THE QUICK POCKET REFERENCE FOR TIN-LEAD AND LEAD-FREE SOLDER ASSEMBLY

Solder plus Support

Manufactured in the United States of America

This work was written by employees of AIM, 25 Kenney Drive, Cranston, RI 02920 USA

Table of Contents

Introduction

This is a condensed pocket reference designed to help the production line engineer assess his/her assembly problems. The content focuses on SMT, through-hole, hand soldering, and testing. For ease of use, this guide is broken down into sections that follow the normal progression of an assembly line.

The purpose of this pocket reference is to offer a more lighthearted approach to help you solve those every day process problems. It is intended more as a memory jogger than a comprehensive troubleshooting manual.

We hope you find this a useful tool time and time again. We invite you to browse our website and urge you to contact us for any needs or questions you may have.

Written by:
Karl Seelig
Kevin Pigeon
Sean O'Brien

WWW.AIMSOLDER.COM
(401) 463-5605
INFO@AIMSOLDER.COM

Overview

This pocket guide contains general information about solder assembly. Included in this revised edition is a section related to lead-free assembly. The sections that will have the most difference as it relates to lead-free are the following:

Printing aperture design

Reflow profile

Reflow defects

QFN components

Package-on-package assembly

Wave soldering

Hand soldering

All lead-free sections are highlighted in green to speed lead-free review. If a lead-free defect is not found in a quick search, check the tin-lead section, as many defects are common to both alloy families.

SMT

Solder Paste Handling

A surprising amount of SMT defects occur before the solder paste has been opened. Many of the problems encountered while using solder paste may be attributed to the methods by which the paste is transported, received, stored and applied. By controlling these handling methods, many paste-related problems can be reduced or eliminated. Following is a list of essential factors to concentrate upon to get the most out of your solder paste.

Key Words: Heat, Moisture, Freezing, Transport, Receiving, Storage, Printer Area Conditions, Paste Preparation, Stabilizing, Mixing, Shear-Thinning, Re-Using Paste

Because solder paste is made of two ingredients with very different densities (metal and flux medium), it is normal in some formulations for a bit of flux to separate out of the paste and rise to the top of the material. Excessive **heat** can greatly exacerbate the separation of the flux medium from the paste. This will alter the rheological properties of the paste, not allowing the paste to "flow" as it is intended. Therefore, efforts should be taken to avoid the exposure of solder paste to excessive heat.

Moisture is a contaminant therefore can be detrimental to solder paste. All solder pastes are somewhat hygroscopic (having the tendency to absorb moisture), and therefore efforts should be taken to avoid the introduction of solder paste into moist (humid) environments. Water soluble pastes are more prone to cold slump. Moisture can cause and increase powder oxidation, which in turn requires more of the activator to expend itself on cleaning the solder powder and less for cleaning the components and substrate. This may result in poor or non-wetting. Moisture also causes slumping that can lead to bridging, may result in solder balls when the paste is reflowed, can result in flux/solder spatter, and can reduce tack time.

In general, the **freezing** of solder paste is not recommended. Freezing can result in precipitating the activators of solder paste out of the solvent. This can negatively alter the wetting characteristics of the paste.

In order to avoid the above environmental detriments as much as possible, solder paste should be **transported** via overnight delivery with cold packs when possible. Then, when the paste is delivered it should be removed from the **receiving** dock and stored *immediately*. Because it will generally extend the shelf life of the material, the ideal **storage** conditions for solder paste are refrigeration.

The ideal **printer area conditions** for solder paste are 40% - 50% relative humidity and 72°-80°F. In addition, no air should blow on the stencil area, as this will tend to dry-out the solder paste.

Tip:
Screen printers that run external environmental controllers by default circulate air across the stencil surface. In some cases this is intentional, in others it is merely a byproduct of design. Through simple modification this air movement across the stencil surface can be dramatically reduced or eliminated.

Proper **paste preparation** is vital to ensure optimized paste performance. It is of critical importance that solder paste *not be used or applied when cold*. Cold paste opened below the dew point of the room will cause moisture condensation on the paste surface, resulting in slump, flux and/or solder spatter, part movement, and/or other related process defects. To avoid these problems, solder paste should be warmed completely before usage. The typical **stabilizing** time for refrigerated solder paste is four to six hours. Do not remove any seal, open, or attempt to mix solder paste until it has warmed completely to room temperature. Though containers and cartridges after a period of time may feel warm to the touch, the core temperature of the solder paste may not be completely ambient. *Do not force warm solder paste*, as this can result in flux separation and the other aforementioned heat-related paste problems.

Tip:
Opening a cold jar or cartridge and mixing it vigorously will cause it to cream-up, and may make it useable, but this does not warm the paste, and is in no way a proper preparation practice.

Once the paste has warmed adequately, **mix** the solder paste lightly and thoroughly in one direction for one to three minutes by means of a spatula for jars. This will ensure an even distribution of any separated material throughout the paste. However, care should be taken not to over-mix the solder paste by stirring too vigorously or for too long. This can result in overly **shear-thinning** the solder paste, which can result in slumping and/or bridging.

Though not a recommended practice, used solder paste from the stencil may be stored in a separate container, then **re-used**. This used paste should be applied back onto the stencil typically with an equal addition of fresh paste in order to revitalize the paste. The ratio of fresh to used solder paste will vary in order to achieve a good printing consistency. It is recommended that small amounts of used paste be added to the stencil throughout the shift or entire day to minimize any degradation it may cause, and to ensure that all used paste is consumed prior to cleaning the stencil off for the last time at the close of the shift or day. It should be noted that many companies choose to discard used paste in order to avoid potential process problems.

Tip:
Do not store used and unused solder paste in the same container. This can add moisture or other contaminants to the new paste resulting in degraded performance.

Printing

Now that you've purchased an expensive automatic in-line printer, your problems have been solved, right? Not necessarily. This high dollar printer boasts high accuracy and repeatability, but due to the variables in your process it may not always print as precisely as you expect. Following is a list of essential parameters to concentrate upon when setting up your printer.

Key Words: Printer Setup, Board Support/Nesting, Board Parameters, Conveyor Width, Squeegee Speed, Squeegee Pressure, Top-Side Wipe, Scooping/Gouging, Bleed-Out, Separation Distance, Separation Speed, Snap-off/Print Gap, On-Contact Printing, Stencil Cleaning

How do you make sure that your printer will perform the best it can, something so clearly critical to a successful front end printing process? Well, let's find out. After you have done everything the printer has told you to do for **printer setup**, use some (un)common sense: look into the printer. What are you looking for? Let's break down each of the critical parameters of printer setup, and we'll show you.

One of the most overlooked parameters during setup is **board support/nesting**. Surprisingly, it takes very little pressure to cause a PCB to flex when it is only supported by its two outside edges. Proper support of the product being printed is an integral part of the setup process; without it, many subsequent adjustments may be rendered ineffective, or may have to be made in excess to compensate for poor setup.

Tip:
The purpose of board support is to eliminate board flex during the print cycle. Just remember, if you have to ask yourself whether or not there is enough support, there probably isn't.

Having the proper information for **board parameters** is just as critical as having proper board support. With regard to printer setup, board parameters are the dimensional X, Y, and thickness measurements of the PCB.

In most automated printers, the X-Axis runs left and right, (along the conveyor axis), the Y-Axis runs front to back, and thickness (or Z-Axis) is up and down. The more automated printers utilize the X and Y dimensions to mechanically center a board's position. However, of these three axes, board thickness is the most critical.

In automated printers the board thickness parameter will affect the snap-off or print gap, so it doesn't do any good to set and verify snap-off until the proper board thickness has been entered. Some automated equipment also uses the board X and Y dimensions to determine parameters such as squeegee print stroke, stencil wiper stroke, paste dispense location and bead length.

After setting board thickness, one check for accuracy is to load or step a board through the loading process, until it is raised to the inspection or vision height. At this point the top of the PCB should be flush with the topside of the conveyor rail. Any step up or down can result in squeegee or stencil damage, or at least could result in poor print quality.

Tip:
When entering the board thickness dimension into the setup menu, don't guess or assume the thickness is known. When considering this setting, verify it with a set of calipers. Ensure the topside of the PCB is level with the topside of the conveyor rail.

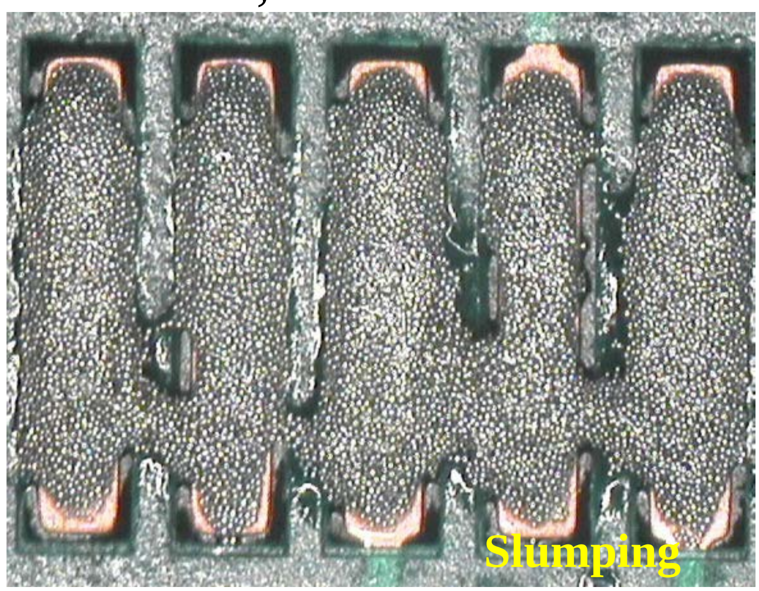
Adjust the **board conveyor width** as tightly as possible, allowing only enough play to ensure that boards won't jam when entering or exiting the printer. This seems like a trivial setting, but it can actually increase printer repeatability and reduce cycle time by minimizing the number of board to stencil alignment passes.

Squeegee Speed, AKA Print Cycle Speed, is the traverse or travel speed of the squeegee assembly during the print cycle.

Squeegee speeds can vary widely between applications, but typically range around 1 to 3 inches per second (2.54 to 7.62 centimeters), with many printers capable of 0.2 to 8 inches per second (0.508 to 20.320 centimeters).

Squeegee speed and squeegee pressure or force are directly proportional. The faster the squeegee speed is the higher the squeegee pressure or force generally must be to achieve a good clean wipe of the top side of the stencil.

As squeegee speed and pressure increase, so does the amount of heat that is generated at the squeegee/stencil interface. This promotes greater paste shear, which can result in several problems such as slumping, under stencil bleed-out, pad-to-pad bridging. High-speed printing also can cause squeegee blades and stencils to wear more quickly, resulting in incorrect solder deposition or volume, insufficient pad coverage, and drag-out. High-speed printing typically requires better board support and more frequent stencil cleaning cycles.



Unless there is no other alternative for your process, there is no benefit to printing high-speed. When setting up a high-speed printing process, the print cycle should first be adjusted to an on-demand cycle rate, where the printer is printing “just in time” for the placement equipment.

Critical parameters such as print speed and pressure and separation speed and distance should be controlled tightly to maintain print quality. Non-critical parameters can then be utilized to optimize print cycle times. This method of printer control provides for a more repeatable print process and will allow your solder paste to maintain a more stable rheology.

Tip:
A slight increase, (even 1 inch/second), in print speed can help with printability issues such as thick paste, paste sticking to squeegees and poor paste release from the stencil by shear thinning the solder paste.

If high-speed printing is necessary, remember to tightly control critical printing parameters and speed up non-critical.

Squeegee Pressure is the downward pressure, measured in PSI or KG (pounds or kilograms per square inch), exerted by the squeegee blade onto the stencil surface during the print cycle. The purpose of squeegee pressure is to provide the force necessary to push the solder paste across the width of the printable area in a controlled roll, filling in all stencil apertures while providing a clean **top-side wipe** of the stencil surface. A typical starting point for squeegee pressure is .7 to 1.5 lbs

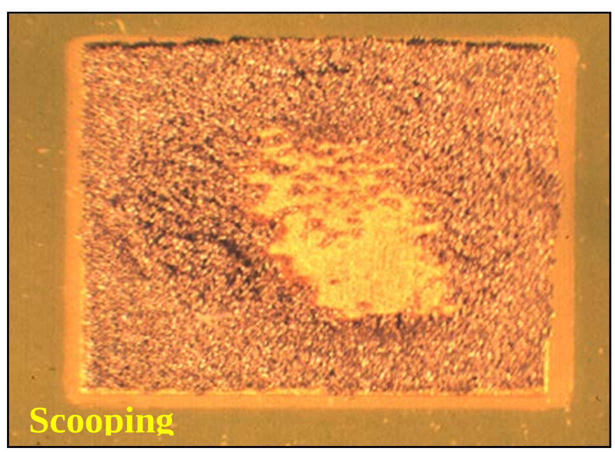
(.3175 to .6804 kg) of pressure per linear inch (2.54 cm) of


Unsatisfactory Top-Side Wipe

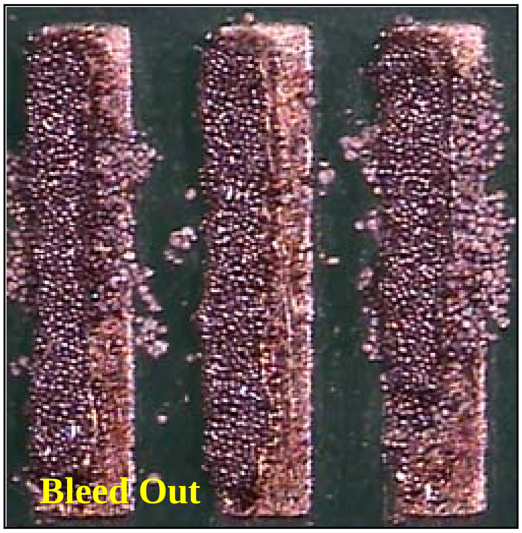
printable area (not blade length). Using the printable area vs. blade length to establish initial squeegee pressure settings will help to ensure that the initial squeegee pressure is not too excessive.

Ideally, the squeegee blades should be the length of the printable area plus ½ - ¾ inch (1.27 – 1.90 cm) overhang on each side. Longer squeegee blades may be used, but any excessive overhang should be supported from the bottom side of the stencil. Longer blades also allow the solder paste to spread to the unusable print areas of the stencil. Make sure you scrape all unused paste into the usable area often.

