TO KILL A CIRCUIT BOARD: PERILS IN MANUAL SOLDERING & CLEANING PROCESSES

ABSTRACT

Manual soldering and cleaning processes are among the least controlled processes in printed circuit board assembly. As a result, they create special challenges to both quality and long term reliability. This paper describes some of those key challenges and provides ways to address them in assembly to minimize problems. The topics to be covered include:

- Design Considerations
- General Manual Soldering Recommendations
- Material Selection and Qualification Criteria
- Cleaning Recommendations
- Process Monitoring and Control
- Relevant Industry Standards
- Case Study

Through awareness of the issues, proper design and appropriate material selection and process control, companies can successfully use manual soldering and cleaning processes in high reliability products. Without proper planning, however, these processes can lead to a path of ruin.

Key words: manual soldering, hand soldering, manual cleaning, spot cleaning

INTRODUCTION

Manual soldering must be optimized for each component on an assembly, and once the procedures are developed for a given component, they must be rigidly followed. The optimized conditions are dependent upon the surface finish, amount of copper present and the component to be soldered. To determine the optimum manual soldering process, extensive analysis must be performed to ensure no damage occurs to the component, the PCB, or neighboring components. New equipment or materials may be required and new techniques may be employed to protect other components.

DESIGN CONSIDERATIONS

Surface Finish

If the design bill of materials (BOM) contains any wires or components which require manual soldering, numerous tactics can be used to maximize quality and reliability. Ensuring reliable soldering begins with the selection of the PCB surface finish. The surface finish influences the process yield, the amount of rework, field failure rate, the ability to test, the scrap rate, and the cost. Selecting the lowest cost surface finish only to find later that the eventual total cost is much higher should be avoided. The selection of a surface finish should be done with a holistic approach that considers all important aspects of the assembly. In this paper, the focus will be on finishes which can impact manual soldering but some of the many considerations for selecting a surface finish are:

- Cost sensitivity
- Volume of product and finish availability
- SnPb or Pb-free process
- Is shock and drop a concern?
- Is high yield ICT (in circuit test) important?
- Is direct wire bonding required?
- Is the user environment (corrosion) a concern?
- Are fine pitch components (<0.4 mm) used?
- Is wave or selective solder required (PCB > 0.062")?
- Are cosmetics of the PCB a concern?
- Is manual soldering required?

Some of the most popular Pb-Free surface finishes are:

Surface Finish	*Manual Soldering Challenges*
ENIG (Electroless nickel , immersion gold) and ENEPIG (electroless Pd added)	Slightly less solder spread, incomplete pad coverage
Immersion silver (ImAg)	None
Immersion tin (ImSn)	None
Organic solderability preservative (OSP)	Incomplete pad coverage, PTH hole fill, copper dissolution
Pb-free HASL (SAC)	Copper dissolution
Pb-free HASL (Sn100C)	None

Table 1 Soldering Challenges by Surface Finish

On ENIG, ENEPIG, and OSP finishes, incomplete coverage of solder pads is not uncommon. Although exposed copper is considered primarily a cosmetic issue, it can be a problem in certain environments, especially where sulfur is present.

Plated Through Hole (PTH) Fill

The primary challenges with manual soldering to OSP are achieving 100% plated through hole fill and avoiding copper dissolution. Achieving 100% PTH fill is difficult with manual soldering to an OSP surface finish because the OSP deteriorates with each thermal cycle. This allows the underlying copper to oxidize which reduces solderability and drives the need for very active fluxes. A typical product experiences two or more thermal cycles, such as bake, reflow, and wave, by the time manual soldering occurs. Since solder fill is also driven by capillary action, design and component parameters are important. Hole diameter, hole aspect ratio, and adequate thermal relief for large copper planes should be considered by the designer. When practical, designers should utilize thermal relief on all copper planes since this reduces the thermal transfer rate between the PTH and the copper plane. This allows for easier solder joint formation during soldering (especially for Pb-free) and results in better hole fill. Figure 1 shows an example of a design with breaks in the copper plane to provide some thermal relief.

