PCB “No-clean”- The Misnomer

Nick Nagrodsky
Hardware Design Engineer- Alstom Transport
AREMA C36 and C37 member
Nicholas.nagrodsky@transport.alstom.com
321-435-7230
1990 W. Nasa Blvd, Melbourne, FL, 32904

Number of words: 2867

Abstract

Due to updated regulatory guidelines and requirements, namely CENELEC and AREMA compliance; PCB assemblies are stressed more during the development process and environmental qualification testing than ever before. New product designs are subjected to a battery of electromagnetic susceptibility tests that were not previously required. How an electronic assembly is manufactured can mean the difference between meeting EMI/EMC compliance requirements and fielding a reliable product to customers, or struggling to meet industry standards and being plagued with difficult to reproduce field anomalies.

A key contributor to compliance testing and field anomalies can be the result of “no clean” flux and solder paste residues left on PCB assemblies after the manufacturing process. The generally accepted position by the manufacturing industry is that by definition, “no-clean” means electronics assemblies that are manufactured with these types of chemicals do not require cleaning post assembly. Testing and research has shown this is becoming a large issue, not just for the rail industry, but for the entire electronics industry. No clean residue when exposed to voltage potentials, humidity, and temperature cycling cause a reduction in impedance between circuits that can lead to susceptibility and isolation failures. If not addressed, it may result in an increase of field failures due to corrosion and other anomalies, such as “No Trouble Found” (NTF) tickets when assemblies are sent back to the suppliers. In order to expect the highest reliability and safety from rail signaling products, we must be proactive in preventing this misnomer regarding “no clean” assemblies.
Why No-Clean

During the electronic assembly process, a material known as flux is used during the solder process. This flux serves several purposes. It is used to facilitate heat transfer of the solder joint, provides wetting during the process, and aids in the removal of oxidation during soldering to achieve a solid solder joint. Similarly, solder paste is used to connect surface mount components to the pads of the PCB prior to soldering due to its sticky or tacky nature. When the assembly is heated, this paste melts and provides both a mechanical and electrical connection from the component terminals to the PCB pads. Prior to 1987 when the Montreal Protocol was agreed upon, electronic PCB assemblies used high solid content rosin flux and most assemblies were physically cleaned post soldering to remove any excess residue left from the soldering process. Because this Protocol was meant to protect the ozone layer by removing the use of substances that were known to cause ozone depletion, the use of high solid rosin flux was replaced with more environmentally friendly, lower manufacturing cost “no-clean” forms of flux and solder paste. With low solid content and the pitch of “no cleaning required”, much of the electronic industry rushed to switch to these new forms of flux and solder paste to save both time and money since added process steps such as cleaning were deemed unnecessary. An interesting observation is that portions of the military and other high reliability product manufacturers continued to require cleaning even though vendors and contract manufacturers proposed that these new fluxes and solder pastes were non-corrosive and cleaning the assembly was a waste of time and money.

Impact to the Rail Industry

As time has passed, the rail industry as a whole have increased their knowledge as technologies have advanced in product designs; offering safer, high reliability products that are more immune to the environments primarily driven by more stringent regulatory requirements and guidelines. An example of this would be around environmental and electromagnetic susceptibility tests. Agencies such as AREMA and CENELEC have provided increased clarity and definition for these environmental tests and require much higher standards than in the past. One can simply compare AREMA 2010 vs. AREMA 2013 versions and notice a new section, 11.5.2, which provides a suite of electromagnetic susceptibility tests not previously required in earlier released versions of AREMA. New requirements such as EFT (Electrical Fast Transients) and RFI (radio frequency interference) were included to better mimic the harsh field conditions that many Wayside and Onboard products must survive in for sustained periods of time. Meeting these new requirements can be a challenge but a disturbing pattern has begun to emerge in equipment manufactured using the no-clean approach. Why is there generally an increase in the number of NTF’s repair reports of returned products from customers?

Visual inspection of PCB assemblies may reveal a sticky or white residue on the surface of the assembly. A few examples of this residue can be seen in Figure 1 below:
What is this residue and more importantly what are its effects on the performance of electronic assemblies?

**Residue**

When there is insufficient heat during the solder process, either in temperature or duration, it results in the formation of free metal salts from remnants of the activators in the flux or solder paste [1]. When the metal oxide and acid from the flux is combined and heated during the soldering process, the resultant residue is typically salt and water. The water evaporates due to the localized heat leaving the salt. Instead of the salts being contained as they are in the solder process when using rosin fluxes, the residue from no-clean flux hardens and prevents the salts from being encapsulated. The salts are now free to live on the assembly, absorb water, and provide the means for electro-migration, a.k.a. dendritic growth. Dendritic
growth is the bane of the electronic industry, particularly in areas where high reliability is required, such as the rail industry. This dendritic growth can cause functional or other related issues after years of reliable operation. What causes these dendrites to form?

