What’s You PCB IQ?

Marissa Oskarsen & Chrys Shea, Printed Circuit Girls and Geeks

Are You a Clean Freak?

Take The Printed Circuit Girls and Geeks' 10-Question Pop Quiz on PCB Cleaning

QUESTIONS

Part 1: True or False?

1. The percentage of circuit assemblies that are cleaned after soldering is on the rise.

2. Sometimes "no-clean" fluxes must be cleaned.

3. No-clean flux residues are more difficult to clean than organic acid flux residues.

4. Partially removing a no-clean flux residue is more dangerous than not removing it at all.

5. Lead-free flux residues are tougher to clean than tin-lead residues.

Part 2: Multiple Choice

6. The most important consideration in the circuit assembly cleaning process is:
   A. Flux residue
   B. Under-component clearance (gap height)
   C. Prior heat exposure
   D. Cleaning agent
   E. Cleaning machine

7. The halides used to activate some fluxes are often associated with electrochemical reliability issues, and chlorides are often identified as the deadliest of all the flux-borne PCB predators. What is the maximum allowable concentration of chloride ions on a PCB that is considered safe?
   A. 2ug/in^2
   B. 5ug/in^2
   C. 10 ug/in^2
   D. 20 ug/in^2 Cl
   E. None of the above
   F. All of the above

8. The driving force behind the post soldering cleaning renaissance is:

A. Higher reliability requirements
B. Increasingly harsher service environments
C. Miniaturization
D. Higher signal speeds
E. None of the above
F. All of the above

9. To assure adequate cleaning under low standoff, bottom termination components like QFNs and MLFs:
A. Gang-relieve the solder mask around the component
B. Specify a solvent-based presoak before the automated cleaning process
C. Don't spec them at all!!! They are designed for assembly with no-clean processes only.
D. None of the above
E. All of the above

10. "DFC" or Design For Cleaning incorporates which of the following considerations:
A. Conductor spacing
B. Minimizing thermal mass differentials across the PCB (Copper balance and component population)
C. Component placement, footprint and orientation to provide flow channels for the cleaning agent
D. None of the above
E. All of the above

ANSWERS

1. The percentage of circuit assemblies that are cleaned after soldering is on the rise.
True. Twenty years ago our industry adopted "no-clean" soldering technology, and post-reflow cleaning was predicted to become extinct, just like the dinosaurs. But like the dinos in the movie Jurassic Park, PCB cleaning has not only returned; it has proliferated.

While there are a number of reasons that cleaning is on the rise, the biggest factor is miniaturization. Designs with finer pitch components that have lower standoffs and tighter spacing are more sensitive to flux residues than roomier designs. (Now if you want to go back and change your answer to question number 8, go ahead. We won't tell anyone. Only you will know that you're a cheater. And if you're good with that, who are we to judge?)

2. Sometimes "no-clean" fluxes must be cleaned.
True. We know it sounds completely backwards, but it is true! A lot of OEMs like residue-free assemblies, but most CEMs like no-clean solder pastes because they're *way* more process-friendly. So assemblers often opt to build the PCB with no-clean solder paste to get higher yields and then they clean the no-clean residues. It doesn't

seem to make sense, but it makes money, and dollars are a lot more important than sense.

3. No-clean flux residues are more difficult to clean than organic acid flux residues.

True. As their name implies, no-clean fluxes are designed to stay on the PCB. They're full of rosin, which encapsulates the acids so they can no longer work on the metals. Remember that chunk of amber that protected the mosquito's DNA in Jurassic Park? Amber is fossilized rosin, and the rosin in solder paste is what protects the flux's DNA, which might contain halides, acids, salts...all kinds of corrosive goodies that are best kept locked up. The rosins used today are mostly synthetic resins, and they do an excellent job of encapsulating all the nasties. They're almost too good at their job, because special chemicals are needed to dissolve them.

