Tech Article

Why Clean a No Clean Flux

Author:
Emily Peck, MicroCare Senior Chemist

Industry:
Electronics

Published:
Global SMT & Packaging Magazine

Many small-component PCBs (printed circuit boards) are assembled today with no-clean solder pastes. No-clean solders speed up the manufacturing process by eliminating the need for cleaning circuit boards after reflow. The solder paste did not need removal and stayed in place without any detrimental effect to the function, performance and planned life expectancy of the circuit board.

As a result, millions of electronic assemblies are now produced using no-clean solder pastes. Experts estimate that over half of the printed circuit boards made today use a no-clean solder paste for production. In some instances, manufactures equip their entire assembly lines with no-clean pastes and then selectively clean only the boards they choose. For instance, they may opt to clean only high-reliability boards, the ones used for security, military, aerospace and medical devices, where they know cleanliness is critical. The other, non-critical boards may skip the cleaning process altogether, saving manufacturers both time and labor.

Increasing PCB Complexity

However, as printed circuit boards become more complex and more tightly packed, some PCB makers realize that no-clean soldering doesn't always work well with today's modern electronic circuitry. The growing demand for smaller electronics is forcing manufacturers to squeeze micro components like flip chip, micro BGA, CSP and QFN packages into tighter spaces on the boards. Low standoff components like MOSFETs are commonplace. I/O (input/output) counts are increasing and circuit boards are becoming multi-functional.

This trend in circuit board miniaturization, complexity and high density may cause a greater likelihood for problems if the PCBs are not properly cleaned. This applies especially to those produced using no-clean solder pastes. Dirty PCBs can be vulnerable to many types of problems including parasitic leakage, electrochemical migration, shorting and dendrite growth. Today, the need to clean no-clean flux residue is no longer optional, it is becoming essential for long-term PCB performance, functionality and reliability.

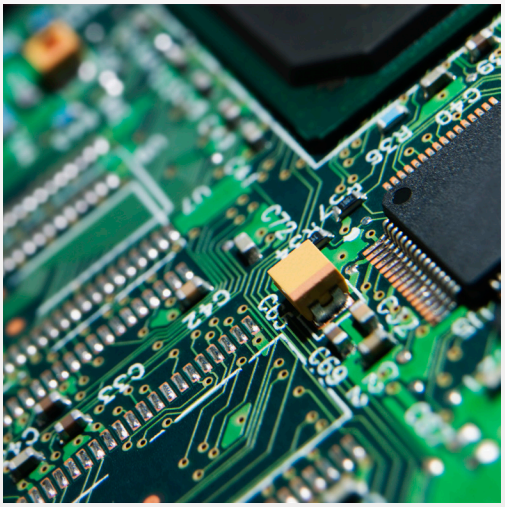
The irony of no-clean fluxes is that they were supposed to ease the need for PCB cleaning. Instead, they may now present one of the most difficult PCB cleaning challenges. When the salt activators in the no-clean flux come in contact with heat or other chemicals, they leave behind a white residue that may corrode fragile circuits and enable dendrite growth. Unlike other fluxes, no-clean flux was intended to stay on the board. Therefore, it is now some of the most stubborn and difficult contamination to clean from printed circuit boards.

Improved Board Performance

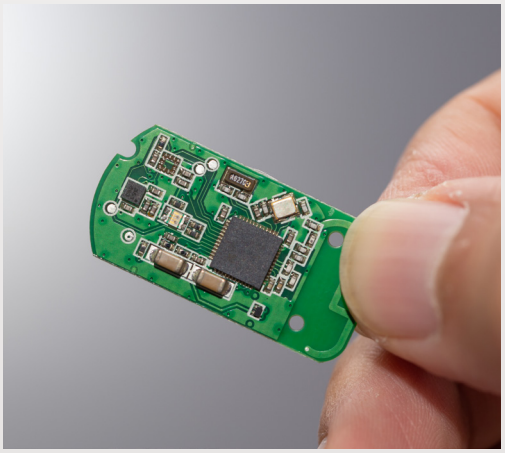
As modern PCBS continue to become more complex, many assemblers today are choosing to clean all of their PCBs, not just the high-reliability boards. Preventing unpredictable performance, costly board failures, product recalls and product returns is a big motivator. So, despite the advent of no-clean soldering, many manufacturers are still cleaning.

The primary reason to remove no-clean flux and its residue is to prevent malfunctions in circuits and to prevent interference with signal transmission. For instance, if too much no-clean flux builds up on a circuit board, or if white residue is left behind, it creates noise on the board. This is especially true on high-voltage systems.

MicroCare™



Today's modern PCBs are complex and tightly packed with components.



PCBs are becoming smaller.

Solder balls are also trapped between tightly-spaced components. Other contaminants like ink and fingerprints may also need to be cleaned away for optimal circuit board performance.

Assemblers may also want to clean no-clean fluxes for other reasons. Improved cosmetic appearance of the boards, easier inspection and quality control or better conformal coating performance are among the most popular.

Improved Aesthetics

Flux residue can detract from the cosmetic appearance of printed circuit boards. This is a key concern for consumer electronics. For some customers, flux residue represents careless or sub-par work. Cleaning the flux residue from circuit boards may help make them more visually appealing and more acceptable for customers.

Easier Inspection and Quality Control

No-clean flux and leftover residue may make inspections difficult. QC (quality control) inspections and troubleshooting field repairs are more challenging if flux is left on the circuit boards. It can also cause automated visual alignment systems to fail. By thoroughly cleaning no-clean fluxes it may help make PCB inspections easier and more accurate.

