 years of electronic engineering design and assembly experience. His expertise in fluid dispensing and solder paste technology assists others in identifying the most cost effective method for assembling products.

The benefit to pre-wiping a mis-printed board would be that it keeps the cleaning equipment from being quickly contaminated with flux and powder residue.

The downside is that depending upon the method used to wipe the mis-print from the board, you could be causing more harm by packing solder spheres into the tiny vias and other small pockets or crevices, and the cleaning equipment or process may not be capable of dislodging and removing them.

Mike Scimeca
President
FCT Assembly
Mike Scimeca created FCT Assembly after the purchase of Fine Line Stencil, Inc., and consists of two major operations: stencil manufacturing and the manufacturing of electronic assembly products such as solder paste, flux and solder bar.

Wiping the solder paste from a mis-printed circuit before machine cleaning is not advisable for two reasons:

1. Wiping mis-printed solder paste from the board surface pushes solder paste into vias. The solder paste in vias is difficult to clean and increases the risks of stray solder balls.
2. After wiping the solder paste, thin layers left on the board dry hard. Harden solder paste is more difficult to clean.

The best approach for cleaning a mis-printed solder paste is to clean the board as printed using an automated cleaning process that provides programmable wash, rinse and dry steps. The board should be cleaned within a few hours of the mis-print.

Thanks to Mike Bixenman, Kyzen Corporation, leader of the IPC-7526 Stencil Cleaning Handbook Commitee for assisting with this answer. IPC-7526 is a free-download document from IPC, click the Product Search button to search for Product ID 7526. You'll see the download link.

Jack Crawford
Director - Certification & Assembly Technology
IPC
Mr. Crawford is Director of Certification and Assembly Technology for IPC. He is technical liaison to the IPC committees that maintain critical industry standards and has presented numerous papers internationally.

Improve cleaning? Not necessarily. Is it a good idea? In general, yes.

Reducing the amount of gross contamination (raw solder paste) will help extend the life of wash bath solutions, allowing them to clean more misprinted boards and stencils.

Overall machine maintenance will be less as filters will last longer and waste treatment will be reduced. Disposing of solid waste containing heavy metals tends to be easier than treating liquid wastes, particularly if you use a solder recycle service. Hand-wiping misprints should not be done haphazardly.

Care should be taken not to smear paste into areas and crevices that make process cleaning more

difficult, entrap solder spheres or cause damage to solder surfaces.

Craig Hood
Director of Intl Sales and Global Marketing
Petroferm Inc.
Mr. Hood is Director of International Sales and Global Marketing for Petroferm's Cleaning Products Division, a manufacturer and supplier of cleaning agents for defluxing and stencil cleaning.
NOTE: Mr. Hood is no longer working at Petroferm Inc.

Not a whole lot.

A manual wipe could damage the solder mask.

As long as the cleaner parameters are optimized, there should not be a problem in removing the misprinted paste

Karthik Vijay
Technical Manager - Europe
Indium Corp.
Currently with Indium Corporation and responsible for technology programs and technical support for customers in Europe. Over 15 yrs experience in SMT, Power, Thermal & Semiconductor Applications. Masters Degree in Industrial Engg, State University of New York-Binghamton.

Actually, it's just the opposite. Hand wiping misprinted boards has a negative impact on the cleaning process. This allows solder balls to become entrapped in vias and create the possibility of failures.

Here at EMC Global, our customers simply put the misprinted boards into a rack, and clean them in our stencil cleaners, with the paste on. The filtration in the machine will easily accommodate the residual paste during the cleaning process, and boards come out clean and dry.

Christopher Perry
Worldwide Sales & Marketing Manager
EMC Global Technologies, Inc.
Mr. Perry has been with EMC for 12 Years, working with cleaning and tooling applications used in electronic manufacturing environments. EMC produces all types of solvent and aqueous cleaners for stencils, misprints and fluxed boards.

In general, wiping excess paste off a mis-printed circuit board is an ESSENTIAL step in cleaning the mis-print, unless you have a really really big budget for fresh solvent.

Most companies use alcohol for cleaning stencils, which is fine in small quantities. But even a fairly small mis-printed board will have many grams of solder paste on the board, all of which needs to be removed. Alcohol will not be up to the task.

Alcohol saturates with contamination at 2% by weight. So, even in a medium-sized dip tank it will not take long for the alcohol to be saturated with solder paste and unable to clean any further. The only option is to empty out the tank and add fresh solvent.

So you have a couple of options:

1. Reduce the amount of solder paste being removed. This will dramatically extend the life of the solvent bath and improve the cleaning process. Wiping does this nicely and conveniently.
2. Switch to a strong solvent with a high flux-carrying capacity. This solvents are widely available, and work very well. But they will be 5-8 times more expensive than using cheap alcohol.

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.

Reader Comment

I think the question should be "Why are you getting misprints?"

Mark Maheux, Honeywell

Reader Comment

We have adopted the use of a Ultrasonic Process which contains a small % (~ 2 to 5%) of a saponifier with DI water. We clean the boards with the mis-printed portion of the board in the solution. We do not allow the solution to go over the top only touch the surface that has been mis-printed thus allow the metal to fall to the bottom of the tank.

We run the ultrasonic for 4 minutes at 35degrees, then Water Wash using a batch Centrifuge system for 15 minutes then 15 minutes in a Vacuum bake. When all said and done the boards is good to go back in to the process.

Ray Whittier, Sr.SMT Process Engineer, Vicor Corporation - VI Chip Division

---

Reader Comment

Please dont wipe down boards prior to misprint, the powder will simply be forced into holes and wells on the circuit, thus having to wash the PCB much longer, and subjecting the solder mask to longer immersion times in the stencil cleaner. If the mask is undercured this will be detrimental to the resist.

Greg York, BLT Circuit Services Ltd