Excess squeegee pressure can result in **scooping/gouging** (especially with polymer blades) and paste **bleed-out**. Bleed-out can be detected by evidence of paste being packed in to the edge of the solder mask. This is generally a row


Scooping

of solder spheres from the powder that is packed against the edge of the mask and the board. Another indicator of too high of a squeegee pressure is flattened particles on the edge of the print. To correct this, reduce the squeegee pressure until there is a light smear of paste remaining on top of the stencil. Then, increase the pressure until you get a clean topside wipe of the stencil surface. For other related defects refer to the *squeegee speed section*.



Tip:
Adjust squeegee pressure just high enough to achieve a good clean topside wipe of the stencil surface. Leaving paste behind on the stencil surface can promote poor aperture release, torn prints, insufficient solder coverage and pre-mature paste dry-out.

Excessive squeegee length can accelerate paste dry out. Make sure all paste is scraped from unused areas of the stencil and folded into usable areas on a frequent basis.

Light bodied pastes with low viscosity require less pressure than heavier solder pastes; one pound of pressure per linear inch of squeegee blade generally is sufficient. Heavier pastes with high viscosity normally require 1.5 pounds of pressure per linear inch of squeegee blade.

Print speed and squeegee pressure are directly proportional, so by lowering print speed, you will be able to reduce overall squeegee pressure, thus eliminating pressure related problems.

Separation Distance is an adjustable distance to which the PCB and stencil separate at a pre-specified controlled rate, referred to as *separation speed.*

At the completion of the print stroke, and after any squeegee up or separation delay, this separation sequence begins. Working in conjunction with separation speed, this controlled parting continues until the separation distance set point has been reached, at which time the speed of PCB and stencil separation increases to its maximum.

This function allows for a controlled clearance of the solder paste from the stencil apertures, providing for more uniform and repeatable paste deposits.

Separation distance should be set great enough to allow all deposited paste to clear the stencil apertures prior to the increase in separation speed, which occurs when the separation distance set point has been reached.

Separation distance is indicated in either thousands of an inch or millimeters. A recommend starting point for separation distance is 0.100 inches to ensure total paste separation from stencil apertures; this may be reduced to a lesser distance once production has begun to satisfy cycle time requirements. If separation distance is reduced too much, however, print definition and quality will suffer.

One should ensure that the separation distance is increased by a sufficient amount to compensate for any negative snap-off or Z height resulting from board warpage or other reasons. Any future additions of negative snap-off should be added to separation distance, ensuring enough clearance distance for paste and apertures.

Tip:
When utilizing a controlled separation distance, larger distances equal longer cycle times. There is no real benefit to increasing the separation distance to greater than that which is sufficient to allow paste and apertures to clear.

When setting up a new or unfamiliar product, you should set the separation distance to maximum, and then refine the setting once production has begun.

Separation Speed is an adjustable velocity that, in conjunction with *separation distance,* works to control post-print separation of PCB and stencil.

At the completion of the print stroke and after any squeegee up or separation delay, the separation of PCB and stencil begin at a controlled speed defined by separation speed. This controlled speed and separation continues until the pre-set *separation distance* has been reached, at which time separation speed increases to its maximum.

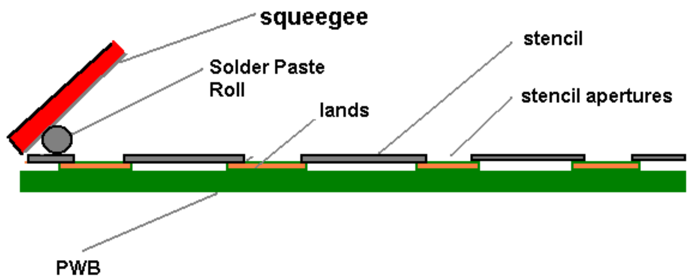
As with separation distance, this parameter is designed to aid the separation of solder paste from the stencil apertures. Separation speed can be indicated in thousands of an inch, millimeters, or as a percentage of axis speed.

In general, the slower the separation speed the more reliable and repeatable the paste deposition. Too fast of a separation speed can result in dog-ears/peaking/wicking, clogged stencil apertures and poor paste coverage. The *Printing Defect Analysis* section provides a detailed discussion of these defects.

A recommend set point for separation speed is typically .010”-.020” per second (.254mm-.508mm per second), 1-2, or 10-20% of axis speed for fine pitch and micro-BGA. For non-critical printing .030”-.050” per second (.762mm-1.27mm per second), 3-5, or 30-50% may be used. Though it may be possible in many cases to run with maximum separation speed, unless cycle time or other constraints dictate, some control is better than none.

Tip:
When printing fine pitch and micro-BGA, set separation speed to a minimum.

Solder Paste On Contact Stencil Layout



Snap-Off/Print-Gap is the programmable distance between the topside of the PCB and the bottom side of the stencil. Snap-off can be used to aid the release of solder paste from the stencil apertures. It can also be utilized to increase paste volume on an assembly.

When some lower viscosity solder pastes are used in conjunction with a print gap, bridging may result by way of paste bleed-out on the bottom side of the stencil. Other pastes may show signs of poor uniformity in paste deposition, or other inconsistent printing results.

If your equipment is calibrated correctly, board thickness is properly set, and snap-off is set to zero, this should result in **on-contact printing**. The PCB and stencil, (during the print cycle), should only contact each other lightly when they meet. No upward deflection of the stencil surface by the board (known as negative-Z) should be noticed. If executed correctly, this setting allows for a full "gasketing" of the stencil to the component pads during the print cycle, thus preventing bridging related to paste bleed-out, even on UFP components.

On-contact printing also provides for a more uniform paste deposition and more consistent paste height.

Tip:
When testing for on-contact printing, make sure the board vacuum is turned off prior to checking for any snap-off or print gap.

Though there are distinct advantages to using snap-off, one of the more characteristic problems related to this is the irregularity of deposition quality.

Stencil cleaning cycles will depend greatly on factors related to stencil cut and aperture quality, printer alignment accuracy and repeatability, board surface finish quality, print pressure, squeegee type, paste viscosity, and even environmental conditions.

Stencils should be cleaned frequently enough to ensure total removal of any bottom side residues, but not so infrequently as to allow the same residues to dry or cake on, making their removal much more difficult.

Some applications may require cleaning after every board printed, while others may be able to clean the stencil a surprising one time per shift, or even not at all. Regardless of how often you do clean your stencil, one sure method of determining whether or not the job is getting done is to remove the stencil and visually inspect the bottom side.

As far as stencil cleaning solutions go, it is recommended that you first attempt to perform dry wipes when using an automated under stencil wipe. If a dry wipe doesn't do the job, then an appropriate stencil cleaning solution is recommended.

A good cleaning solution should facilitate the removal of under stencil residues without damaging the solder paste it contacts. In a best-case scenario, the chosen cleaning solution should enhance your paste by reducing dry-out and promoting better release from stencil apertures.

When cleaning the stencil by hand, cleaning solutions should be applied to a lint-free cloth, not directly to the stencil surface. This should eliminate any splatter of the solution onto the solder paste already on the stencil.

Use all cleaning solutions in moderation, and remove any excess cleaner. Many solutions leave behind an oily residue that can inhibit the solder paste from rolling during the print cycle. Solutions also can mix into the solder paste, causing a breakdown of the pastes chemistry, leading to extremes such as dry-out or slump.

Tip:
To establish a baseline for bottom side stencil cleaning, first verify proper printer setup, then starting with a clean stencil, begin printing. Count the number of boards printed while inspecting solder paste definition quality, usually paste deposits tend to start getting a little “fuzzy” around the edges when the stencil begins to get loaded up. Take note of your board count, clean the stencil and resume printing, this time de-rate the cleaning count by 2 to 5 boards. At the end of the new count, inspect the last board printed. If print quality still looks good, use this count as a starting point for stencil cleaning.

Though IPA (Isopropyl Alcohol) often works well as a stencil cleaning solution, it is typically not compatible with most flux chemistries and can significantly reduce a solder pastes useable life by causing dry-out.

If a little is good, alot is not necessarily better. Any foreign substance entering a paste’s chemistry is considered a contaminant and can damage the paste. Only materials that are 100% compatible should be used.

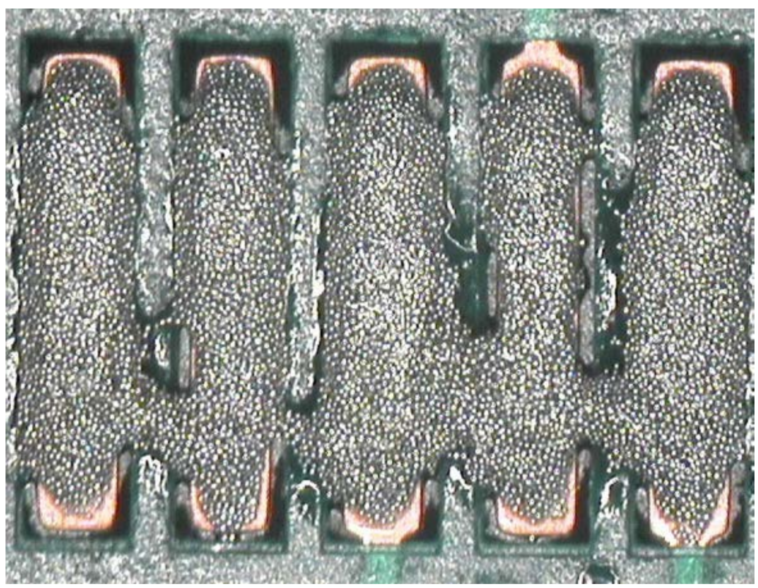
Printing Defect Analysis

We’ve already provided the causes and cures for some printing problems such as scooping and bleed-out above. Here are some more.

Key Word: Bridging, Torn Prints, Over-Driving the Board, Clogged Stencils, Dog Ears/ Peaking

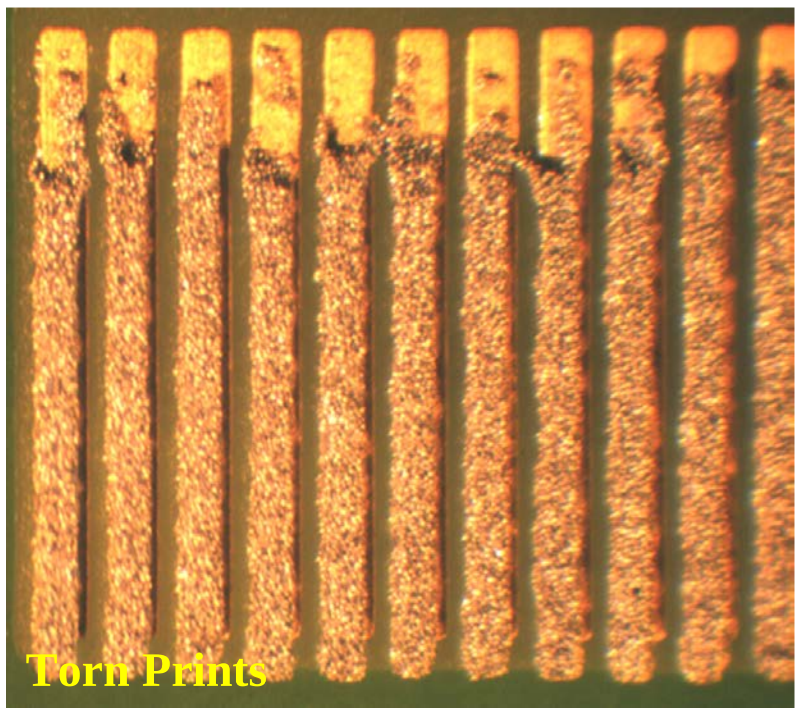
First, determine if the **bridging** is occurring in the printer and not in another place further down the line, i.e. placement or reflow. If the board comes out of the printer OK, but bridges down the line, it could be a placement issue. If it goes into the oven OK, but comes out bridged, then it is probably "hot slump". This is another paste-related problem, and is covered later in the *Reflow* section.

More often than not, bridging is related to the solder paste quantity or rheology. Either too much paste has been printed or the viscosity of the paste is too low. If either of these conditions exist, when the components are placed the paste may squeeze out from under the sides of the leads. Another possible culprit is squeegee pressure. If squeegee pressure is too high, the paste can be shear-thinned so that it loses body to the point of slumping, which could lead to bridging.



If you have determined that the bridging problem is occurring in the printer, and you have confirmed that you are not over-driving the board, then go back and check the snap-off parameter again. Chances are that someone has adjusted something. Reset parameters and move on to the next step.

If bridging is always in the same localized area, check for proper board support. This is very common in panelized and double-sided assemblies. Remember, solder paste is not intelligent enough to drive you crazy by being inconsistent in one spot on the board- it has to have help!



A very common printer setup problem is the snap-off setting. If you are experiencing **torn prints** you should go back and confirm this setting. Although you may already have set this parameter on the setup screen, you may not have checked to see if you are getting actual on-contact printing. Refer to the *procedure on verifying on-contact printing* on our web site at **<u>www.aimsolder.com</u>**.

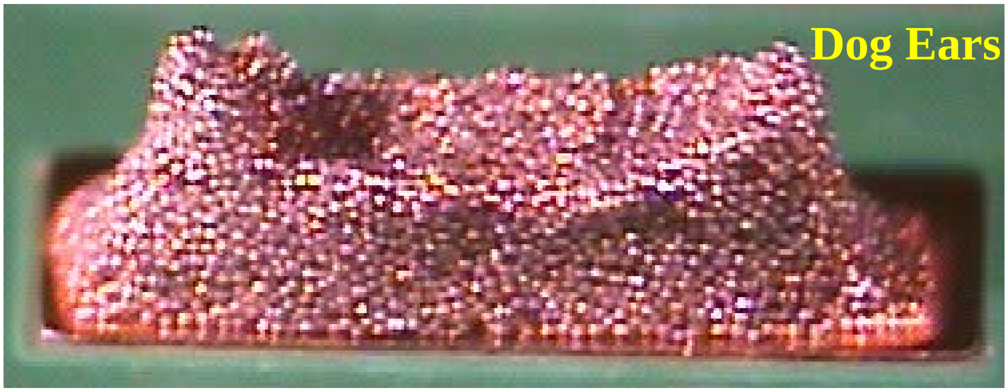
When a board is **over-driven** into the stencil and the printer goes to make the separation, paste may be torn from the pad as it pulls away. This occurs because the stencil has followed the board since it was pushed upwards with the board, and as your very precise printer separates, it is assuming that the board was making only marginal contact. It did not know that the board had been forced by as much as 100 mils into the stencil.