Better Conformal Coating Performance

Some PCB manufacturers are finding that no-clean residue may sometimes prevent proper adhesion of conformal coatings. This is a real problem for circuit boards in outdoor or other harsh environments. Flux residue can absorb moisture. This trapped moisture may then be released during curing operations, causing the coating to separate from the board. This can allow corrosive materials, dust, or water to penetrate the PCB assembly causing corrosion, signal transmission problems, or even complete component failure. Comprehensive removal of any no-clean flux and its residue prior to conformal coating may help prevent this from happening.

When cleaning no-clean fluxes there are some best practices. If followed, they help PCB makers get quality cleaning results. This translates to the better PCB performance, reliability and longevity they expect.

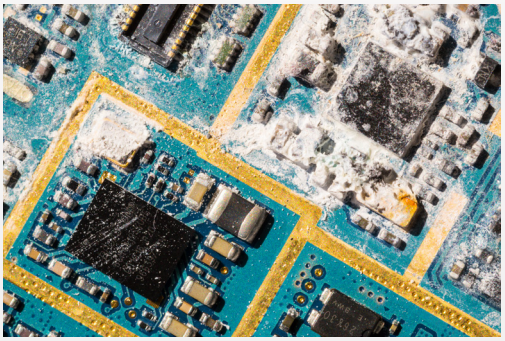
Early Specification

Traditionally, many PCB designers and engineers didn't think about PCB cleaning until the very end of the design process. But as PCBs become more complex, they recognize the need to determine their cleaning options earlier in the design process. They need to determine their cleaning needs before making final decisions about coatings, solder pastes and other materials. It can be much easier to resolve any cleaning problems with early fluid specification prior to production. This is especially true when manufacturing high-reliability PCBs. For instance, medical or military products where cleanliness is critical. Or in applications that require validation processor other stringent requirements.

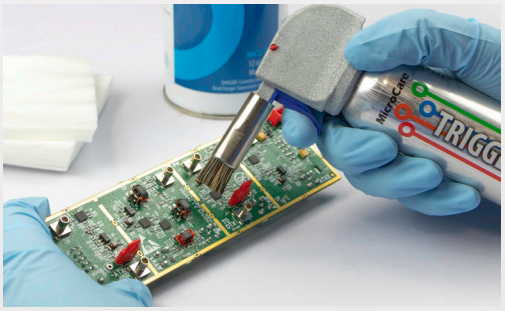
Testing

In addition to early fluid specification, testing happens earlier too. Many manufacturers are starting to test-clean in small batches prior to full-scale manufacturing. They are finding it beneficial to produce and clean a small batch first, ensuring cleanliness before going into full production mode. Once the cleaning

MicroCare™



Solder paste and flux needs to be removed for optimal performance.



Dispensing tools scrub contaminants and help control fluid flow.

fluid and process works on a smaller scale, then they scale-up to the higher volume they need. Through this small-batch preemptive testing, they are minimizing surprises and ensuring better board reliability before it becomes a major headache.

Follow the Process

For best results in both soldering and cleaning, follow the solder paste manufacturer's recommended reflow process. Exposing no-clean solder pastes to extreme heat may cause the flux residue to darken and solidify, making the baked-on flux harder to remove. The oven needs to be hot enough to cure but not so hot that it burns the flux to the point it cannot be cleaned.

Check Materials Compatibility

No-clean flux removers are available in a variety of strengths. Assemblers should choose a cleaning fluid by carefully matching the fluid to the contaminant. In addition, the cleaning fluid should be compatible with the PCB substrate and other materials on the board. The cleaner should be strong enough to effectively remove the flux residue, yet not cause damage to metal, plastics or coatings.

Choose Tools for Safety

When possible, PCB assemblers should choose a nonflammable no-clean flux remover for workplace safety. Also, using a cleaning fluid dispensing tool can easily scrub away contaminants and help boost worker safety by regulating the amount of fluid and fumes they are exposed to.

Keep Options Open

If a PCB producer is still having trouble with a no-clean residue, it might be time to try a different cleaning fluid or even a different no-clean solder paste. They could try a paste with less solids or resins in the flux. The lower the solids content, the lower the flux residue left on the board therefore making it easier to clean. Also, a no-clean flux with fewer halides may be easier to remove. If all else fails, they may consider switching to a different solder paste that is more cleanable. MicroCare can provide cleaning fluids for a number of flux residues including everything from water-based and lead-free to R, RA, RMA, OA and SA fluxes.

Ask for Help

If you are looking for help in determining the correct cleaning fluid or method to use, MicroCare can help. We specialize in developing cleaners to remove no-clean flux residue. MicroCare has field engineers that run on-site audits to evaluate cleaning methods. We have also conducted comprehensive, in-lab cleaning tests with some of the industry's most popular solder pastes and fluxes to ensure their cleaning success. You can rely on MicroCare to recommend the cleaning fluids and methods that will work best.

About the Author:
Emily Peck is a Senior Chemist at MicroCare which offers benchtop and vapor degreasing critical cleaning solutions. She has been in the industry more than 6 years and holds a MS in Chemistry from Tufts University. Peck researches, develops and tests cleaning-related products that are used on a daily basis in electronics, medical, fiber optic and precision cleaning applications. For more information, visit www.microcare.com.

MicroCare™

Follow Us!