The Triple Threat

Dendrites form when 3 factors combine. First, there must be a voltage potential between 2 electronic components or circuits. Think of this as the anode and cathode similar in concept to a diode. There must be a junction or medium by which current can begin to flow, regardless of how small that current may be. Moore’s law has proven electronic components will continue to be ever more densely populated and packed closer together on PCB assemblies, decreasing the distance between components and increasing the opportunities of current flowing through unintended paths. Second, an actual medium is needed to allow current to flow. This is where the residue begins to provide a high impedance path. No-clean flux and solder paste is touted as non-corrosive and has a low solid content. The typical breakdown of flux residue by weight is 2% wt. solids, 96% wt. alcohol, 1% wt. water and 1% wt. additives [2]. The assumption is that all acids and ester oils evaporate during the solder process, leaving the assembly free of residue containing any significant amounts of solids or corrosive content. When the solder process is not performed at a high enough temperature or at the lower end of the acceptable temperature profile, the third factor, humidity or moisture, comes into play. Humidity provides the necessary moisture for this flux residue to allow the formation of metal salts which results in the formation of dendrites. The dicarboxylic acid part of the flux residue does not fully evaporate and since acid is dissolved in water, current flow can occur. Initially this current flow is minute, but as time passes and the crystalline structure begins to grow and a path continues to form, the impedance of this unintended current path continues to decrease. Further exacerbating this reaction, long-chain organic ester oil allows for dust and other particles to become trapped and increase the likelihood of absorbing moisture. No-clean fluxes and solder paste residue require much more aggressive cleaning in order to sufficiently remove this residue and simple washing with water can actually be worse than not cleaning the assembly at all. This is because water will simply push these metal salts around the assembly and under components instead of sufficiently removing them. Now that there is an understanding of the theory of how no-clean flux and solder paste may affect assemblies, what objective evidence is available showing examples of this process in action?

Industry Testing

Design of Experiments were conducted to evaluate the impact of no-clean flux with no post-solder cleaning on high voltage isolation. A PCB assembly with no active components which was manufactured with a particular no-clean flux and no post-solder cleaning was used for evaluation. There was visible flux residue between adjacent pins of a through-hole connector. Surge testing was performed between these connector pins at increasing voltage levels until the arc point was discovered. With no humidity or cleaning, the arc point was found to be ~3300VAC. At this point, the assembly was put inside a thermal chamber and temperature cycled from -40°C to +70°C over a period of 2 hours. Upon completion of the temperature cycle, the PCB assembly was surged tested again while at elevated temperature. This time the assembly’s arc point was 1300VAC. This represents a loss of 2000VAC, or roughly 60% of its isolation, simply by elevating the PCB assembly temperature and allowing flux residue to remain on the
assembly. Next, the assembly was allowed to cool and reach ambient temperatures and then surge tested again. The threshold increased to 2000VAC, still well below the initial level of 3300VAC.

Investigating other external industry research revealed similar results had been observed by others. Johanson Dielectrics Inc. published a white paper, appropriately titled, “Arc Season and Board design Observations” [3], in which they investigate and describe the loss of isolation due to both no-clean solder paste residues as well as improper board design and assembly techniques. Their viewpoint tends to lean on the side of water-soluble flux causing more issues when not properly cleaned and ways to improve isolation through PCB design techniques. However, they also captured useful data as a result of before and after testing when no-clean solder paste residue is left on an assembly and exposed to high temperature and humidity. The test setup consisted of high voltage 1206 3kV 120pF surface mount capacitors installed on a PCB with no-clean solder paste. AC breakdown testing was performed on 80 samples and a bell curve was drawn showing the voltage limit before arc occurred. In Figure 2 below, notice a tightly coupled range of values with a mean arc breakdown of ~3000VAC as expected.

![Figure 2](image)

Johansen Dielectrics next took another sample size of 80 capacitors installed on a PCB with no-clean solder paste and exposed them to 85°C at 85% relative humidity for a period of 48 hours. After exposure, the assemblies were left at ambient for 1 hour before performing the AC breakdown testing to determine the effects of high humidity and temperature on the assembly when no-clean solder paste was used. See Figure 3 below for the AC breakdown distribution curve:
The mean AC breakdown voltage was ~2680VAC, representing an average decrease of 320VAC or ~10%. However, the distribution curve was significantly wider with breakdown occurring over a wide range well below the mean threshold. Their research notes the breakdown values were below the specification limit of 1500VAC and a broader distribution curve. Now envision the varying degrees of reliability if these 80 capacitors were placed on 80 different PCB assemblies made with no-clean solder paste/flux and put into revenue service for 20 or 30 years in a high temperature/high humidity environment like the rail industry demands.