If you over-drive the board, you also can end up with inconsistent prints on multi-up panels- one panel will print fine, but the others may bridge or have opens. If the printer is not properly aligned, the board often will touch in one area and not in another. If this occurs as a result of a warped board, do not try to compensate for it by over-driving the board- this only will cause additional grief.

To set the snapoff parameter correctly and remedy the torn prints, make sure that the board just barely touches the bottom of the stencil. You should also check: Are your support pins clean (no globs of paste stuck under them)? Does your edge clamp buckle the board? Does your board sit flat every time? If the answer to any of these questions is “no”, all you are doing is creating more work for your repair people.

A common problem related to an incorrect separation distance is **clogged stencils**. This is a result of a trampoline effect of the stencil where the solder paste remains in the stencil apertures after being torn from the board pads.

Another related defect is **dog-ears or peaking**, where the ends or corners of a paste deposit



protrude higher than the remaining body of paste. Review the printer setup section on *separation distance and speed* to correct this.

Not all solder paste is created equal. Some lead-free solder pastes have a tendency to stick to the squeegee rather than dropping off to the stencil. If this happens, after several prints there will be insufficient paste being deposited. If this situation does occur, more management of the print process is required. This will require removing the paste build up on the squeegee with a spatula to maintain sufficient quantities of paste on the stencil, ensuring proper aperture fill.

It is also important to understand that depending on lead-free alloy, pad finish and paste chemistry, lead-free solder paste may not flow to the edge of the pad. If this defect occurs, an aperture-to-pad ratio modification of 1:1 may be required dependant upon paste chemistry.

Component Placement

You thought your printer was expensive? Here comes the placement equipment salesperson to solve all of your placement issues. Before you get too comfortable, though, beware that some SMT defects may be traced back directly to the pick & place machine.

Key Words: Parts Moving or Being Thrown Off, Board Bounce, Board Support, Placement Pressure, Billboarding

People who sell placement equipment always like to show you how fast their equipment can place parts. What they don't tell you is that very high-speed placement equipment typically causes a great deal of abrupt movement of the board, which can result in **parts being moved or even thrown off of the board**. This may be due to the acceleration/deceleration of the board and/or board bounce.

Since the rate of acceleration/deceleration is related to speed, the only real method of correcting these would be to de-rate (slow down) the equipment placement rate.

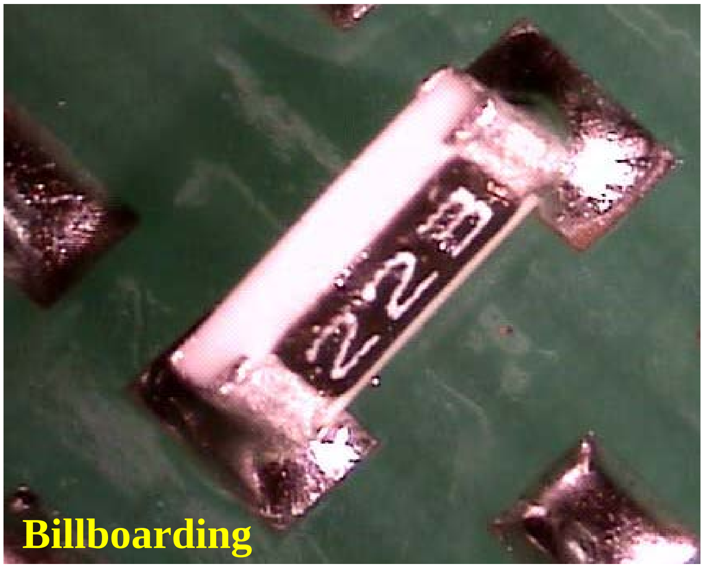
The more common cause of part movement is **board bounce**. Board bounce can be minimized with support. Support the board with alot of pins or go the top shelf method of tooling fixtures.

Very often, solder pastes are accused of being too low in tack. “Your paste doesn’t hold my parts in place” is a common accusation. Sometimes an increase in paste viscosity may cause this defect to disappear but did you really fix the problem? No, that’s what we refer to as a band-aid. Do your homework first: look at all of the variables.

Just as in the printer setup, it is vital to have adequate **board support** in the placement equipment. Variables such as thin or more flexible circuit boards and high placement pressures allow for a flexing of the PCB during placement. The accelerated lift of the placement nozzle allows the now flexed PCB to snap back into position, resulting in a launched or otherwise disturbed placement. This can be seen in many device types, but is most commonly visible in larger devices such as tantalum capacitors, due to the inertia of the component mass.

It is also important to check the **placement pressure**. There’s no reason to mash components into paste. If the pressure seems fine, check that you are using the proper thickness component, or that the proper thickness dimension is programmed into the placement equipment. If the machine is told that the part is thinner than it is, this will result in a mashed, then bouncing, component.

Unlike tombstoning, **billboarding** is often traceable back to the placement process. Billboarding usually relates to discrete components, resistors and capacitors. Unlike tombstoning where a discrete has one termination soldered with the other side standing on end, billboarding is indicated by both terminations being soldered, but with the component up on edge.



Billboarding

If you are experiencing billboarding problems, some things to check for are the pick position of the component from the feeder, the feeder advancement speed, the type of feeder tape, static interference, and the cavity tolerance or slope of the feeder tape.

Reflow (Sn63/Pb37 and Sn62/Pb36/Ag2 Alloys)

Now that the board has been printed, gone through placement, all of the parts are there, and the paste is not squeezed out from under the leads to the point of bridging, you are ready to send it through reflow. This is the magic tunnel where you cannot see what is happening to your assembly. So, take a slow walk down to the end and pick up a board. This is where all the sins of not being very precise will come back to haunt you.

Key Words: Profiling, IR, Convection, Ramp-Soak-Spike (RSS), ∆T, Ramp-to-Spike (RTS)

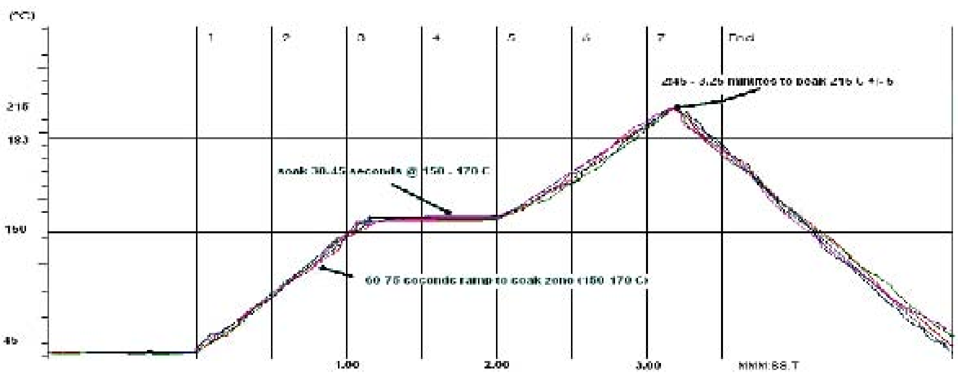
To profile or not to profile? Unfortunately, you don't have a choice. Yes, it's time consuming, tedious, and boring. However, there is no better means to observe what is happening to an assembly during reflow than by attaching thermocouples to it and spending the time and resources to produce an optimized **profiling** process. If the reflow process is developed incorrectly, or is out of control, any range of defects, including premature field failures, can occur. As you will see below, in general, solder pastes like fast profiles that spike to reflow within 3 ½ minutes. However, the exact profile to use will depend upon many factors such as oven limitations, board layout, component types, etc.

Profiling must be performed with a populated assembly; otherwise you will not be able to gauge the actual heat absorption of the product. Furthermore, profiling must be performed on the product that you are trying to optimize. In other words, a profile that works optimally for one assembly may not work at all for another. This mandates that in order to achieve optimum profile results on all products, you most likely will need to profile each of these separately (especially if the assemblies vary in size, density, and components).

With recent industry-wide changes and improvements made to reflow ovens, the requirements and parameters of solder paste reflow profiling also have changed. Older-style (**IR**) ovens often have required the ramp-soak-spike (RSS) profile in order to achieve optimum reflow results. Contrarily, many of the newer, more efficient ovens that transfer heat through forced **convection** are able to reflow solder paste equally well or better by following a ramp-to-spike (RTS) profile.

The main function of the soak zone of the **Ramp-Soak-Spike (RSS)** profile is to bring the temperature of the entire assembly to equilibrium before reaching the reflow temperature of the solder. Because **ΔT** (the greatest difference of temperature found across an assembly) and the problems associated with it often are not encountered on an assembly that has been reflowed in a more efficient oven, the soak zone may be rendered unnecessary (although this is product density related). Because the soak zone is unneeded, the profile may be altered into a linear (RTS) profile.

The RSS profile was most commonly associated with RMA and no-clean chemistries due to oven capabilities at the time of their conception. The RSS generally is not recommended for use with water soluble chemistries, as the long soak zone of the RSS profile may break down the activators of the paste prematurely.



As seen above, the RSS profile begins with a steep ramp up to approximately 150°C within a target time of 60 seconds. The ramp rate should be controlled at 2-3°C/second maximum in order to prevent solder spatter and thermal shock to the components.

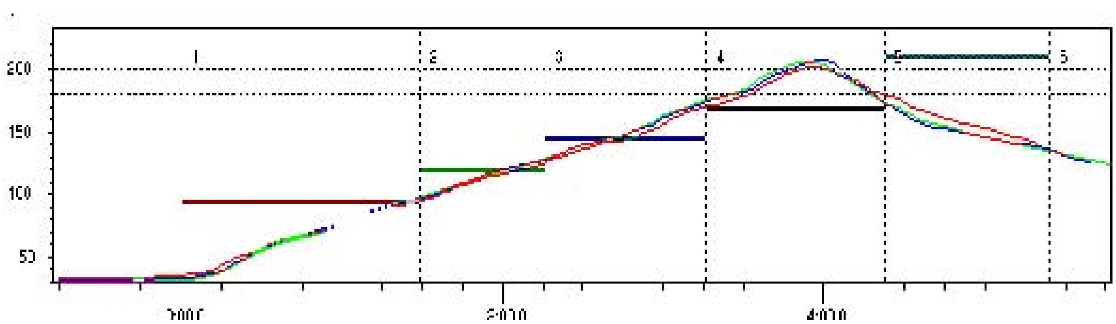
Following the ramp area, the profile soaks the PCB assembly between 150-170°C, again within a target time of 60 seconds. 170°C should not be exceeded in the soak zone, as this is the point where the activation system in many solder pastes begins to break down exponentially.

After the soak, the assembly will enter the spike zone, where the assembly will be reflowed above 183°C for a target time of 45 seconds ± 15 seconds. The target time above liquidus (183°C for Sn63/Pb37) is 45 seconds; this is ample time to eliminate flux entrapment and voiding and will increase the pull strength of the solder joint.

The entire profile should last between 3 to 3.5 minutes from 45°C to a peak temperature of 215°C ± 5°C. The cool down rate of the profile should be equal to or less than 4°C per second in order to prevent thermal shock damage to components. In general, the faster the cool down rate, the finer the grain structure and the stronger the solder joint will become.

The **Ramp-to-Spike (RTS)** profile may be used with any chemistry or alloy (including lead-frees), and is preferred for use with water soluble solder pastes. If for any reason a large ΔT exists on the assembly, such as with processes utilizing fixturization, or if your equipment cannot achieve proper ramp rates and peak temperatures, the RTS may not be the appropriate choice of profile.

The RTS profile has several advantages over the RSS profile. Because the ramp rate in a RTS profile is so much lower (0.7°-1.8°C per second), there is much less concern of defects related to thermal shock to the board and components resulting from too high a ramp rate.



As seen above, the RTS profile is simply a gradual linear ramp from ambient to peak temperature. In general, setting up the

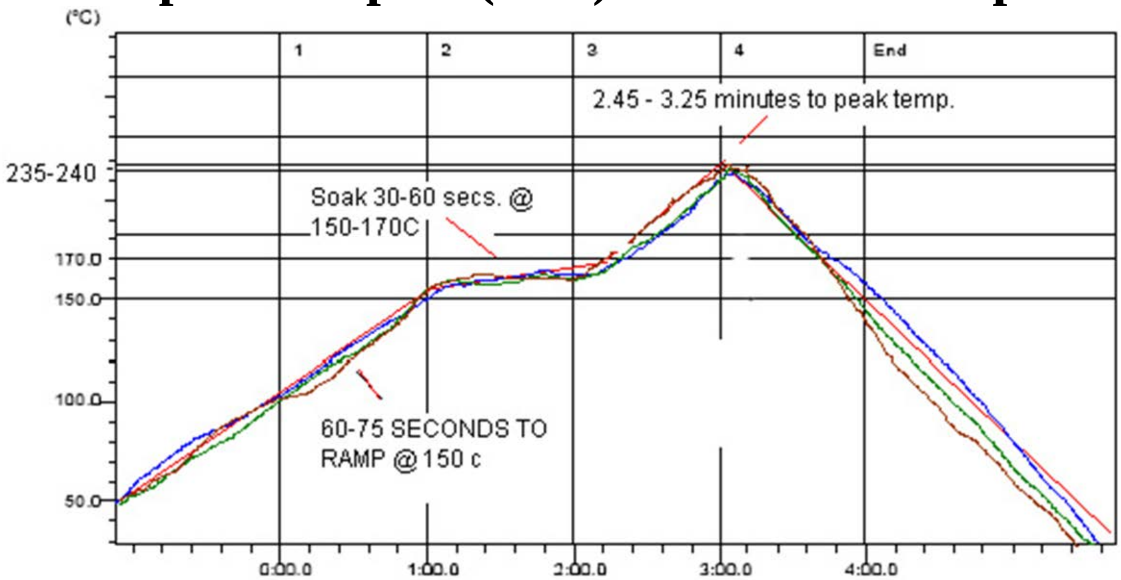
RTS profile is quite simple. The ramp zone serves as the preheat zone for the assembly, wherein thermal shock to the assembly is prevented, the flux is activated, the volatiles are driven off, and the assembly is prepared for reflow. The typical ramp rate for a RTS profile is 0.7-1.8°C per second, with the first 90 to 120 seconds of the profile made as linear as possible.

As with the RSS profile, the RTS profile length should be a maximum of 3 – 3.5 minutes from ambient to a peak temperature of 215°C ± 5°C. Again, the peak temperature should be controlled at 215°C ± 5°C, with time above liquidus at 45 seconds ± 15 seconds and a cool down rate within 4°C per second.