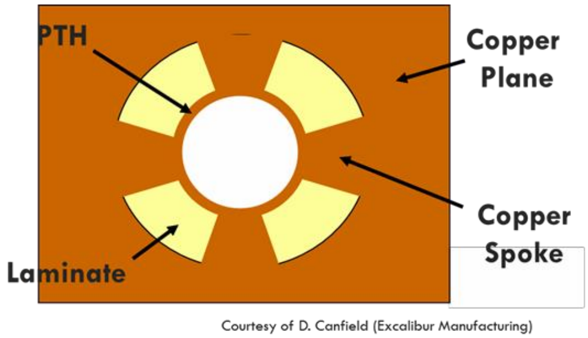

Figure 1 Thermal Relief Design Example

Copper Dissolution

Another design consideration when manual soldering is the amount of Cu present. This affects copper dissolution, which is the reduction or elimination of surface copper conductors due to repeated exposure to Sn-based solders. It is a significant concern for industries that perform extensive rework or manual soldering. Contact time with the solder is the major driver of dissolution with indications of a 25-30 second limit maximum allowable time before damage begins. Most manual soldering processes provide little or no control over contact time which is driven by design, equipment, materials, and operator skill. To avoid dissolution, place limits on contact time and rework processes, do not use OSP finishes, and use an SNC alloy for soldering. SNC alloys, like SN100C, have a reduced rate of dissolution of Cu due to a reduced diffusion rate through the Sn- Cu intermetallics. The desire to avoid copper dissolution is one reason why most Pb-free HASL, manual and wave/selective solder processes have converted to SNC alloys.

In Figure 2, Jasbir Bath[2] showed a completely dissolved copper pad and trace for an assembly with ENIG Plating after a 60 second exposure in a 274ºC solder fountain. Additional studies have shown that dissolution begins at solder contact times of less than 20 seconds[6].

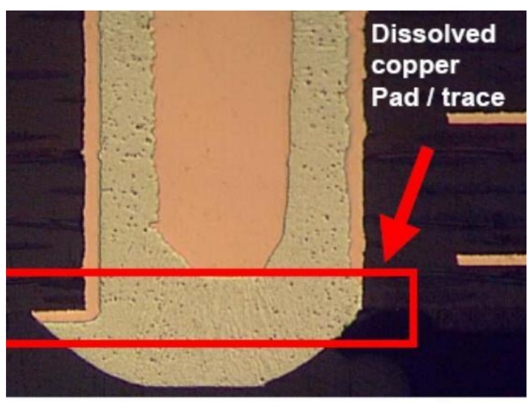

Figure 2 Cu Dissolution in a Solder Fountain

Component Selection and Placement

Other important design considerations for manual soldering include proper component and wire selection and placement. Component features that impact manual soldering include the size, type, and plating of the component leads and the lead to hole ratio. Kester Solder[1] published a metal solderability chart which illustrates a relative solderability assessment of various lead finishes. A summary is shown in Table 2.

Metal Surfaces	*Solderability*
Platinum Gold Copper Tin Solder Palladium Silver	Easy to Solder
Nickel Brass Cadmium Lead Bronze Rhodium Beryllium Copper	Less Easy to Solder
Nickel-Iron Kovar	Difficult to Solder

Table 2 Metal Surface Solderability

For wire selection, consider using solid wire or the lowest strand count for a given AWG size. For example, a commonly used wire size for standard hookup applications is 24 AWG. It is available in configurations from 7/32 (7 strands of 32 AWG) to 41/40. The 7/32 size is preferred because it's less susceptible to flux wicking corrosion.

Designers must also provide adequate spacing between manually soldered components and adjacent components and circuitry. This allows the best soldering equipment to be used and it also prevents damage to nearby components and circuitry. Ceramic capacitors (MLCCSs) are especially vulnerable to thermal shock cracks which occur due to a rapid, excessive change in temperature. This can occur during reflow, cleaning, wave solder, manual solder, and rework processes. Cracks result from the inability of the capacitor to relieve stresses during transient conditions. Transient thermal analyses have shown that the maximum tensile stress occurs near the end of the terminations. Cracks manifest themselves in three ways:

- Visually detectable (rare)
- Electrically detectable
- Micro-crack (worst-case)

Variations in voltage or temperature drive crack propagation. If detectable during test, the cracks may cause an increase in electrical resistance or a decrease in capacitance. **Error! Reference source not found.** shows an example of a thermal shock crack near the end termination.