Organic acid fluxes, also known as water-soluble, -washable or -cleanable are designed to be removed after soldering, and MUST be removed. Most can be removed with water, but dissolve faster and easier if chemistry is added to the wash. Sometimes it's the same chemistry used on the rosin fluxes; sometimes it's not. It all depends on the fluxes' DNA.

4. Partially removing a no-clean flux residue is more dangerous than not removing it at all.

True. Partially unlocking the protective shell of rosin is a really bad idea. It will turn your circuit board into your own personal Jurassic Park, but instead of different species of prehistoric creatures, you'll have different species of metal oxides that make your PCBs *look* like prehistoric creatures.

5. Lead-free flux residues are tougher to clean than tin-lead residues.

True. To step up to the higher heat of lead-free soldering, solder pastes now use synthetic resins that are much harder to dissolve than the typical rosins of the tin-lead era. Plus, the longer, hotter thermal cycle really bakes the resins onto the PCB, adding to the challenge.

6. The most important consideration in the circuit assembly cleaning process is:

A. Flux residue
B. Under-component clearance (gap height)
C. Prior heat exposure
D. Cleaning agent
E. Cleaning machine

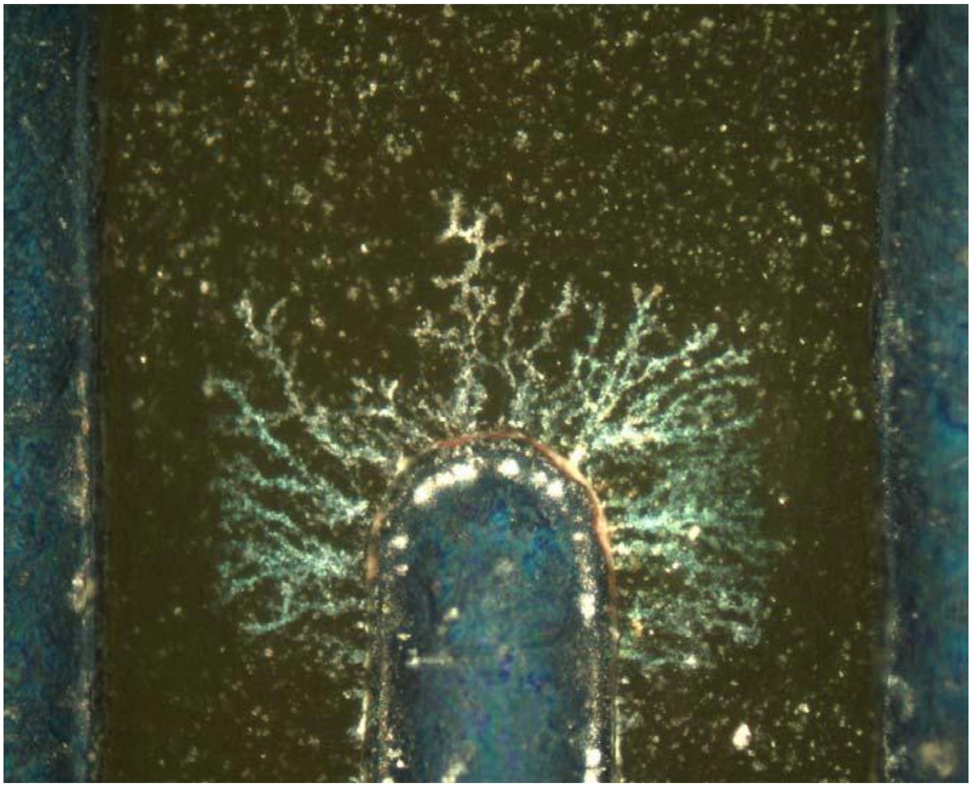
Answer: **D.** While they're all important, matching the solvent to the soil is the key. And it's a bit tricky. There's always the old trial-and-error method, which is clearly suboptimal, or there's the scientific approach that uses Hansen solubility parameters.[1] Applying the premise that like dissolves like at the intermolecular bond level, chemists study the bonds of different molecules. They look at three different types of bond energies to compare the soils with the solvents. If the energies are close, the two are paired and tested further for compatibility. If they are not, well, the elimination process has just begun…