Reflow (Lead-Free Alloys)

Reflow is the SMT process area that will be most affected by a switch to lead-free processing. Most lead-free alloys require higher reflow temperatures than the 210-220°C peak temperature of tin/lead; 235-245°C is common with tin/silver/copper alloys. This higher reflow temperature dictates that one should minimize ΔT and maximize wetting through the reflow profile (including cooling), and could possibly mandate reflow equipment changes.

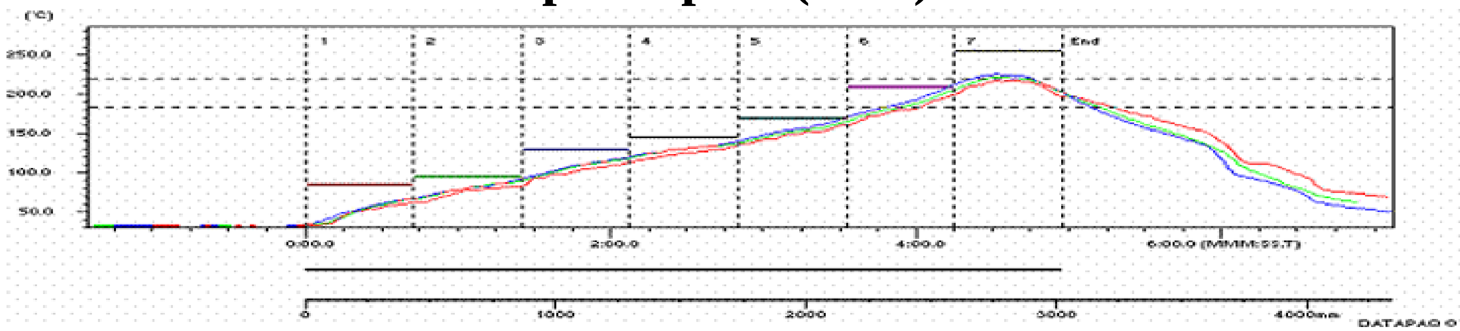
Ramp-Soak-Spike (RSS): Recommended profile



Ramp-Soak-Spike Profile Guidelines

- The typical initial rate of rise for the RSS profile is 1.7 to 2.1°C/second.
- Ramp up to 150°C and then soak the assembly for 30 to 60 seconds.
- The soak zone should be controlled between 150-170°C. Above this point the paste will lose its activator.
- Proceed to spike immediately once the PCB has reached thermal stability.
- Peak temperature is 240ºC ± 5ºC.
- Time above liquidus is 45 ± 15 seconds.
- The total profile length should be between 2 ¾ - 3 ½ minutes from ambient to peak temperature.
- Cool down should be controlled within 4°C/second.

Ramp-to-Spike (RTS)



Ramp-To-Spike Profile Guidelines

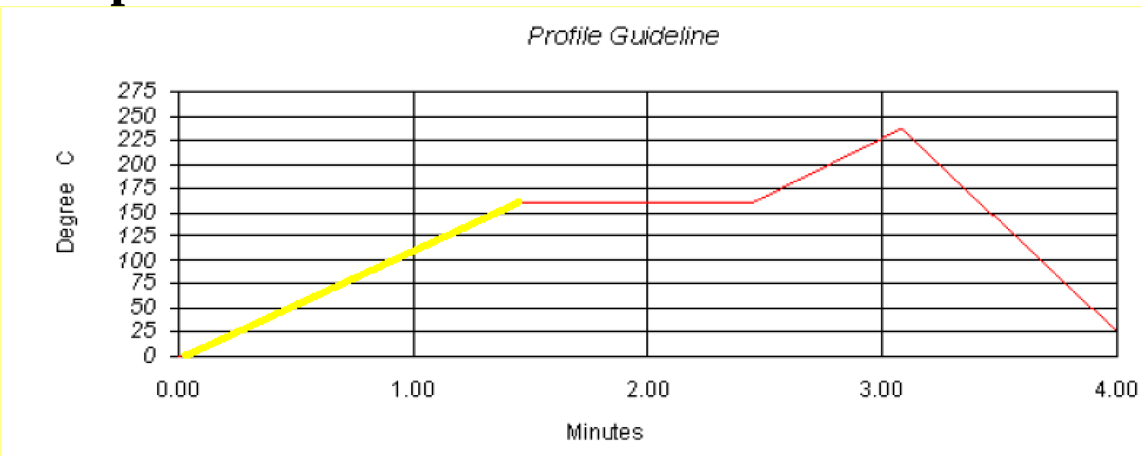
- The typical rate of rise for the RTS profile is 0.8 to 0.9°C/second.
- The profile should be a straight line or concave; it should not be convex.
- 2/3 of the profile should be below 150°C.
- Peak temperature is 240ºC ± 5ºC.
- Time above liquidus is 60 ± 15 seconds.
- The total profile length should be between 3 ½ - 4 minutes from ambient to peak temperature.
- Cool down should be controlled within 4°C/second.

Tip:

For more information on lead-free soldering, go to **<u>www.leadfree.com</u>**.

Understanding the Specific Functions of the Profile

Ramp Rate



Ramp is the first section of a reflow profile. This allows the components to heat up without thermally shocking them. The solder paste begins to dry due to solvent evaporation. The activation system is just starting to clean the powder surfaces and the pads it is in contact with.

Troubleshooting: Preheat rate is too high

- Flux/solder paste spatter (rapidly boiling)
- Premature paste activation
- Depletion of activators
- Popcorning
- Poor wetting, can't solder successfully without flux
- Affects residues (amber)
- Affects temp. peak (no activation/carrier left)
- Dull, grainy joints

Troubleshooting: Preheat rate is too low

- Flux spreads everywhere carrying fines with it which cause solder balls
- Solder beads – Paste squeeze
  - Solder paste displaced from component pads during printing or placement collects under the center of the components during the reflow process. Once the solder has reached liquidus, surface tension pulls the component down to the pads, causing the molten solder to squeeze out from under the component.

Soak Zone

Next is the soak zone. The soak is to stabilize the board prior to the ramp to liquidus. Its function is to reduce the ΔT and stabilize the board. (An RTS profile does not have a specific soak zone. However, due to a slower ramp rate, it has thermal stability throughout.)

Troubleshooting: Poor control of the soak zone

- Dull grainy solder joints - Time at temperature
- Diminished flux carrier
- Low quantity flux residue from depleting carrier
- Voiding - Too high a soak can also lead to voiding and flux entrapment
- Tombstoning - Occurs in the later part of the soak zone N2 or air. Transition from solidus to liquidus does not occur simultaneously (ΔT). One termination turns liquidus prior to the other allowing surface tension to stand the opposite side of the component up, out of the solder deposit.

Reflow

Reflow is where everything happens if the first two stages of the profile are correct. The reflow section is where the activation of the flux is completed and the alloy reaches the liquidus stage.

Troubleshooting: Excessive time above liquidus

- Dull grainy solder joints
- Poor or de-wetting
- Malleable / soft solder joint (large grain structures)
- Low flux quantity
- Poor flux quality

Troubleshooting: Insufficient time above liquidus

- Dull, grainy, un-reflowed solder
- Non-wetting - poor solder flow
- Typically good flux quality
- Higher than normal flux quantity
- Brittle solder joints due to poor intermetallic formation
- Flux entrapment - voiding

Troubleshooting: Excessive peak temperature

- Dull grainy solder joints
- De-wetting
- Poor flux quality, (amber or burnt flux residues)
- Low flux quantity
- Large ΔT between components and substrate
- Component damage

Troubleshooting: Low peak temperature

- Dull grainy solder joints – insufficient solder reflow
- Poor or non-wetting
- Typically good flux quality
- Higher than normal flux quantity
- Flux entrapment / voiding
- Brittle solder joints due to inadequate intermetallic formation

Troubleshooting: Too fast of a cooling rate

- Thermal shock / component damage
- Crackling of residue material itself
- Tombstoning – N^2 processes typically
- Finer grain structures

Troubleshooting: Too slow of a cooling rate

- Larger grain structure
- Dull grainy solder joints
- Weaker, more malleable solder joint

Overall Profile Length

Troubleshooting: Too long of a profile

- Poor solderability
- Poor or non-wetting
- Dull grainy solder joints
- Poor flux quality – amber or burnt residues

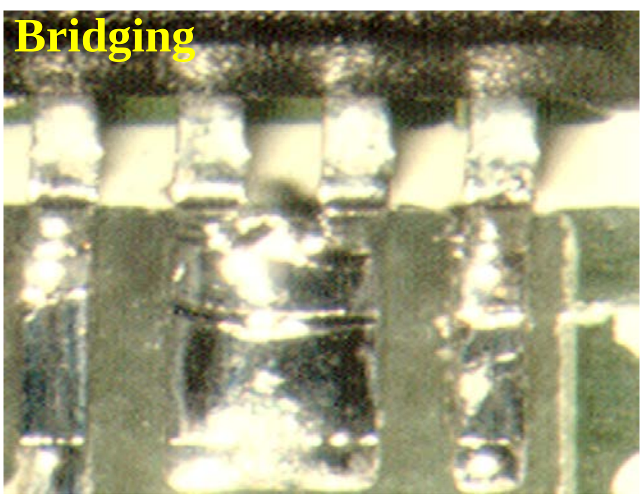
Troubleshooting: Too Short of a profile

- Poor or no solderability
- Poor or non-wetting
- Brittle solder joints
- Dull grainy solder joints
- Excessive flux quantity

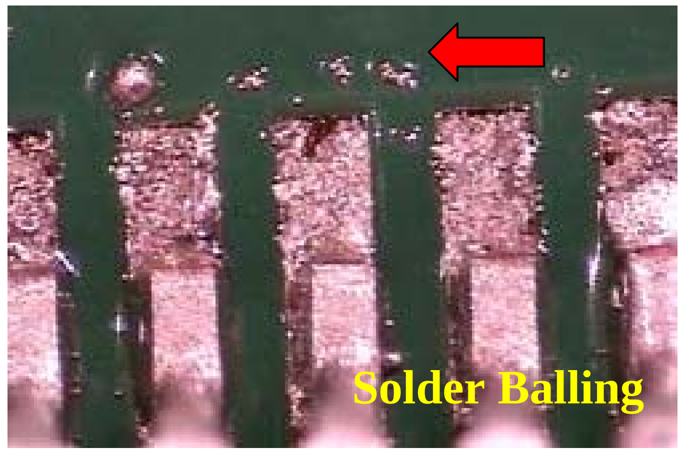
Reflow Defect Analysis

So, you've profiled your assemblies and you're certain that everything will come out perfectly. But what if it doesn't?

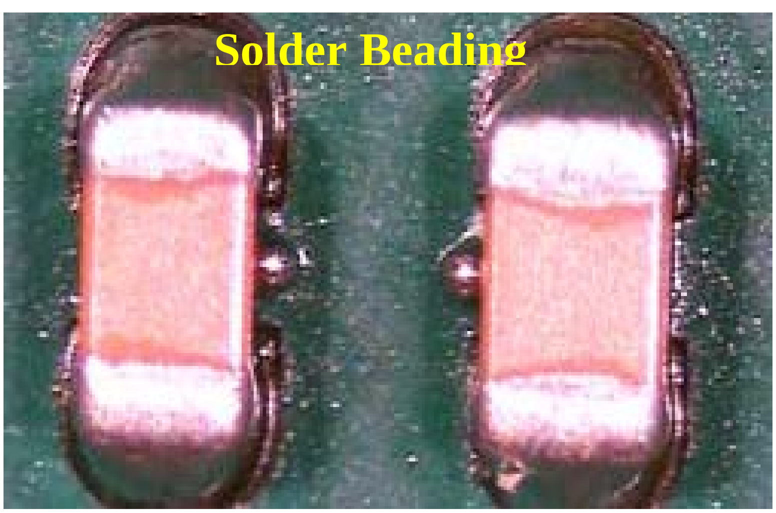
Key Words: Bridging, Hot Slump, Solder Balls, Solder Beading, Home Plate Design, Tombstoning, Popcorning, Component Warpage, Coplanarity Issues, QFN, Shorts, Voids, Opens, Deficiencies, Component Floating, Aperture Reduction, Placement Pressure, Consistent Depth, Profile



Bridging that was not there when the board went in the oven may appear as the board leaves the oven- this is **hot slump**. This normally is paste-related, but sometimes can be profiled out (especially with no-cleans and some rosin based pastes): increasing the ramp rate to 2.5-3°C per second to a 150°C soak may help. Remember, though, that the longer you soak the paste the less activity you will have when it reaches reflow. Water-soluble pastes are another story: if you try to soak these, most of the time you will kill the activity of the paste (unless there are halides in it).

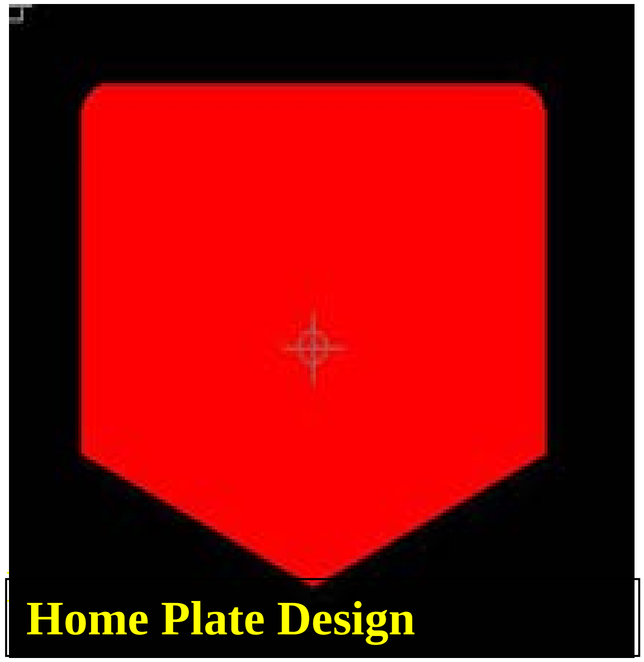


Although sometimes related to oxidized paste or paste that has sat on the PCB too long before reflow, **solder balls** are very often reflow profile related. In general, this can be attributed to too slow or too fast of a reflow ramp rate. Heating up an assembly too quickly will not allow the volatiles in the paste to be driven off before paste becomes molten. A combination of volatiles and molten solder will result in solder spatter (balls) and flux spatter.