**Solutions**

What options are there to minimize the impact of no-clean fluxes and solder paste? One possible option is to prevent the assembly from being installed in environments with varying or high humidity. This can be accomplished through the use of conformal coating or by placing the equipment in an environmentally controlled location. One caveat for conformal coating is that the PCB surface must be extremely clean when the conformal coating is applied to achieve proper adhesion and to avoid trapping surface contaminants and moisture within the PCB assemblies that provides the environment for electro-migration. Other factors must be taken into consideration when conformal coating as well, such as the thermal blanket that has now been placed on electronic assemblies, many of which can have high power dissipation. Additionally, the likelihood of installing environmental control equipment in all instances and locations seems both improbable and costly to the end user.

Another option would be to perform the solder process at a slower, albeit higher final temperature. "The Effects of Solder Flux Residues on Corrosion of Electronics" [2] provides interesting data to support this approach. Figure 4 shows the amount of flux residue remaining on the assembly based on the temperature reached during soldering. Figure 5 shows the percentage of residue remaining when varying the rate of temperature rise, showing that a slower temperature rise allows sufficient time for evaporation to occur.
While this option appears attractive on the surface, it is not without its own deficiencies. During wave soldering, there is only direct contact with the bottom side of the assembly, so any residue that remains on the topside does not automatically get removed as it is not exposed to the same temperatures as the bottom side. Increasing overall temperature or extending the duration of the soldering process requires significant investigation, ensuring all components on the assembly can endure the increased...
temperatures and durations achieved to sufficiently remove enough residues while not degrading component life.

The next option, when combined with the previous one, offers the best compromise. If solder processes are regularly monitored and there is a reasonable increase in temperature during soldering, a combination of visual surveillance as well as regularly scheduled chemical composition testing on a physical assembly built on the manufacturing line will provide a closed loop process as to whether the solder process is sufficiently removing enough residue. One should be careful when discussing such standardized chemical testing processes such as Surface Insulation Testing (SIR) as this does not guarantee that an electronic assembly will be sufficiently clean. IPC approved SIR testing is used to test the process, not the product. A standardized comb pattern PCB is applied with high temperature and humidity for a period of time (typically 16 hours to several weeks) with some amount of DC voltage (typically 10-100 VDC) applied between 2 locations on the assembly and then the impedance is measured between the locations. The inference is that if the comb pattern meets the SIR test limits, the electronic assemblies must also meet the test limit and be free of any significant amounts of residue or chemicals capable of producing corrosion of electro-migration. The issue with this approach is that it does not take into account residues that become trapped under or inside components or varying temperatures due to thermal mass of components as well as top-side vs. bottom-side temperatures. This test method only provides a general analysis of the soldering process and tends to provide a false sense of security. Another form of IPC testing, known as Ion Chromatography provides a much more detailed analysis as to the actual chemical composition of residues left on the PCB assemblies being manufactured. However, one must also be careful when having this test performed, as the assembly is entirely submersed in deionized water.

One must be careful when solely relying on standards such as IPC and legacy manufacturing processes and the “We’ve done it this way for XX years” mentality. Per this writing, IPC-A-610, “Acceptability of Electronic Assemblies”, Section 10.6 outlines PCB cleanliness and pass/fail criteria based on flux residues. Many caveats are made that if no-clean flux is used, the criteria is only defined as a failure if the flux inhibits electrical connections or inhibits visual inspection. Misleading notes such as, “White residues resulting from no-clean or other processes are acceptable provided that residues are from chemistries used have been qualified and documented as benign…” are used. The problem arises from the customer and contract manufacturer deferring to the no-clean flux or solder paste vendor for their datasheet as evidence that these residues are harmless. For commercial use, this is likely the case. But for long-term, high reliability products where operating conditions can be extreme, we must push for the best product we can for our industry.

References
1. Defluxing Déjà vu- Michael Konrad, Aqueous Technologies
2. Effect of Solder Flux Residues on Corrosion of Electronics- Kirsten Stentoft Hansen, Morten S. Jellesen, Per Moller, Peter Jacob Schwencke Westermann, Rajan Ambat

List of Figures
1. Figure 1- Flux residue examples
2. Figure 2- AC breakdown curve for No Clean with chip and soldermask- Arc Season and Board Design Observations- John Maxwell, Enrique Lemus- Johanson Dielectrics Inc.
3. Figure 3- AC breakdown curve for No Clean with chip and soldermask at 85°C/85% Humidity- Arc Season and Board Design Observations- John Maxwell, Enrique Lemus- Johanson Dielectrics Inc.
4. Figure 4- Flux residue amounts after heating to various temperatures- Effect of Solder Flux Residues on Corrosion of Electronics
5. Figure 5- Flux residue amounts after various temperatures rise rates- Effect of Solder Flux Residues on Corrosion of Electronics
PCB “No Clean” - The Misnomer

Nick Nagrodsky

ALSTOM
What Is It?