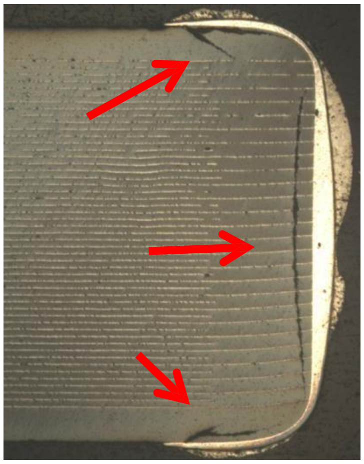

Figure 3 Ceramic Capacitor Thermal Shock Crack

As mentioned before, maximizing space between MLCCs and any manual soldered components is critical. To further reduce the likelihood for thermal shock crack formation, designers should aim for a maximum case size of 0805 for SAC305 and a maximum package thickness of 1.2 mm. C0G and X7R dielectrics are also preferred since they are stable and temperature compensating. Use the manufacturer's recommended bond pad dimensions or smaller since smaller bond pads reduce the rate of thermal transfer. Most MLCC manufacturers do not recommend using a soldering iron to attach or rework MLCCs. This practice should be avoided at all costs.

Finally, the NASA Electronic Parts and Packaging (NEPP) program has published a comprehensive analysis of these challenges in a paper titled "Effect of Manual-Soldering-Induced Stresses on Ceramic Capacitors (Part I)[5].

MANUAL SOLDERING RECOMMENDATIONS

Along with the discussed design considerations to minimize the impact of manual soldering on the assembly, using proper manual soldering tools also improves the quality of the manual soldering process. Manual soldering processes may use solder irons, hot air tools, and solder fountains. Variation occurs from operator to operator so training and practice are critical to ensure uniformity across all assemblies.

Soldering Iron Recommendations

Some general soldering iron recommendations include:

- Use soldering irons with the greatest thermal recovery, such as a high power soldering iron with a rapid feedback loop
- Use the largest tip commensurate with the size of the joint being soldered and with the available working space
- Use custom tips if needed
- Use the largest cored solder wire diameter for the size of the joint and available working space
- Avoid the use of liquid fluxes whenever possible

Typical tip temperatures for Pb-free solder range from 650-800F so it's extremely easy to damage boards and components.

Figure 4 from MetCal[3] shows basic recommendations for solder tip size. Note that an optimal tip size is approximately the same size as the lead or pad to be soldered. If the tip is too large, it can encroach on and damage the solder mask. If the tip is too small, more contact time is needed due to the smaller tip area and its' reduced ability to transfer heat.

Figure 4 Solder Iron Tip Size Recommendations

Figure 5, also from MetCal, helps illustrate why no-clean flux-cored solder wire seldom works as well as RMA-cored solder wire. To reduce visible residues and minimize corrosion risk, no-clean cores are typically much smaller than RMA cores. Larger cores can be requested to improve solderability but they do leave more residues behind.

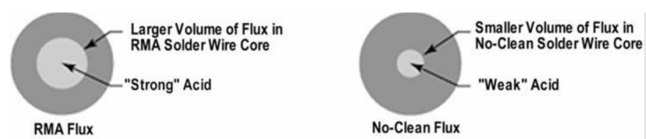

Figure 5 Flux Core Size vs Flux Type

Manual Soldering Accessory Recommendations

The following images show accessories, like needle tip dispensers, solvent dispensers, sponges, brushes, and tip cleaners, which must be routinely replaced or cleaned to prevent cross-contamination. Using needle tip flux dispensers is not recommended since there is poor control over flux volume and flow. Excess flux can run over the PCB and under components leaving non- activated flux residues behind. Most solvent dispense bottles are never cleaned; they're simply refilled over and over again. Sponges, brushes and tip cleaners should be frequently replaced and different solder alloys should not share the same materials.

Figure 6 Manual Soldering Accessories

Prior to manual soldering, consider using a portable preheater to shorten contact time of the solder iron tip with the assembly, to help fully activate the flux and to reduce risk of thermal shock. A portable preheater from Zephyrtronics is shown in Figure 7.

Figure 7 Zephyrtonics Portable Preheater

Solder preforms can be used to improve manual soldering uniformity for both SMT and PTH joints. They are available from many solder suppliers and can be made in the size, solder alloy, and flux of choice. Solder preforms with flux can be placed on a pad or on a PTH lead on the topside of the PCB where they can improve topside PTH fill. Using preforms controls both volume of solder and volume of flux applied. Examples of preforms are shown in Figure 8.