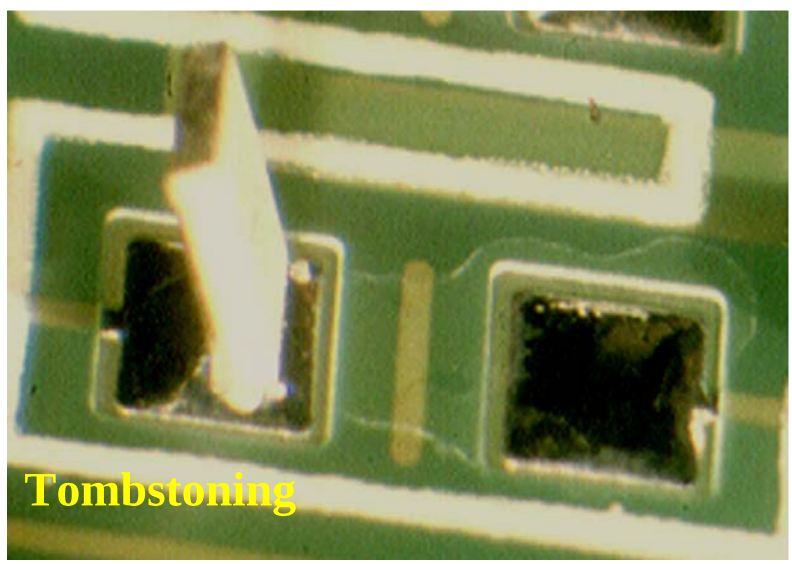


Solder beading can be a result of poor profiling, but in general is related to paste volume and paste viscosity. It is recommended to use a reduced stencil aperture to pad ratio when printing.

One of the more common stencil aperture reductions used to minimize paste volume is the **home plate design**. This reduces the amount of paste printed and helps to keep the paste from spreading off the pads, where it may ball up and become a bead. Remember that it's not only the reduction that matters- the location of the paste on the pad is also important.

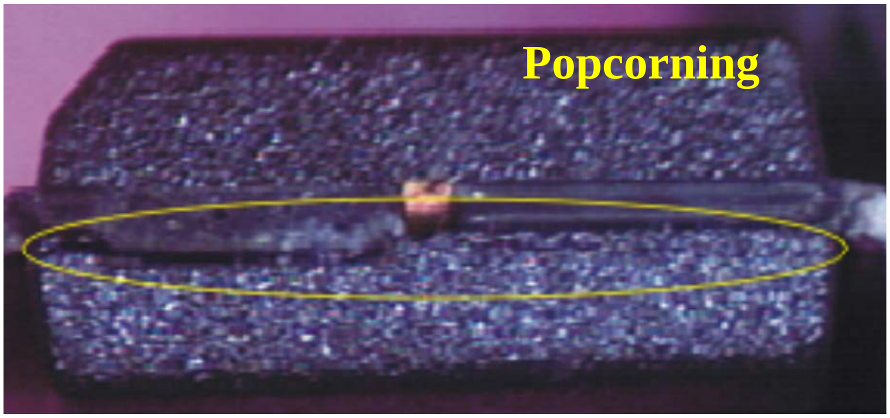


Another frequently-encountered defect is **tombstoning,** which is typically a wetting related defect. This is more commonly seen during vapor phase reflow, where one pad of a component reaches liquidus before the other, and the surface tension of the solder pulls the component up on end. In today's technology, large ΔT's in your reflow profile or unsolderable PCB pad or component surfaces can readily cause tombstoning. If you are processing with nitrogen, tombstoning also can be caused by your cooling rate.



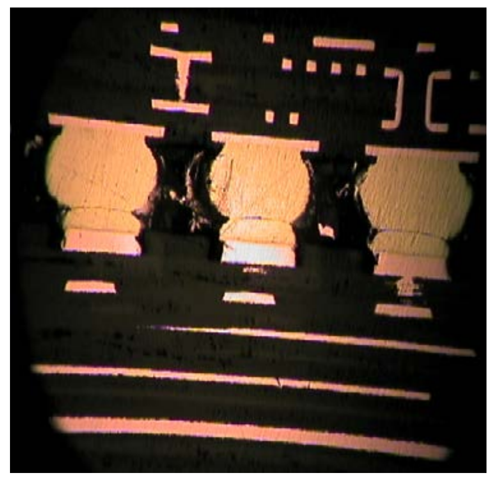
The way to remedy tombstoning is to go through the solid-liquid phase of the solder slower. This is the area of the profile where you just begin to transition into reflow. If you are using the Sn63/Pb37 alloy, this phase occurs around the 183°C melting point. As you transition into or out of the liquidus area, ensure that your ΔT's are at a minimum. Also keep in mind that, in general, the more density difference you have due to ground planes, the more likely it is to experience these defects, and the more attention is required at the transition into liquidus. Furthermore, the quantity of paste deposit can affect tombstoning: in general, the thicker the paste deposit, the more tombstones will be experienced. This can be eliminated by using different powder loading or modified alloy to slow wetting and eliminate the part from standing up.

Other factors that can relate to tombstoning are component and/or paste mis-registration. Ensuring that paste and components are deposited where they should be can be very helpful in minimizing tombstoning

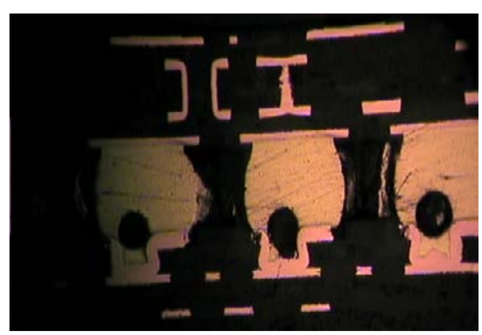


Popcorning is evident by ICs splitting during reflow. Popcorning is directly related to ramping through 100°C too fast and often is related to the quality and storage conditions of the components. The rule of thumb is that the more moisture that components have absorbed, the slower the ramp rate through 100°C should be. Improving the quality of your components and your storage methods, or pre-baking your devices would also be very helpful, of course.

Head-in-pillow has proven to be a significant issue related to lead-free solder paste. Depending on the manufacturer of the BGA and paste chemistry, head-in-pillow (pictured right) can occur. This defect may occur due to **component warpage**, either as received (sometimes known as **coplanarity issues**) or more likely from warpage during reflow. If the component is out of coplanarity by .008" and the print is .005", head-in-pillow will occur. This is a costly defect since many of these will pass inspection only to later show up as field failures. Head-in-pillow can be solved by a paste chemistry change, coupled with profile changes, or in the case of coplanarity issues, a thicker paste deposition is needed.



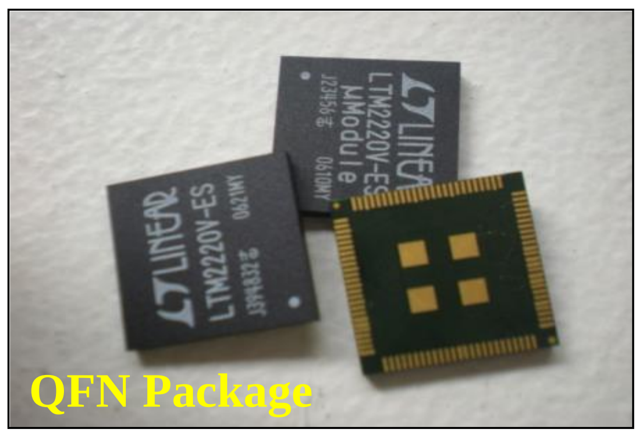
Another BGA-related defect that becomes more apparent with the use of lead-free alloys is voiding in via-in-pad (pictured right). This defect is also related to chemistry and board design.



Whenever board design can be controlled to stop this via-in-pad, the manufacturability of the product greatly improves.

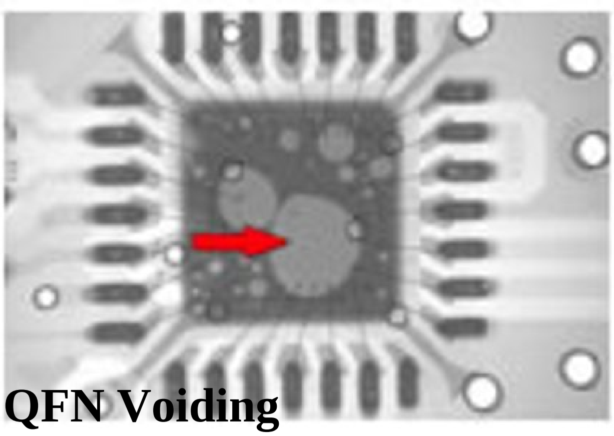
General BGA voiding can be moderately reduced by profiling if the process is skewed. However, if the reflow profile is in the proper working range, minimal improvements can be achieved by profiling. The most critical part of profiling is time at soak. Sometimes, a 120 second soak will reduce voiding. Board finish, as well as paste chemistry will have the largest impact on voiding. The more solderable the board finish and the more active the paste chemistry, the lower the voiding will be.

The **QFN (Quad Flat No Lead) Package** is a flat plastic package that contains perimeter leads underneath the device and larger pads in the center. Its popularity in surface mount technology is due to its compact size and slim body type. It is a small, contained package with a near chip-scale size footprint. Its thin body allows it to fit into many tight tolerances. The QFN offers excellent thermal and electrical performance and utilizes perimeter pads to ease circuit board trace routing. These qualities make the QFN an optimal choice for many situations where electrical performance, weight, and size are important in design. In spite of their many benefits, QFNs also present a number of significant manufacturing and reliability concerns.



QFN Package

Defects such as **shorts**, **voids**, **opens**, and **deficiencies**, often make assemblers wary of them, in particular the QFN's stout reputation for voiding. When combined with a lead-free process, the tendency toward void formation becomes even more of a concern.

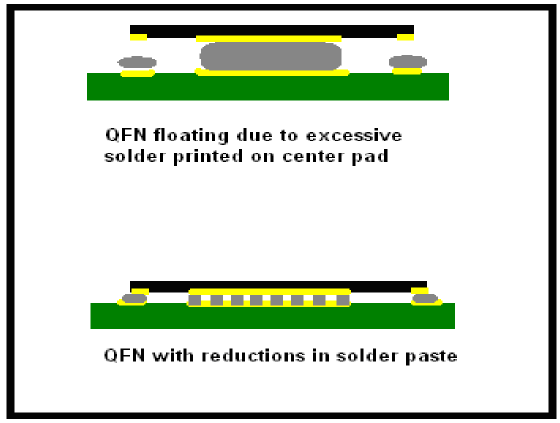


QFN Voiding

Voiding in QFNs occurs readily as volatiles become entrapped beneath the pad, typically upon reflow. Unlike Quad Flat Packs (QFPs), QFNs contain no leads and therefore no standoffs to allow for stress absorption and the escape of volatiles. As shown above, x-ray inspection is used to reveal defects such as large lake-like voids, showing holes that are typical of the type of voiding found in QFNs.

Another primary issue with QFNs is **component floating**. The ultra-small QFN package is typically designed with a large metal contact pad in the center. When solder paste is printed over this entire pad, or if the thickness of the paste is excessive, there may not be sufficient component weight and/or surface tension to pull the component down to the surface of the board; this can result in the QFN literally floating atop the molten solder.

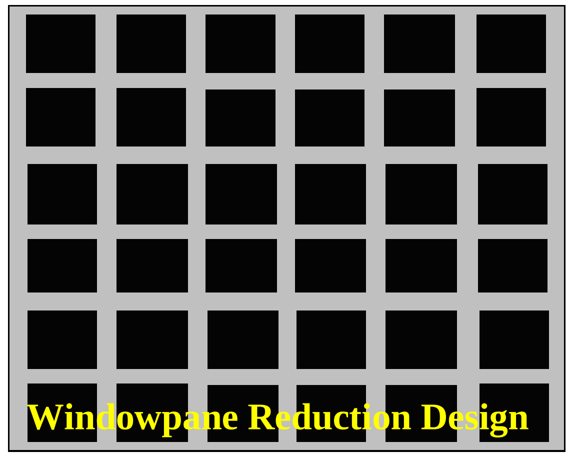
When the center pad aperture is designed to be too large, the paste deposits in this area may essentially be higher than the smaller pads along the edges of the QFN. As a result the QFN might not solder to the edge pads. This higher volume of solder in the center may also cause the QFN to tilt to one side during reflow and pull out of the opposite side leaving an open along the edge ultimately causing lifting of the QFN component.

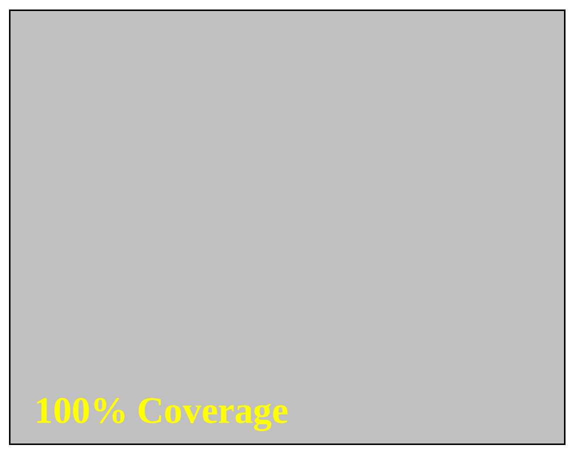


Ways to Reduce Voiding

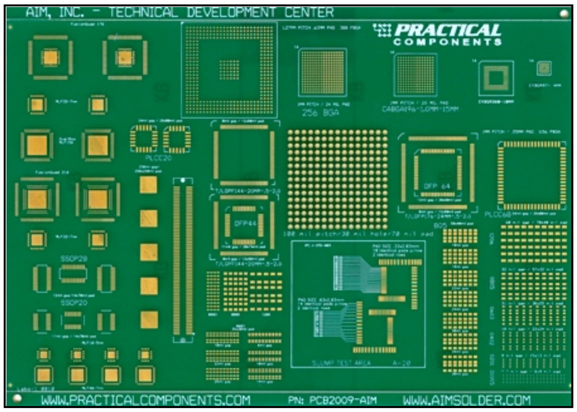
Adding Vias. To prevent floating, one must reduce the volume of solder paste printed on the center pad. The most efficient way to accomplish this is to follow the guidelines for stencil aperture reduction. This is accomplished by breaking down the stencil aperture into a series of smaller pads, reducing coverage to 50-80% of the pad area. This will not only reduce the volume of solder deposited, but also reduce the potential for voiding, splattering, and outgassing.

The best approach, of course, is stencil modification during the design stage. In many instances, adding vias to the pad and creating a divided, grid-type ground pad will reduce voiding. Although flux may still become entrapped in the via, outgassing does not usually form voids because volatiles can escape easily. However, Contract Manufacturers usually cannot alter the design of boards they are contracted to build, so the best solutions are modifications in aperture size, stencil thickness, and stencil design. Reflow profile modifications have minimal impact on voiding.