Flux residues play a role in cleanability. Their basic chemical composition and their post-reflow conditions such as thermal exposure, degree of oxidation, and lag time before cleaning all factor into the equation. Gap height is a consideration because as the clearance under the component shrinks, so does the cleaning agent's access. Prior heat exposure dictates how baked - or even charred - the residues are. Finally, the cleaning machine type, nozzle configuration and process parameters like wash time and temperature all play a major role in the cleaning process. But without the right solvent – they key that chemically unlocks the residues – nothing else really matters.

7. The halides used to activate some fluxes are often associated with electrochemical reliability issues, and chlorides are often identified as the deadliest of all the flux-borne PCB predators. What is the maximum allowable concentration of chloride ions on a PCB that is considered safe?

A. 2ug/in^2
B. 5ug/in^2
C. 10 ug/in^2
D. 20 ug/in^2 Cl
E. None of the above
F. All of the above

Answer: **E.** None of the above, or **F**, all of the above. This is one of our trick questions: cleanliness requirements depend on conductor spacing.



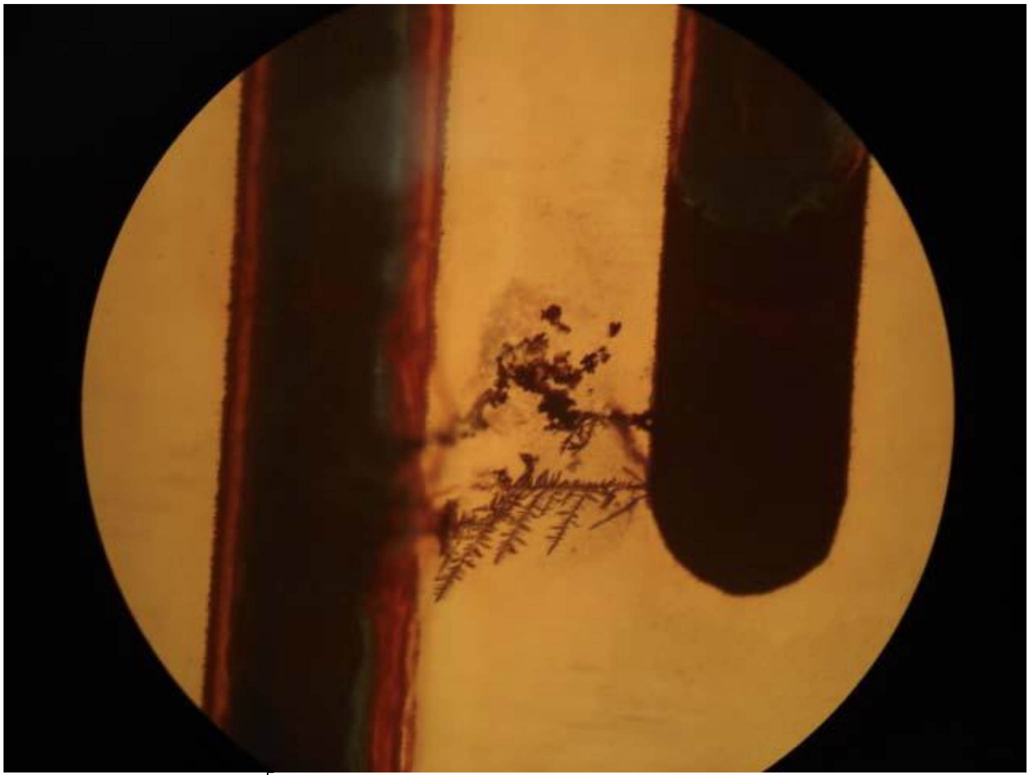
Top view of dendritic growth on test pattern treated with no-clean flux and exposed to elevated temperature and humidity under an electrical bias. Source: DfR Solutions[2]

Dendrites are the conductive metallic filaments that make PCBs look like Triceratops, shown above. They are the result of electrochemical migration (ECM), which occurs when unprotected ions, often halides, are combined with a strong enough electric field on a PCB.