Figure 8 Solder Preforms

Finally, avoid using liquid flux in manual soldering. As mentioned earlier, when using needle tip flux dispensers, it is extremely difficult to prevent flux from running under and around adjacent components. If liquid flux is needed, implement methods to ensure precise delivery such as flux pens or automated dispense equipment. Use of a portable preheater provides uniform heating, which volatilizes as much of the liquid as possible. It is absolutely vital to select a flux designed and validated for manual soldering processes. This is typically NOT the same material as the wave solder flux, which is designed to hold up through preheat and dual wave contact.

MANUAL SOLDERING MATERIAL SELECTION CRITERIA

All process materials used should be designed and validated for manual soldering applications. Wave solder fluxes and reflow pastes were designed with long, stable preheat profiles in mind. Surface insulation resistance (SIR) data should be reviewed but is not considered a sufficient evaluation for manual soldering processes. The SIR data provided by suppliers is typically performed on test coupons which are not representative of real product and the testing is not performed in conjunction with other process materials. On an actual assembly, there is a complex interaction of pastes, fluxes, adhesives, masks coatings, and other materials. The user must verify compatibility of manual soldering materials with all of these adjacent materials.

Manual soldering materials must also be selected based on whether tin/lead or Pb-free alloys and processes are used. For Pb-free, the SNC alloys are recommended to reduce the likelihood of copper dissolution.

Ideally, all manual soldering materials are free of halogens/halides. This class of materials creates the highest risk for corrosion-related failures. Since there is not yet a common definition for what constitutes halide-free or halogen-free, look carefully at the data sheets for material contents. As previously mentioned, if a less solderable lead finish and surface finish are selected, stronger, more active fluxes may be required. Flux type and acid number give some indication of relative flux activity. Shea et al provide an excellent overview of criteria *in Selecting Fluxes for Lead-Free Wave Soldering*[4]. Table 3 shows how Indium Corporation lists properties for their core flux in solder wire. Note the two different columns for halogen content.

Core Flux Formulas

Flux Number	J-STD-004	J-STD-004B	QQ-S-571f	Halogen Content	JIETA ET-7304 Halogen Free	Residue	Residue Removal	Preferred Alloys	Application/ Comments
CW-102	ROL0	ROL0	"R"	< 50 ppm	Yes	light amber	NC, So, Sa*	SnPb, Pb-Free	Military type "R" for legacy applications, very low activity

Table 3 Cored Solder Wire Properties

Finally, supplier relationships should also be considered. Developing a partnership with material suppliers can result in improved material selection, compatibility, and overall process results.

Process Material Qualification Recommendations

Review and verify surface insulation resistance (SIR) data for all process materials selected. Then, validate compatibility and performance of all the materials combined using SIR testing. IPC has several test vehicles available for both SIR and cleanliness evaluation.

The IPC-B-52 test board shown in Figure 9 is intended to be a process qualification vehicle, where the materials of construction and source of test boards are representative of the end product. The IPC-B-52 (IEC TB-57) kit includes the latest generation of test coupons and is similar to designs that NPL, Rockwell Collins, & IBM have used for this purpose. The kit consists of a main SIR board, a test board, an IC (ion chromatography) test coupon, and a solder mask adhesion and SIR mini-coupon. PCBA package types include:

- 0402 – 1206
- QFP (no 0.4mm pitch)
- SOICs and BGAs
- Through-Hole Header
- Comb patterns (5 mil)

This testing is not specifically called out in any current IPC-TM-650 test method. IPC-9203 is available as the Users Guide to the IPC-9202 and the IPC-B-52 Standard Test Vehicle package. Once successful coupon testing has been completed, results should be verified using an actual product.

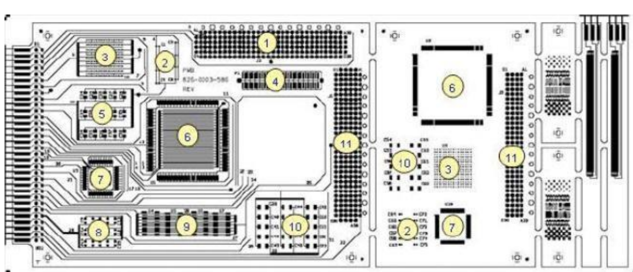

Figure 9 IPC-B-52 Test Vehicle

Ideally, thermocouple-based manual soldering profiling is performed for all processes and all products. However, this can be time-consuming, expensive, and destructive. Actual PCB surfaces, leads, and surrounding component temperatures can be non-destructively verified using small, temperature labels like those shown in Figure 10. These are widely available, inexpensive, and easy to use. They also come in a wide range of sizes and temperature ranges. As a given temperature is exceeded, the indicator dots changes color. After soldering, simply remove the label and record the results.