- Flux- Used when soldering electronic components to a PCB (printed circuit board) to facilitate heat transfer of the solder joint as well as wetting and removal of oxidation during the solder process.

- Solder Paste- Used for attaching SMT components to a PCB. Initially adheres components in place due to its sticky nature, then its heated and melted and forms an electro-mechanical connection.

Claimed Benefits of No-Clean

- “No post solder cleaning required.”
- Cost/process savings.
- Non-corrosive to the PCB surface.
- Low solid content (acids and ester oils) reduces contamination. (Flux makeup is 2%wt solids, 96%wt alcohol, 1% wt water, 1%wt additives).

Why Do We Care?

- Updated regulatory requirements and guidelines (e.g. AREMA 2013 11.5.2 and CENELEC EN50121) require that electronic assemblies be stressed more during environmental qualification testing than in the past.

- Visual residue is seen on PCB assemblies after manufacturing. What is this residue and what is the impact?

- A/B testing when residue is present on an assembly shows wide variation in isolation/immunity.

- What options are there to prevent this from impacting product reliability?

New to AREMA 2013 (cont.)

<table>
<thead>
<tr>
<th>DC Power Ports Immunity</th>
<th>AREMA® C&amp;S Manual</th>
<th>Part 11.5.2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Environmental Exposure Class</td>
<td>Basic Standard Test Parameters</td>
<td>Performance Criteria</td>
</tr>
<tr>
<td>Electrical Fast Transient</td>
<td>Internal</td>
<td>External</td>
</tr>
<tr>
<td>Surge</td>
<td>Internal</td>
<td>External</td>
</tr>
<tr>
<td>Conducted</td>
<td>Internal</td>
<td>External</td>
</tr>
</tbody>
</table>

Flux Residue Examples

- Note 1: All tests shall be done without external surge protection supplied, unless the external surge protection is an integral part of the product.
- Note 2: Include appropriate combinations of Line to Line, Line to Earth, Line to Circuit Ground, etc.
- Note 3: Reference the basic standard for test method details including coupling methods.
Flux Residue Examples

AREMA 2016 Annual Conference & Exposition

What Causes The Residue?

Insufficient heat during solder process
Flux/paste activators leave water + salt remnants on PCB
Water evaporates leaving free metal salts
No-clean salt residues live on PCB and absorb water
Free metal salt + water provides means for dendritic growth

Source: Flux residues can cause corrosion on PCB assemblies - www.edn.com

AREMA 2016 Annual Conference & Exposition

Additional Factors

Moore's Law - Electronic Miniaturization

Triple Threat for Dendritic Growth

Source: Moore's Law - Electronic Miniaturization

Industry Testing

- High voltage 1206 3kV 120pF X7R capacitors mounted on PCB’s.
- Sample of 80 components
- Ambient humidity/temperature
- Baseline AC breakdown
- ~10% variance
- Average breakdown 3002VAC

Source: Arc Season and Board Design Observations
John Maxwell and Enrique Lemus - Johanson Dielectrics Inc.
AREMA 2016 Annual Conference & Exposition

Industry Testing

- High voltage 1206 3kV 120pF X7R capacitors mounted on PCB’s.
- Sample of 80 components
- High temp/high humidity
- ~30% variance
- Average breakdown 2681VAC

Source: Arc Season and Board Design Observations
John Maxwell and Enrique Lemus - Johanson Dielectrics Inc.
AREMA 2016 Annual Conference & Exposition
Industry Testing

Flux residue trapped between capacitor bodies and solder mask during reflow soldering when the flux is liquid. This is due to capillary forces wicking the flux between the solder mask and capacitor body.

Solutions

Option 1-
Change manufacturing process to remove more residue.

Downside-Significant time and cost to ensure all components can handle new temp profile.

Option 2- Test for residue and contaminants.

- SIR Testing- High temp/high humidity voltage bias applied to look for current leakage.
- ROSE Testing- Works well for water soluable/rosin fluxes, not for no-clean. Results vary based on board size and technology used.
- C3/Ion Chromatography- Analyzes ionic residues to determine the chemical makeup on the assembly (chlorides, bromides, etc.).

Downside- Significant cost to chemical test all assemblies and testing can be destructive.

Option 3- Combination approach.

- Moderately increase soldering temperatures. I.E. 120°C to 170°C cuts residue almost in half.
- Visual inspection. Any sticky/white residue post soldering should be flagged as part of quality control.
- Regularly scheduled chemical analysis should be performed based on pre-defined limits.

Option 3 offers the lowest cost to implement while providing the most impact to ensure long-term reliable operation for the end user.