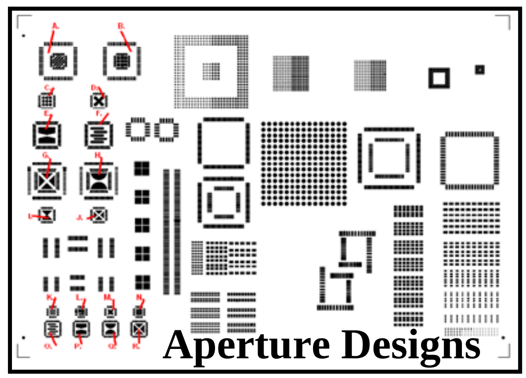




Aperture Reductions. A variety of designs and percent reductions were utilized to test the effectiveness of decreased aperture size. The most consistent and overall favorable results were achieved utilizing a 30-50% reduction with a windowpane reduction design versus 100% coverage. Typical solder paste coverage prior to reflow was 50-80%. Reduction designs may be dependent on the type of assembly being manufactured. Voids at vias can adversely affect performance on high-speed RF applications. It may be necessary to mask or plug vias in order to prevent solder from flowing inside the via during reflow. In addition, reducing the large ground plane pad in the center by utilizing a grid pattern of smaller pads in order to break it up or masking it off will allow some outgassing between the pads under the component.



Pictured left is the AIM Technical Development Center QFN Test Assembly (Part Number PCB2009-AIM), designed in partnership with Practical Components. The corresponding illustration below shows various aperture designs that have been tested and show positive results in reducing large lake-like voids beneath QFNs.



Aperture Designs

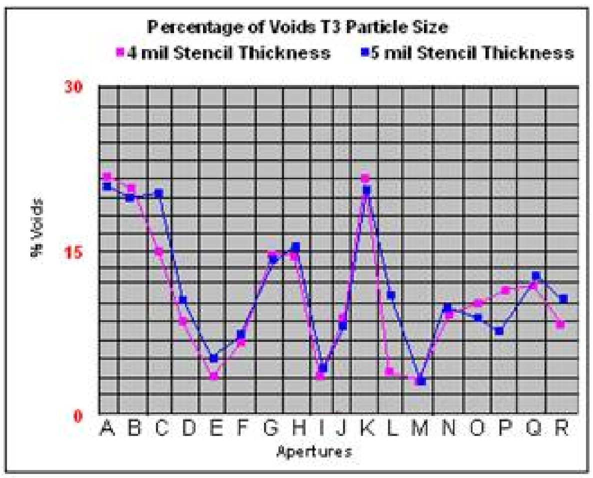
Based on statistics shown below, some aperture designs have a bigger impact than others. There is no direct relationship between percent reduction of paste deposits and voiding; however, seems that paste reduction in conjunction with the use of specific geometries impacts voiding volume and shape.

Aperture based on percent of paste compared to 100%

A. 57.26	I. 58.82	Q. 42.96
B. 47.14	J. 56.78	R. 43.20
C. 85.62	K. 77.80	
D. 84.06	L. 76.93	
E. 38.10	M.75.76	
F. 52.97	N. 78.10	
G. 59.08	O. 48.69	
H. 46.79	P. 34.85	

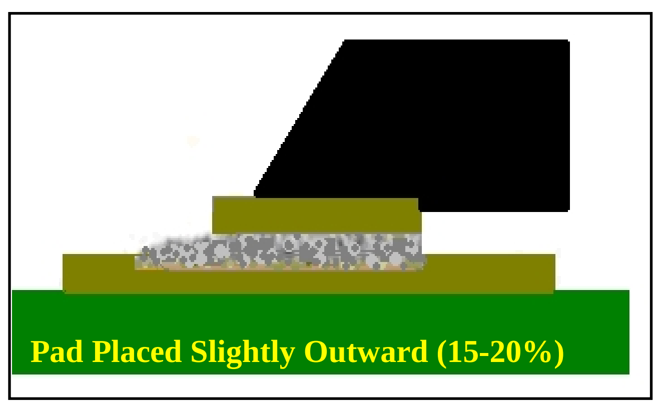
Note: Aperture design with proper reduction and geometry helps to eliminate voids. There is a large variation in volume of solder paste on these different geometries.

- E. 38.10%
- F. 52.97%
- I. 58.82 %
- M. 75.76 %



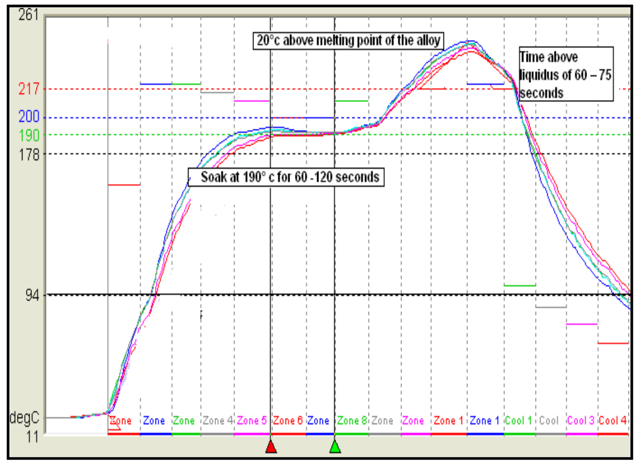
It was determined in most cases that a 4 mil stencil offered significant improvement in voiding over a 5 mil thick stencil. Nevertheless, it is important to note that reducing stencil thickness can have a negative impact on solder connections on the I/O pads.

Placement and Pad Design



Even and consistent **placement pressure**, as well as **consistent depth**, is imperative to successful QFN soldering. The component should be under tension on all sides to facilitate soldering. Moving the pads slightly outward 15-20% will help maintain tension and allow for some outgassing. Pads should not be "necked down" (narrowed) under the component for single row QFNs. This will allow for proper flow and help ensure that the traces do not fail during thermal cycling.

Profile



A ramp-soak profile with a higher soak temperature and time has proven to reduce the quantity and size of voids when stencil thickness and aperture reductions have been targeted first. The typical profile should ramp to 190° C from ambient in 75-90 seconds and soak at 190°C for 60-120 seconds, continuing to 20°C above the alloy's melting point with a time above liquidus of 60-75 seconds.

Uncertainties in this evolving product cannot be overlooked. As explained, changes made with the design stage are most effective, particularly minimizing the material that must outgas volatiles within the flux medium. This is the primary cause of most voiding issues. Due to the large size of the center pad, adding vias to the pad and making a divided, grid-type ground pad encourages the easy escape of volatiles when out-gassing occurs, resulting in void reduction and/or elimination. Profile adjustments are marginally effective. The alternate strategy is to minimize the volume of paste deposited under the QFN's large center pad through modified stencil design to deposit less paste. Doing so can minimize voiding, prevent floating, and achieve better results by making these two primary problems less likely to occur.

Package-on-Package Assembly Issues

Key Words: Open Joints, Warping, Open Solder Connections, Solder Balling, Package Warpage, Excess Paste, Paste Shorts, Excess Flux, BGA Voiding, Package Cracking, Solder Mask Damage, Mask Misalignment

Package-on-Package (POP) construction is an emerging technique for saving space on a PCB assembly. This feature allows more power and functionality within the same footprint of a single Ball Grid Array (BGA). POP involves stacking packages on top of one another to integrate them into one, more powerful component; much the same way one adds floors to an office building under construction to create a high-rise building with more floors, and consequently more available offices. The expansion in POP is vertical, thus height can eventually become a factor in processing.

This is usually not of great concern since advanced package and silicon design now allow the building blocks of the POP to be thinner using wafer thinning and flip chip interconnection. The number of stacked packages in a typical POP is currently no more than two; the bottom component being the active IC (with the most connections to the PCB) and the top unit(s) being added memory.

POP devices are stacked one on top of the other either during the original component manufacture, or during PCB assembly. POP packaging techniques may include direct soldering, wire bonding or conductive adhesives for device to device interconnection.

The primary problems associated with POP technology are **open joints**, **warping** of the two levels of component substrate and also issues with the underlying printed circuit board (PCB). In addition to assembly issues, reworking these stacked components, or just the top mounted part, can be challenging. For example, will the entire POP device be reworked? Or only parts of it? What are those implications? And, when a POP is inspected using 2-D x-ray inspection, the data can become difficult to interpret due to the multiple levels of ball interconnection and wire-bonding that may occur within the package.

In the instance where a POP device has been previously built and is simply being attached to an assembly, the attachment method will not differ significantly from that of a normal BGA device. Solder paste will be printed or dispensed onto the pads of the component footprint, the device placed on the footprint, and the POP reflow soldered into place in typical BGA fashion, however other considerations come into play. First, the POP may well be higher in mass than a comparatively-sized BGA, and thus requiring more careful profiling. Second, the complexity of the POP device, and its multiple layers and solder connections, make the attachment tricky. Some defects to look out for are, individual layers warping (due to their material composition) and TCE mismatch with other layers. These defects may ultimately cause a failure of solder connections.

It has been found that package warpage during reflow heating is intimately related to the package stacking yield on board.[1] Because the bottom package for POP is very thin (in order to keep the total mounted height low), the warpage variation is large from room temperature to reflow temperature due to CTE mismatch among the die, molding compound, and substrate in the package. Additionally, solder connections of the upper POP BGA stacked layers may reflow again, balls can melt and blend with the lower-temperature connecting alloy, creating problems traditionally associated with soldering BGAs at too high a temperature.

So the goal is to attach the assembled POP to the new PCB assembly without reflowing the upper stacked layers and subjecting them to yet another reflow thermal cycle.

---

[1] Bernard, D., and Willis, B., "The challenges of package on package (POP) devices during Assembly and inspection," Proceedings of SMTA International, 2009

For bottom layer POP package attachment to the PCB, there is no specific difference for solder paste printing than on any other fine pitch application. What is necessary is good control and repeatability of the printing process with 100% paste transfer, as one would require for a typical BGA assembly.

In the instance where POP packages are assembled on the PCB during PCB assembly, there are a few considerations. As previously mentioned, the logic device is on the bottom of any stack, as there are more connections out of that package than from the memory devices. The OEM, or contract assembler, can take the two or more device layers and form the POP interconnections, as well as the board interconnections, during assembly or the POP elements can be already combined by the original manufacturer as an individual package for direct placement, as if it were a BGA.

Second level assembly of the POP package involves, first of all, very accurate placement, not beyond the capabilities of current generation auto placement equipment. Accurate control of the Z dimension is critical due to the custom height requirement of the POP relative to the other standard components or the bottom-level BGA.

Upper-level or second or third BGA layer attachment to the base BGA is unique in that slight variations in planarity – like stacking blocks, saucers, or any other object – become exacerbated the higher one goes. These variations are of course very slight, and may increase, or relax, as the POP (and each individual layer) passes through thermal cycling to reflow. Thus, more than one method of attachment to the base BGA is available, and is being investigated as the technology of POP develops. One method is ‘dipping’ the BGA in flux to a depth of approximately 50% of the height of the balls beneath the device, then placing the device atop the bottom layer for reflow attachment. Another more forgiving method that is better at compensating for slight variations in coplanarity is dipping the device in solder paste rather than flux.

In terms of upper layer attachment to the bottom BGA, solder paste dipping has become a popular technique because it helps overcome the variations that occur between the ball mounting positions on the different levels of the POP device. Using paste has been found to be a far more forgiving process than using flux, and it is easier to inspect on the surface of the terminations before placement. The term ‘Dip Paste’ has been coined to describe a paste with the special qualities that satisfy the needs for this application. The dip paste is applied directly onto the balls of the device on the placement platform. In ‘Dip Paste’, we look for certain characteristics including slump resistance, tolerance to humidity, excellent activity and wetting qualities, pin-probe testability and lead-free compatibility, among others. The chemistry should be compatible for use in air reflow and have minimal voiding, obviously.

The depth of the paste in the applicator and the depth of insertion of the device into the paste must be strictly controlled (very tight Z-axis programmability). Trials have shown that if the solder balls are pressed into the paste by more than 50% of their height, then the solder paste tends to wrap around the ball terminations, which increases the amount of paste pick up. This can then lead to excessive paste deposits, which, in turn, leads to the increasing possibility of solder shorts.

Care needs to be taken during reflow to make sure that there is minimal evidence of vibration in the conveyor system. Stacked packages, by virtue of their multi-layer structure, have more opportunities to suffer from misplacement.

One solution to the voiding issue is the use of a one-step underfill, a low surface tension, reworkable, one component epoxy resin that aids wetting and at the same time helps prevent voiding. The epoxy can be dispensed after paste application and before component placement. The final assembly of the PCB (and POP) can therefore be reflowed and cured in a standard lead-free paste profile. By eliminating a second cure cycle and second assembly step, the one-step underfill does it all at once, saving significant amounts of time. Better results with POP assembly/attachment are realized; accomplished through excellent capillary action, faster reflow characteristics, rapid cure speeds, and good fluxing action, aided by the use of the underfill.

There are many possible, specific defects that can occur in POP assembly. These can include:

· **Open solder connections**
· **Solder balling**
· **Package warpage**
· **Excess paste**
· **Paste shorts**
· **Excess flux**
· **BGA voiding**
· **Package cracking**
· **Solder mask damage**
· **Mask misalignment**

As POP technology advances, materials and processes will be optimized to meet the challenges presented by this emerging and promising packaging approach.

Wave Soldering

So you've aced surface mount assembly and you're ready for wave soldering. Now you're going to see some real defect levels.

Key Words: Flux Application, Foam Fluxing, Spray Fluxing, Preheat, Conveyor Belt Speed, Pot Temperature, Solder Pot Contaminants, Phosphorous, Embrittlement

When it comes to **flux application**, don't put on too much, don't put on too little. **Air knives** are used when **foam fluxing** and are adjusted to remove only the excess amounts of flux. **Spray fluxing** is the preferred method for no-clean and VOC-free flux application. It is much more controllable over the old method of foaming. Spraying will give you a more reproducible process window.

Regarding **preheat**, don't get too cold… don't get too hot. The only purpose for this is to drive off the solvent and minimize thermal shock. Don't overcook the board or you'll kill off activity. If you see too much smoke and hear that sizzle, crackle, pop, go ahead and give it a bit more heat.

Conveyor belt speeds will vary between assemblies, but typically range between 3 to 5 feet per minute. The typical wave process length is 1 ½ to 2 ½ minutes.