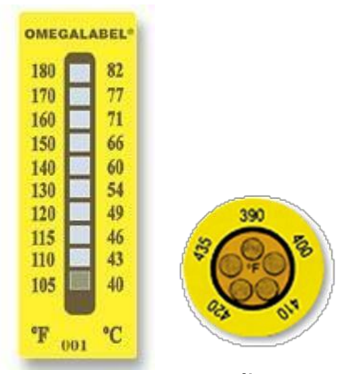

Figure 10 Temperature Indicator Labels

Cleaning Recommendations

For high reliability applications, robust selection and validation of manual solder processes is recommended. If this work is performed, no-clean materials may safely be used in most, but not all, products. Otherwise, in line cleaning or batch cleaning is recommended. Since most high-volume manufacturers prefer no clean processes, batch cleaning dominates in the industry since the costs and requirements for in-line cleaning are usually justifiable only for high volumes.

Some cleaning variables to consider include:

- Use of pure deionized water (DI)
- Use of saponifier (type and concentration)
- Temperature (120F to >175F)
- Nozzle pressures
- Agitation
- Time

Temperature and pressure of the DI water and the angle at which the water strikes the assembly vary with the cleaning equipment used.

The manual removal of flux residues is *not* recommended for high reliability applications. Manual cleaning may result in cosmetic improvements but usually simply disperses flux residues over a wider area.

In the typical manual cleaning process, some type of solvent spray is used to loosen flux residues. This may be followed by further cleaning using isopropyl alcohol (IPA) and a soft bristle brush. This type of manual cleaning process represents a reliability risk. The brushes used in manual cleaning are not routinely cleaned or maintained and simply transfer contamination from one area to another. And, when using no clean fluxes, poorly removed residues are more likely to drive corrosion failures than if the residues are left intact. Finally, rinsing is rarely performed in manual processes. If rinsing can't be performed, cleaning will not be very successful.

If flux residues must be moved manually, a four step process of wet, scrub, rinse, and dry is recommended. Use some form of dispensing system for the solvent to control the flow and volume. Ensure that the material is fresh and pure each time and never sits in a typical dispense or spray bottle since those bottles are rarely, if ever, cleaned. Brushes used in the process should be routinely replaced. Some commercial manual cleaning solvents come with their own brush which guarantees that a fresh, clean brush is used with each canister.

Cleaning Challenges

Contamination is believed to be one of the primary drivers of field issues in electronics today. It induces corrosion and electrochemical migration or ECM. Contamination-related failures are frequently intermittent in nature. This intermittent behavior may manifest itself as no-fault-found (NFF) returns, is driven by self-healing behavior and is, therefore, very difficult to diagnose. These failures are pervasive and have been observed on batteries, LCDs, PCBAs, wiring, switches, under coatings, over coatings, etc. The problem is expected to get worse as continued reductions in pitch between conductors makes future packaging more susceptible. Increased use of leadless and bottom termination components (QFN, land grid array, etc.) results in a reduction in standoff height which reduces the efficiency of cleaning, if cleaning is even performed. Contaminants are concentrated under these low standoff devices. Increased product sales are occurring in countries with polluted and tropical environments; but since air pollution is on the rise worldwide, no companies are immune from the problem.

Cleanliness Standards

Unfortunately, cleanliness standards continue to be a challenge for the industry. There has been an extensive effort to update them but they still lag the needs of the industry. IPC-5704, Cleanliness Requirements for Unpopulated Printed Boards, was revised and released in late 2010. This standard addresses bare board cleanliness but leaves innerlayer requirements up to the customer and is not referenced by any other standard. So, it must be specified individually. IPC-CH-65B, Guidelines for Cleaning of Printed Boards and Assemblies, was significantly updated in 2011. It contains guidelines for all types of cleaning including cleaning after manual soldering but no standards have been revised with updated pass/fail limits for cleanliness of assembled boards.