ECM and corrosion tests using IPC comb patterns[3] (under a 10V DC bias for 168 hrs at 85^{o}C/85%RH) showed that on conductors spaced 6.25mils apart, dendritic growth was observed on PCBs with less than 2ug/in^2 Cl ion contamination. On conductors spaced 12.5mils apart, dendritic growth was seen on PCBs with cleanliness levels as low as 5ug/in^2. Closer conductor spacing makes for higher electric fields (at the same potential), which set the ECM process in motion.

A side view of a trace with dendritic growth and a backlit top view of two bridged conductors are shown below. The snowflake patterns they form are so pretty, aren't they? Yeah, pretty *ugly* when they're causing your PCBs to fail in the field and costing you a fortune to replace!

Source: Cullen, 2004[4]



Source: Michalkiewicz[5]

8. The driving force behind the post soldering cleaning renaissance is:
   A. Higher reliability requirements
   B. Increasingly harsher service environments
   C. Miniaturization
   D. Higher signal speeds
   E. None of the above
   F. All of the above

Answer: **C**. As a circuit's conductor spacing decreases, its sensitivity to flux residues increases. Unfortunately, as spacing decreases, the cleaning difficulty also increases. So as the needs for cleaning intensify, so do the inherent challenges. This has been dubbed “The Miniaturization Paradox[3]” by Dr. Mike Bixenman, Chief Technology Officer of Kyzen, a worldwide leader in the precision cleaning industry.

Rich Breault, President of the EMS firm Lightspeed Manufacturing, adds “High reliability PCBs often require cleaning so conformal or other protective coatings can be applied.” He also notes “Sometimes it’s not just reliability concerns that drive cleaning; on RF applications with high signal speeds, flux residues can affect signal performance and integrity.”

So there are a lot of good reasons to return to cleaning. But the major driver behind its revival is miniaturization. Remember question 1? Did you cheat? Just checking. Not judging, just checking.

9. To assure adequate cleaning under low standoff, bottom termination components like QFNs and MLFs:

A. Gang-relieve the solder mask around the component
B. Specify a solvent-based presoak before the automated cleaning process
C. Don't spec them at all!!! They are designed for assembly with no-clean processes only.
D. None of the above
E. All of the above

Answer: **A**. Relieve the solder mask all the way around the component. If you don't, the flux residue will form a goopy wall that fills the entire gap between the top of the mask and the bottom of component. The wall acts as a dam, and won't let water or cleaning fluid penetrate under the component until it gets washed away. That dam lowers the cleaning process' effectiveness and may require longer cycle times to insure that all the residues underneath the component are fully removed.

Don't even consider specifying a solvent-based presoak for a production operation. It disrupts product flow, takes up floor space, requires labor and adds cost. It's as bad an idea as shutting down the dinosaur containment grid and stealing the reptiles’ embryos. Remember what happened to the guy who cooked up that brilliant plan? He got eaten alive. Trust us; you’ll get eaten alive if you even mention the word presoak to a manufacturing person.

If only solutions were as easy as prohibiting the specification of no-clean components into water-wash processes. It happens all the time. When the silicon that the OEM wants is only available in a QFN package, it goes on the board and cleaning it pretty much becomes the assembler's problem. This is exactly why the assembler needs the solder mask relieved. So please do everyone a favor and gang relieve that mask around the QFNs if you plan on using them in a water-wash process.