The typical **pot temperature** for Sn63/Pb37 is 470°-500°F.

The most common **solder pot contaminants** are copper, gold and nickel. If these exceed upper control limits, you will notice a grainy appearance in your solder joints.

Phosphorous is sometimes used by solder manufacturers to give the *illusion* of lower dross. However, phosphorous can cause pumps to plug, create a build up in the wave baffles, and, if too high in concentration, **embrittlement** in solder joints.

Other contaminants that can cause embrittlement are copper, gold, iron, aluminum, and zinc. Bear in mind that phosphorous is not generally tested for when conducting a solder pot analysis.

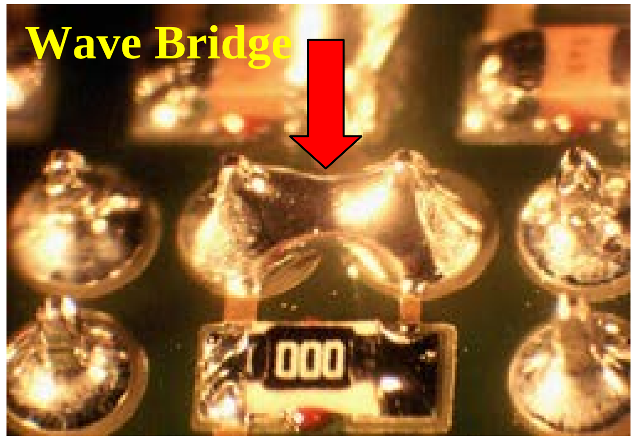
Wave Solder Defect Analysis

Again, you've tried your best to ensure a defect-free process, but what if . . .

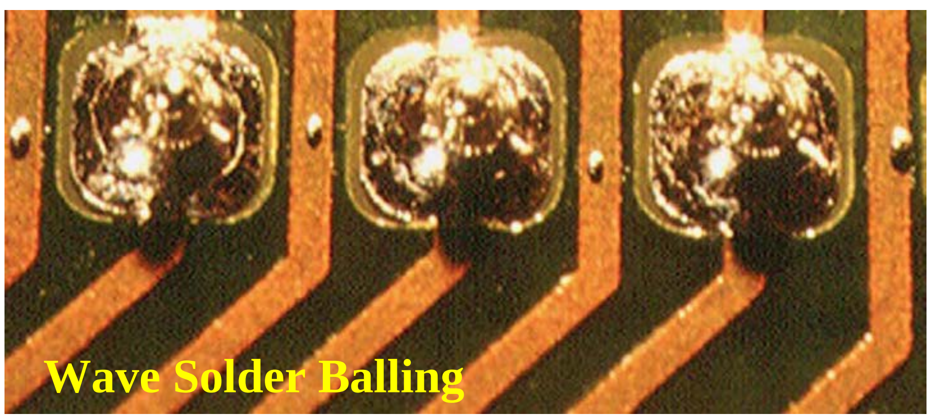
Key Words: Wave Bridges, Micro/Solder Balls, Spider Webbing, Micro Balls, Blow Holes, Skips, Low-Residue Fluxes

Unfortunately, if you are using a low-residue no-clean liquid flux, chances are that you will see all the above. Keep in mind that you are working with technology that was originally designed for high solids fluxes. In general, the higher the solids content in the flux, the lower your defect rate will be. Solids in flux provide a thermal interface, help strip solder off non-metallic materials, reduce bridging, and in general make nice shiny joints. On the other hand, high solids fluxes have thicker residues and often require cleaning. Many companies have chosen to trade these for the higher defect levels and tighter process windows associated with low-residue no-clean fluxes.

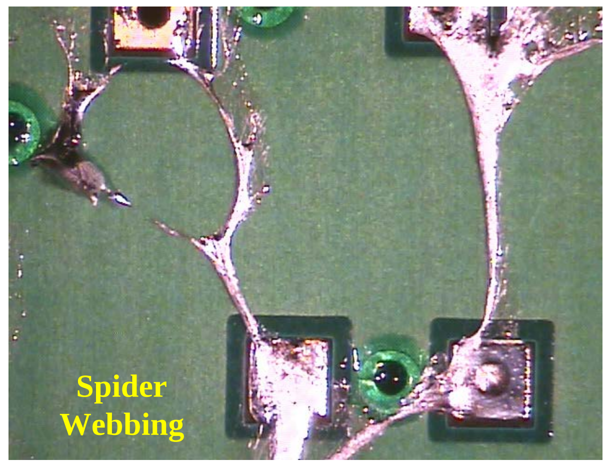
Wave bridges normally are due to the pin density of the boards. To eliminate bridging on high pin count leads, you should trim the leads as short as your specification will allow. In addition to short leads, you can increase space between them by reducing the annular ring through either board design or solder mask placement through your board vendor. Carrying as much of the solids content of the flux through preheat to the wave as possible through profiling can help to reduce wave bridging. The use of a nitrogen wave blanket also will help to reduce wave bridging.



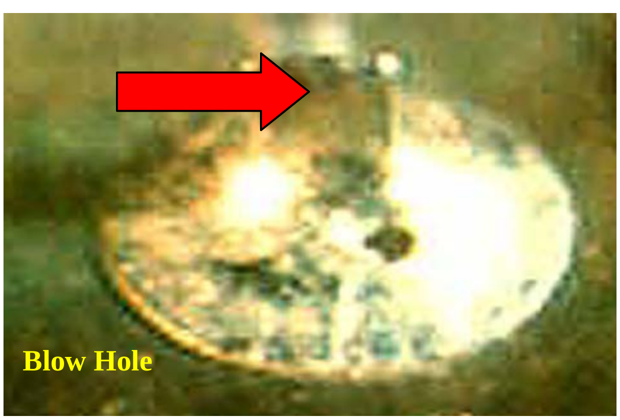
Micro balls on top of the board indicate that the board was too wet when it hit the wave. This is very common on an operator's first attempt with a VOC-free flux. These tiny solder balls blow through the vias or plated through holes to the top of the board as the flux begins to boil. Micro balls indicate that there is a need for more preheat (either in time or temperature) in order to evaporate the solvent of the flux.



Solder balls on the bottom side of a board can be solids content related. A higher solids flux will help to eliminate these. However, more often it is the solder mask that causes bottom side solder balls. Either the solder mask softens and allows the solder to cling to it or the surface topography is too smooth, which allows the solder to cling to the mask. To troubleshoot this, check the mask or run the boards cooler by a reduction of preheat and/or solder pot temperature.

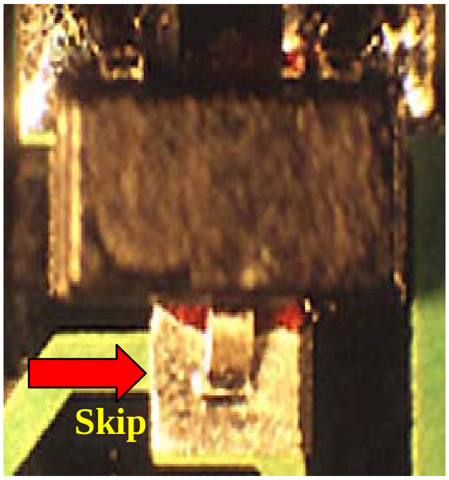


Spider webbing looks as if someone waved a metallic web or net on the bottom side of the circuitry. This occurs when the preheat is too high or long, or the flux application was not sufficient to aid the separation of the solder and the board. A hint that this may occur is if the plume of smoke is very faint as the board hits the wave. To cure this, increase flux coverage or decrease the preheat temperature.



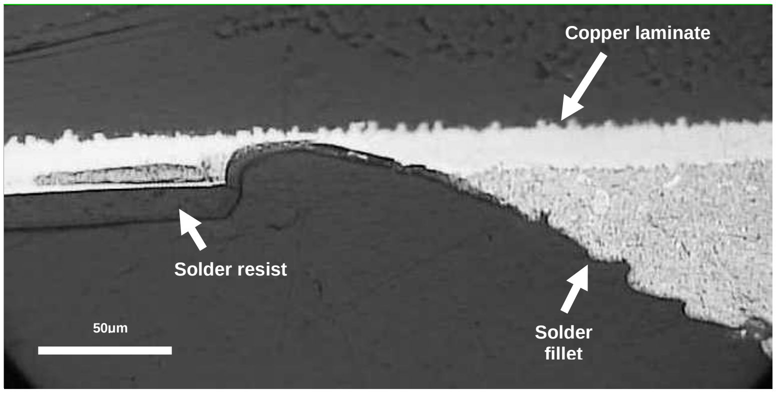
Blow holes normally are the result of a board problem. The symptom of these is a solder barrel with a hole on the side. The cause of this normally is a void in the Plated Through Hole wherein the board outgasses, blowing the solder out of the hole. This is a board supplier issue.

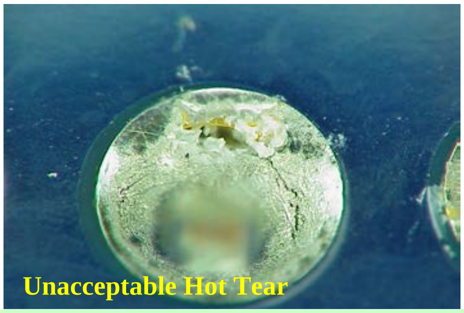
Skips normally occur on the backside of a component. These are due to flux solids content, board angle of contact, or pad design related. Try to keep more flux on the board and change to a steeper peel back or angle of attack. This can help to get the trapped gases out of the way of the solder, thus allowing the molten solder to make contact with the component pad. Flux activity also can help- sometimes a more aggressive flux will allow faster wetting to these areas.



The typical pot temperature for Sn/Ag/Cu including CASTIN® is 265°C to 270°C (509°F to 518°F).

Lead-free wave soldering has many of the defects described above. The two most common are bridging and insufficient hole-fill. This is due to the surface tension of the alloy. Bridging can be corrected by flux chemistry. The higher solids-content fluxes will normally help this defect. However, it is important to understand that lead-free alloys wet slower than tin-lead alloys. Dwell time in the wave needs to be longer. Typically, if tin-lead wets in 2 seconds, lead-free will take 5 seconds. This leads to the next defect, which is hole-fill. Longer dwell times are needed to correct this. Most of the time, insufficient hole-fill will be found on grounded PTH or other high density heat sink areas. This then leads into the next problem with lead-free alloys; selective soldering. Selective soldering has the inherent problem of dwell time. As much as lead-free alloys love dwell time they are also very aggressive dissolvers of base metals. The long dwell time of selective soldering dissolves plated through-holes and pads from the board. This is often called copper erosion.





Another solder defect that is alloy dependent is called hot tears. Hot tears are actually shrink cavities that form during cooling. According to IPC inspection criteria these are acceptable if the bottom of the tear can be seen.

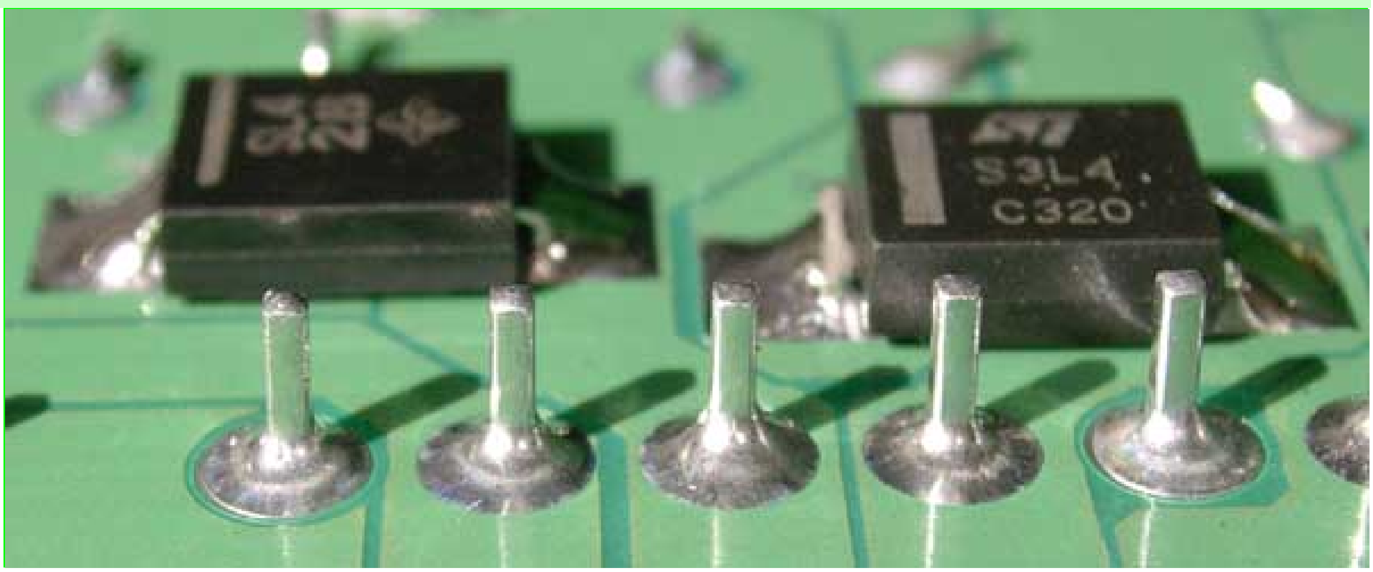
Pictured right is an example of a commonly-seen and acceptable hot tear. Please note that the solder is very grainy. This is an alloy issue.

Below is a comparison of different alloy appearances after cooling.