For companies who lack their own cleanliness requirements, contamination is usually controlled through specifications such as IPC-6012 and IPC-J-STD-001. These were based primarily on the original military specification of 10 mg/in2 of NaCl ‘equivalent’ which resulted in ~two megaohm surface insulation resistance (SIR). Board cleanliness after solder resist has to meet the requirements as *specified* by the customer. IPC-6012B specifies a Resistance of Solvent Extract (ROSE) method, defined by IPC-TM-650 2.3.25 and specifies this measurement should be performed on production boards every lot where:

- Class 1 boards: Sampling Plan 6.5
- Class 2 and 3 boards: Sample Plan 4.0

Under the sampling plan requirements, for example, if a lot contains 500 panels of a Class 2 product, eleven panels should be subjected to ROSE measurements for cleanliness testing.

Process Controls

ROSE is the least sensitive of ionic measurement techniques and is not best practice for cleanliness assessment. The best practice is to assess and control contamination through ion chromatography (IC) testing, the "gold standard." Some, but very few, PCB manufacturers and assemblers qualify lots based on IC results. Larger numbers use IC to baseline ROSE, Omegameter, or Ionograph (R/O/I) results. Ideally, process and lot qualification is performed with R/O/I and then is periodically recalibrated with IC at some frequency like weekly, monthly, or quarterly.

Furthermore, ROSE, Omegameter, and Ionograph tests DO NOT detect WOAs (weak organic acids) found in no clean fluxes. Successfully passing these "cleanliness" tests does not ensure cleanliness on an assembly at all. These types of "bulk" tests are best used for monitoring process equipment. R/O/I tests are most suitable for bare PCB cleanliness testing and for finding halide residues. Some halides are introduced into the board by design or by low quality and cannot be removed with cleaning by the user; they will be released later at soldering temperatures. Ion Chromatography is the only test that finds and quantifies WOAs. Unfortunately, there are no uniform or standard accept/reject limits for contamination on assemblies.

RELEVANT INDUSTRY STANDARDS

There are numerous IPC industry test methods for assessing fluxes, wires, manual soldering stress, and cleanliness. Some or all of these may be used by suppliers and end users depending on need. They include:

SECTION 2.3 - Chemical Test Methods

- TM 2.3.13A Determination of Acid Value of Liquid Solder Flux- Potentiometric and Visual Titration Methods - 6/04
- TM 2.3.25D Detection and Measurement of Ionizable Surface Contaminations by Resistivity of Solvent Extract (ROSE) - 11/12
- TM 2.3.27 Cleanliness Test - Residual Rosin - 1/95
- TM 2.3.27.1 Rosin Flux Residue Analysis-HPLC Method - 1/95
- TM 2.3.28B Ionic Analysis of Circuit Boards, Ion Chromatography Method - 11/12
- TM 2.3.28.1 Halide Content of Soldering Fluxes and Pastes - 6/04
- Reaffirmed
- TM 2.3.32D Flux Induced Corrosion (Copper Mirror Method) - 6/04
- TM 2.3.33D Presence of Halides in Flux, Silver Chromate Method - 6/04
- TM 2.3.34C Solids Content, Flux - 6/04
- TM 2.3.34.1B Percentage of Flux on/in Flux-Coated and/or Flux-Cored Solder - 1/95
- TM 2.3.35C Halide Content, Quantitative (Chloride and Bromide) - 6/04
- TM 2.3.35.1A Fluorides by Spot Test, Fluxes - Qualitative - 6/04
- TM 2.3.35.2A Fluoride Concentration, Fluxes - Quantitative - 6/04

SECTION 2.4 - Mechanical Test Methods

- TM 2.4.14 Solderability of Metallic Surfaces - 4/73
- TM 2.4.14.1 Solderability, Wave Solder Method - 3/79
- TM 2.4.14.2A Liquid Flux Activity, Wetting Balance Method - 6/04
- TM 2.4.36C Rework Simulation, Plated-Through Holes for Leaded Components - 5/04
- TM 2.4.37A Evaluation of Manual Soldering Tools for Terminal Connections - 7/91
- TM 2.4.37.1A Evaluation of Manual Soldering Tools for Printed Wiring Board Applications - 7/91
- TM 2.4.37.2 Evaluation of Manual Soldering Tools on Heavy Thermal Loads - 7/93

- TM 2.4.46A Spread Test, Liquid or Extracted Solder Flux, Solder Paste and Extracted Cored Wires or Preforms - 6/04
- TM 2.4.47 Flux Residue Dryness - 1/95
- TM 2.4.48 Spitting of Flux-Cored Wire Solder - 1/95