10. "DFC" or Design For Cleaning incorporates which of the following considerations:

A. Conductor spacing
B. Minimizing thermal mass differentials across the PCB (Copper balance and component population)

C. Component placement, footprint and orientation to provide flow channels for the cleaning agent
D. None of the above
E. All of the above

Answer: **E**. All of the above. Considering these design characteristics up front can save a lot of time, money and headaches down the road.

- We've beaten the conductor spacing concept to death; if you don't get that by now, you're on a personal path to extinction.
- Minimizing thermal mass differentials across the PCB helps avoid "hot spots" in the reflow process that char the flux residues and render them nearly impossible to remove.
- Cleaning agents can't clean areas they can't reach. Making sure the PCB layout provides at least some access for fluid flow in tight areas can mean the difference between success and failure.

Remember, if a PCB can't be cleaned, it can't be used.

SCORING

Where do you stand on the Jurassic food chain? That depends on your quiz score. Give yourself one point for every right answer, and *deduct* one point for every wrong answer.

If you scored 8-10, you are T-Rex.

You hunt and annihilate unwanted residues and particulates from your PCBs the same way the most ferocious and insatiable carnivore ever to step foot on the planet stalked his prey. You are a cleaning badass, slaying ionic residues that are tinier than the eye can see. But as a modern-day T-Rex, you use the correct cleaning agents instead of your teeth to destroy your quarry. 65 million years later, you are still way cool!

If you scored 0-6, you are a Stegosaurus.

It's a shame, really. You know that you *need* to slay ionic residues like T-Rex, but sadly your brain-to-skull size ratio is just that of a Stegosaur - not exactly efficient, but enough to survive. You choose a cleaning solvent without considering the soils you put on your PCB. It works some of the time… but the rest of the time you are driving up your labor costs as you try to fix your mistakes and hide from your predator, the not-so-friendly customer with contaminated boards.

If you scored LESS THAN ZERO you are a muddy swamp.

Yep…a dark, dank, murky swamp. You don't even qualify as a cold-blooded beast; you're just a big puddle of muck that nobody wants to get near. Look at yourself, will you? Your hands are filthy. Your lab coat is filthy. Your printed circuit boards? Filthy! Your mind? Forget it… we see where this is going. Obviously you don't understand the concept of cleanliness and its impact on printed circuit boards – or on your business!

But please don't drown yourself in despair. Keep taking our quizzes and one day you'll find electronics manufacturing technology as clear as…mud. (*Just kidding – we hope you're swift enough to get this joke. And the next one. for that matter…)*

PCB cleaning is a complex science, but The Printed Circuit Girls and Geeks seek out the best pale-ION-tologists to help simplify it and make it fun for everyone!

References:

1. Wikipedia – Hansen solubility parameters. http://en.wikipedia.org/wiki/Hansen_solubility_parameter

2. "A Review of Models for Time-to-Failure Due to Metallic Migration Mechanisms," E. Bumiller and C. Hillman, white paper downloaded from DfR website. Can be viewed online at http://www.dfrsolutions.com/publications/

3. "Electrochemical Migration on HASL Plated FR-4 Printed Circuit Boards" E. Bumiller, M. Pecht, and C. Hillman, *Journal of Surface Mount Technology*, Vol. 17, No. 2, pp. 37-41, April-June 2004. Can be viewed online at: http://www.dfrsolutions.com/publications/

4. "The Effects of Immersion Silver Circuit Boards Surface Finish on Electrochemical Migration," D. Cullen, *Proceedings of SMTA International,* September, 2004, pp. 497-507

5. "Analyzing and Predicting Electrochemical Migration Failures on Field Failure Returns," R. Michalkiewicz, *Proceedings of Pan Pacific Microelectronics Symposium,* January, 2008

6. "The Miniaturization Paradox" M. Bixenman, *Circuits Assembly Magazine*, May, 2011. Can be viewed online at: http://www.circuitsassembly.com/cms/component/content/article/159/11173-process-doctor