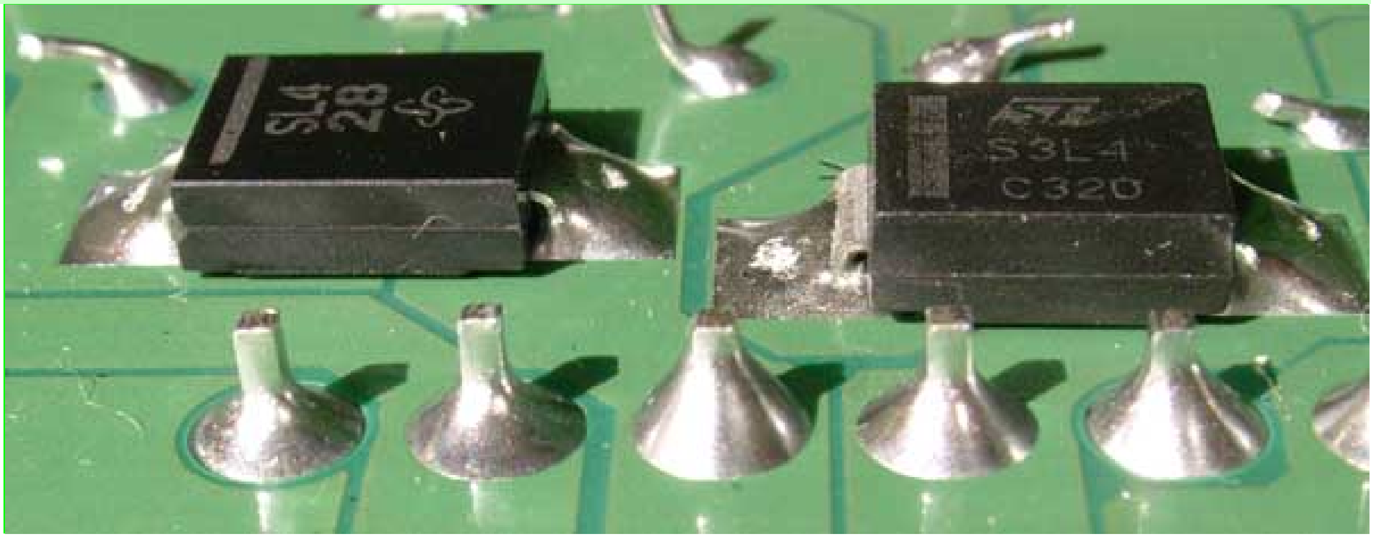
SN100C® SAC 305

Typical comparison of alloy types in wave soldered connections:

Sn63



SN100C®



Lead-free alloy choices for wave and selective soldering are based on copper dissolution, alloy cost, and solder joint inspection requirements. In general, silver-containing alloys add cost to the alloy, and these alloys also exhibit higher copper dissolution. The most common trend in lead-free flow soldering is to move away from SAC305 towards Sn/Cu/Ni/X alloys. Low copper dissolution alloys that are proven in the selective soldering applications are SN100C®, and CASTIN®.

Low residue fluxes in wave soldering leave very little residue because they are low-solids fluxes. Fewer solids mean faster evaporation of solvents and less time for the solids in the flux to do their work. Solids, however, are the part of the flux that do most of the work in terms of wetting and promoting the formation of good solder joints. Getting good soldering results means using these fluxes optimally through tight process control.

Optimizing a wave soldering process using low-residue fluxes depends on a fine-tuned process, whereby minimal process temperatures – just enough to get the job done – and minimal dwell times are desirable. Too much preheat, or too high a temperature setting, and the volatile part of the flux is gone before the solids can get their job done. Excessive dwell times or wave heights can impact solderability resulting in poor wetting, de-wetting, and/or component damage.

It is imperative and especially important that the following key process variables be closely monitored and controlled when using a low residue flux:

Conveyor speed
Preheater temperature
Solder temperature
Wave height
Wave contact time
Wave back flow (peel back)

The flux system chosen also has a profound impact when using low residue flux. AIM recommends the use of a spray fluxer versus a foam fluxing system. An ultrasonic fluxing system is recommended as it moves the flux into tight spaces as needed. Additionally, using this type of system allows for a higher volume of flux to be applied without detrimental effects. Ensuring sufficient flux dispersion is key to achieving optimal results with low-residue fluxes.

Hand Soldering

This is an area that many engineers do not pay attention to until a field failure occurs. Then they discover that one of their operators has a private stock of acid core wire stored under his desk for the hard-to-solder components. This is the final operation that occurs on the board, and it is often overlooked. It should not be.

Key Words: No-Clean Cored Wire, Safest Method of Hand Soldering, Touching Up Solder Joints

When using a **no-clean cored wire**, operators should be gloved. One fingerprint in the wrong place can cause electromigration. If any touch-up flux is being used, it should have been qualified in the *raw* state, since you do not have any method to insure uniform heating has taken place to destroy the active acids. Just because the flux has passed some electromigration or SIR testing does not necessarily mean that it is safe to be left on *your* board. Test it under *your* conditions.

The **safest method of hand soldering** is to, whenever possible, not use any other flux other than what is in the wire. Operators also should be trained to solder properly. The operator should heat the component lead and flow the solder onto the pad. This is the proper method, but you may have to fight with operators to proceed this way because it is normally slightly slower than heating the cored wire directly.

Finally, try to avoid **touching-up solder joints**. Touched-up joints are normally the ones that fail first because of the additional thermal stress and intermetallic formation that occurs during touch-up. In general, if it looks like the solder has reflowed and has made a solder fillet, it probably is OK. In other words, if it ain't broke, don't fix it. For additional information regarding lead-free soldering refer to the technical data sheet for the alloy and flux used.

Testing

Now that you have assembled the board and are sure that all of the joints are good and the components are each in the right place, it is time for those pessimistic quality people to test the assembly to try to prove you wrong.

Key Word: In Circuit testing, pin probe testing (bed of nails/flying probe)

The **pin probe testing** of paste residues (which is becoming increasingly popular) is often problematic. However, in paste-in-hole applications, pin probe testing problems are more pronounced due to higher volume of flux residue. If you have any design input, then make the leads on these devices slightly longer. This will allow the flux to stick to the board and not the lead. If the lead is too short, it will be capped with flux, which can make it very difficult to pin probe test.

If you probe directly out of the oven, you will get different results than if you probe a day later. The warmer the residue, the softer and stickier it is. Deciding when to probe is often a tough call, since you have to deal with both in-line and repaired boards. You will have to make this decision based on your own production needs.

The use of a **true pin testable solder paste** also simplifies the testing process. A true pin testable solder paste is one which has a waxy type coating that is neither sticky nor gooey, and is easy to penetrate without clogging probe tips not only immediately after reflow, but for up to months later. Several types of probes are available depending upon your individual testing requirements.

Glossary/Index

Activator: A chemical that improves the ability of a flux to remove oxides and aid the wetting of parts being soldered.

Annular Ring: The conductive area around a plated through hole.

Billboarding: A discrete component with both terminations soldered, but is laying on its edge.

BGA Voiding: Although specifications on percent voids exist, BGA voids still can form. Provided the voids are not a champagne void created at the board pads due to poor platting, a void is not a reliability risk. This was documented in the IPC SPVC SAC alloy study.

Blow Holes: Small holes or voids caused by out gassing in a plated through hole.

Bridging: A soldered joint that spans two conductors not intended to be connected, creating an electrical short.

Copper Erosion: Lead-free alloys tend to dissolve the contact copper into the bath. This can eventually remove the pad from the board or cause a separation from the PTH to the ring on the board surface.

Delta (Δ) T: The greatest difference of temperature found across an assembly.

Dewetting: Retreating of solder from some or all parts of a substrate that initially was wetted.

Electromigration: The tendency of conductive material to spread from one solder interconnect to another, causing a short circuit.

Halides: Compounds containing fluorine, chlorine, bromine, iodine. These are parts of the activators of certain types of flux and might need to be cleaned due to their corrosivity or conductivity.

Hot Tears: Openings in the surface of the solder that are caused by cooling cavities in SAC305 alloy and other similar alloys

Liquidus: The temperature at which solder reaches its fully molten or liquid state.

Micro Balls: Tiny solder balls as related to wave soldering.

Non-wetting: A surface that has contacted but rejected molten solder.

On-Contact Printing: Zero snap-off, no print gap.

Opens: Two electrical conductors not bridged by solder. This can be due to insufficient solder or non-coplanarity of the lead at its point of connection.

Outgassing: This is the emission of impurities from a PCB or component that occurs when the assembly is exposed to heat or reduced pressure.

Pad: Area on which solder paste is printed and a component is placed.

Package Warpage: Package Warpage, often referred to as "coplanarity", occurs as received and during reflow. When handling finer pitch devices, this issue becomes especially critical. For instance, many BGA packages are received with a .008" warpage specification, however most boards are now printed with .004-.005" of solder paste. This results in some of the balls not contacting the paste ultimately resulting in HiP. This is also critical when using any multiple IO leadless device (LGA, QFN).

Peel Back Angle: The angle at which the PCB contacts the solder wave.

Pin Probe: The conductive member by which electrical contact is made between the PCB pad or lead and the tester.

Popcorning: Eruptions in an IC during reflow, normally the result of moisture absorption.

Reflow Profile: The time vs. temperature graph of a PCB as it is processed through a heat source.

Rheology: The science or study of a materials flow in terms of stress strain and time.

Skips: As related to printing, skips are component pads that were missed during the printing process. As related to wave, areas that were intended to be soldered but were missed due to shadowing or gassing.

Slump: A spreading of solder paste that may lead to bridging. May be cold (occurring before reflow) or hot (occurring during reflow).

Snap-off/Print Gap: The distance between the stencil and the PCB during printing.

Solder Balls: Tiny spheres of solder usually located around a solder joint or remotely around the board.

Solder Beads: A large solder ball positioned between the terminations of a discrete component, usually a resistor or capacitor, but can also be found on large and small transistors as well.

Solder Fillet: The solder meniscus or joint formed by the solder between the pad or hole and the component lead.

Solder Mask Damage: A chemical attack of the mask by the flux typically results in solder mask damage. This is often associated with poor adhesion to the board due to oxidized copper traces or a poorly cured mask. This can be mitigated by controlling the solvents in the liquid flux being used.

Solidus: The temperature at which solder reaches its fully solid state.

Solids Content: The percentage by weight of non-solvent material in a flux.

Squeegee: A plastic, metal, or fiber blade used to push solder across the stencil surface while filling the stencil apertures.

Tombstoning: A soldering defect in which a component is pulled into a vertical or angular position leaving one side unsoldered.

Torn Prints: Paste printing defect which results in printed paste being ripped from the board pads, resulting in clogged stencil apertures.

Viscosity: A measurement of shear stress over shear rate, which is the resistance of a material to flow.

Wave Bridges: Bridging that occurs during wave soldering between pins or components.

Webbing: Wave solder defect recognized by a spider web like extension of solder across the non-conductive portion of a PCB.

Wetting: The formation of an intermetallic allowing the spread of molten solder over a base metal.

Reference Section

Common Solder Alloys

ALLOY	Ag	Cu	Pb	Sb	Sn	Melting Point (°C)	Density (lbs/in.^{3})
Tin							
Sn99					99.9	232	.2628
Sn96.5	3.5				96.5	221	.2657
Sn95	5				95	221-240	.2668
Sn90			10		90	183-213	.2682
Sn70			30		70	183-193	.2889
Sn63			37		63	183	.3032
Sn62	2		36		62	179	.3036
Sn60			40		60	183-188	.3068
Sn50			50		50	183-212	.3202
Sn40			60		40	183-247	.3430
Sn35			65		35	183-250	.3431
Sn30			70		30	183-257	.3509
Sn20			80		20	183-280	.3686
Sn10			90		10	275-302	.3881
Sn5			95		5	308-312	.3980
Sn3			97		3	314-320	.4030
Silver							
Ag2.5	2.5		97.5			304	.4070
CASTIN®	2.5	0.8		0.5	96.2	215-217	.2670
SAC305	3.0	0.8			96.2	217-218	.2680
SN100C®		0.60		Ni0.04	99.4	227	.2670
Ag5.5	5.5		94.5			305-364	.4079
Antimony							
Sb5				5	95	232-240	.2617
Sb2			63	2	35	185-243	.3350
Sb1			79	1	20	184-270	.3680
Lead							
Pb88	2		88		10	268-290	.3880
Pb68			68	2	30	185-243	.3429
Pb80	2		80		185	252-260	.3826
Pb94	1.5		93.5		5	305-306	.3982
Pb93	2.5		92.5		5	299-304	.3980
Pb95	2		95		3	299-305	.3980
Pb96			96		2	252-295	.3956

Solder Paste Powder Size Classification

Particle Size	Mesh	IPC Type
75 micron	-200/+325	2
55 micron	-270/+400	N/A
45 micron	-325/+500	3
25 micron	-400/+500	4
20 micron	-500/+600	5

Common Cored Wire Diameters

Inch	.010	.015	.020	.025	.032	.040	.050	.062	.092	.125
MM	.25	.40	.50	.63	.80	1.0	1.25	1.57	2.34	3.17
Gauge	33	28	25	23	21	19	18	16	13	10

Temperature Conversion Table

°C = (°F – 32) X 5/9 (or 0.55)   °F = °C x 9/5 (or 1.8) + 32

°F	°C	°F	°C	°F	°C	°F	°C
-40	-40	115	46.1	270	132.2	430	221.1
-35	-37.2	120	48.9	275	135.0	435	223.9
-30	-34.4	125	51.6	280	137.8	440	226.6
-25	-31.6	130	54.4	285	140.5	445	229.4
-20	-28.9	135	57.2	290	143.3	450	232.2
-15	-26.1	140	60	295	146.1	455	235
-10	-23.3	145	62.8	300	148.9	460	237.8
-5	-20.5	150	65.5	305	151.6	465	240.5
0	-17.78	155	68.3	310	154.4	470	243.3
5	-15.0	160	71.1	315	157.2	475	246.1
10	-12.2	165	73.9	320	160	480	248.9
15	-9.4	170	76.6	325	162.8	485	251.6
20	-6.6	175	79.4	330	165.5	490	254.4
25	-3.9	180	82.2	335	168.3	495	257.2
30	-1.1	185	85	340	171.1	500	260
32	0	190	87.8	345	173.9	550	287.8
35	1.6	195	90.5	350	176.6	600	315.5
40	4.4	200	93.3	355	179.4	650	343.3
45	7.2	205	96.1	360	182.2	700	371.1
50	10	210	98.9	365	185	750	398.9
55	12.8	212	100	370	187.8	800	426.6
60	15.5	215	101.6	375	190	850	454.4
65	18.3	220	104.4	380	193.3	900	482.2
70	21.1	225	107.2	385	196.1	950	510
75	23.9	230	110	390	198.9	1000	537.8
80	26.6	235	112.8	395	201.6		
85	29.4	240	115.5	400	204.4		
90	32.2	245	118.3	405	207.2		
95	35	250	121.1	410	210		
100	37.8	255	123.9	415	212.8		
105	40.5	260	126.6	420	215.5		
110	43.3	265	129.4	425	218.3		

NOTES

NOTES

NOTES

NOTES

Suggested Retail Price $39.95

AIM
25 Kenney Drive
Cranston, Rhode Island 02920
(800) 225-5246 · (401) 463-5605
(401) 463-2866 Tech Fax · (401) 463-0203 Fax
www.aimsolder.com

Rev. 10-12