SECTION 2.5 - Electrical Test Methods

- TM 2.5.33 Measurement of Electrical Overstress from Soldering Manual Tools - 11/98
- TM 2.5.33.1 Measurement of Electrical Overstress from Soldering Manual Tools - Ground Measurements - 11/98
- TM 2.5.33.2 Measurement of Electrical Overstress from Soldering Manual Tools - Transient Measurements - 11/98
- TM 2.5.33.3 Measurement of Electrical Overstress from Soldering Manual Tools - Current Leakage Measurements - 11/98
- TM 2.5.33.4 Measurement of Electrical Overstress from Soldering Manual Tools - Shielded Enclosure - 11/98

SECTION 2.6 - Environmental Test Methods

- TM 2.6.3.3B Surface Insulation Resistance, Fluxes - 6/04
- TM 2.6.3.6 Surface Insulation Resistance - Fluxes - Telecommunications - 1/04
- TM 2.6.3.7 Surface Insulation Resistance - 3/07
- TM 2.6.4B Outgassing, Printed Boards - 5/04
- TM 2.6.15C Corrosion, Flux - 6/04

CASE STUDY

During the initial transition to Pb-free soldering, PTH electrolytic capacitors (ecaps) began failing due to stresses caused by overheating. This occurred due to rework of adjacent microprocessors. An example design is shown in Figure 11. Through-hole electrolytic capacitors have a lower electrolyte boiling point than surface-mount electrolytic capacitors. The failures resulted from poorly controlled manual soldering conditions.

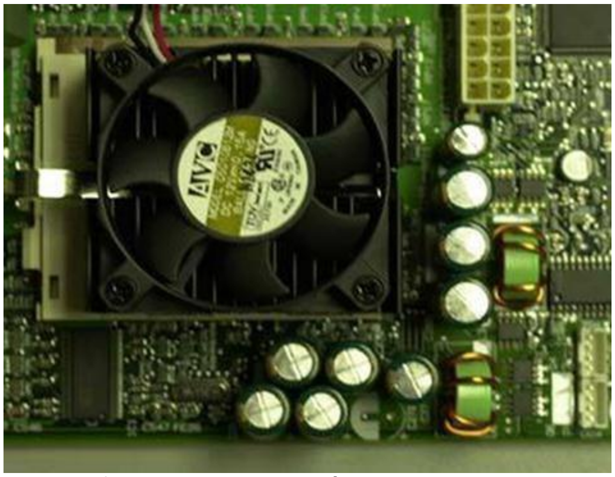

Figure 11 PTH Ecaps Close to Processor

Figure 12 shows an example of insufficient connector hole fill on an OSP finished board. Insufficent fill may violate acceptability requirements and lead to premature failure.

Figure 12 Insufficient PTH Hole Fill on Cu OSP

Figure 13 shows an example of visible excess flux. The amount of tolerable flux residue varies widely based on product, application, and end use environment.

Figure 13 Excess Flux from Manual Soldering

SUMMARY

The risks associated with manual soldering can be significantly reduced by preparing for them early in the design phase. Optimized design practices, appropriate material selection and characterization, and process monitoring techniques should be used to ensure robust results.

REFERENCES

1. Kester Solder, http://www.kester.com/kester-content/uploads/2013/06/questions-about-solderability.pdf, Questions about Solderability
2. Bath, Jasbir, http://www.leadfreeassembly.com/inemi-lead-free-rework-with-pictures-09222005_qc_jb1.pdf
3. MetCal Manual Soldering Basics, http://ecee.colorado.edu/~mcclurel/Metcal_Manual_Soldering_Basics.pdf

4. Shea, Chrys, et al., Selecting Fluxes for Lead-Free Wave Soldering, http://www.sheaengineering.com/PDFs/LF%20Flux%20Selection%20Presentation%20FINAL.pdf
5. NASA Electronic Parts and Packaging (NEPP) Program, Effect of Manual-Soldering-Induced Stresses on Ceramic Capacitors, (Part I), https://nepp.nasa.gov/files/16346/08_002_01%20GSFC%20Teverovsky.pdf
6. Hamilton, C; A Study of Copper Dissolution During Pb-Free PTH Rework Using a Thermally Massive Test Vehicle, May 2007.