# Reflow Soldering Process Considerations for Surface Mount Application

by Jim Bergenthal

KEMET®
Electronics Corporation
P. O. Box 5928
Greenville, SC 29606
Phone (864) 963-6300
Fax (864) 963-6521
www.kemet.com

F-2102A 7/95
Reprinted 10/97

## INTRODUCTION

Reflow soldering, like wave soldering, is not a new manufacturing process. The hybrid industry has used and refined the art of reflow soldering for many years. However, with the advent of Surface Mount Technology (SMT), reflow soldering has expanded in the number of types and has been studied, refined and explored as never before.

Many different opinions have been expressed about the best process. We have found that the best or optimum process is the solder process which resulted in meeting the goals of reflow soldering for the SMT application. You can select the optimum solder reflow process for your application if you:

- Understand the goals.
- Understand the process and modify the process to meet the goals.
- Realize the best results are obtained from the process which treats your product with care.

## GOALS

The goals for reflow soldering are of two basic types. The first is more traditional and includes:

- Uniform solder joints.
- Minimum repairs and part replacements.
- Minimum solder skips, solder balls and part movements (tombstoning, etc.).
- Maximum cleanliness of the completed assembly.
- Maximum flexibility to allow soldering a large number of circuits with minimum changeover time.

The second type of goal is not normally thought of by process engineering when first establishing the reflow solder process. These goals include:

- Minimum time above the liquidous solder temperature to reduce solder grain growth, resulting in a more durable solder joint.
- Minimum stress and damage to the Printed Circuit Board (PCB).
- Minimum damage and stress to the SMT parts.
- Minimum "leaching" of part termination materials.
- Optimum conditions to minimize movement of parts (all the neat names of tombstoning, drawbridging, etc.).

These goals are in line with increasing product quality and reliability.

To achieve these goals, it is important to understand the process, to modify it to meet our goals and to insure the product is protected. There are many technical and commercial descriptions of the reflow process already in print. Our look at the process will be a little different as we look at each phase, explain what it is attempting to achieve, show concern for important items, and point out some "do's and don'ts". Each of the specific processes will be reviewed for its positives and what the critics are saying about it. Methods of improving on the process, to balance these criticisms, are also dis-cussed. The aspect of treating the product with care will be interwoven in the discussion and further discussed for some specific cases.

*Let us look into the process further.*

## REFLOW SOLDER PROCESS DESCRIPTION

The basic reflow solder process consists of:

- Application of a solder paste to the desired pads on a printed circuit board (PCB).
- Placement of the parts in the paste.
- Applying heat to the assembly which causes the solder in the paste to melt (reflow), wet to the PCB and the part termination resulting in the desired solder fillet connection.

A. Solder Paste
The solder paste mixes are improving as the demands of reflow soldering for SMT increase. Selection and specification of the optimum paste is a key item in the reflow solder process.

B. Placement of the parts in the paste is not difficult if the pad design considers all the applicable tolerances. (See KEMET Application Bulletin "Surface Mount - Mounting Pad Dimensions and Considerations"). Care should be taken during the transportation of the PCB's not to smear the solder paste or move parts. Inspection of placement accuracies and subsequent manual movement of parts in wet paste has been shown to increase repair rates after soldering.

C. Application of heat to result in the eventual solder joint must consist of the following discrete items:

- Preheat cycle is intended to drive off most of the volatile solvents contained in the paste before the flux begins to work. This assists in initiating fluxing action on the solder powder and the metal surfaces to be joined.
- Additional preheat time to elevate the temperature of the PCB, solder paste, and terminations to a temperature near the melting point of the solder.
- Additional heat transfer to elevate the temperature over the liquidous point of the solder.
- Temperatures to be achieved are the liquidous melting point of solder. Liquidous points for -
  60 Sn / 40 Pb solder is .........188°C
  63 Sn / 37 Pb solder is .........183°C
  62 Sn / 36 Pb / 2 Ag solder is 179°C
  Additional heat is required to insure activation of the rosins. However, heat should be limited to minimize the times some parts are above critical temperature.
- Assisted temperature cooldown to the solder solidification temperature, followed by gradual (static) cooling to temperature near cleaning temperature.

## PREPARATION

To insure good reflow solder results, the following items should be considered. The temperature profile and equipment choices are only part of the equation for success.

- **Printed Circuit Boards**

  Bake boards at elevated temperatures prior to application of solder paste. This eliminates excessive moisture from the board. Moisture in the board, under solder resist layers, trapped within layers, etc., can lead to excessive solder defects. A time of 4 hours minimum at 65°C is generally adequate. Baking should be done within 8 hours of use.

  PCB assemblies, which are to be double side reflow soldered, should not be cleaned after the first side solder. This could induce a detrimental level of moisture, which will increase the defects when the second side is reflow soldered.

  Circuit boards are usually of two basic types. Bare copper pads (with a protective sealer to prevent oxidation) have been used successfully in many applications. When bare copper pads are used, care must be taken in the process and repair sequence. The protective sealer is designed to be easily washed off by normal cleaning and fluxing agents. Normal correction of errors, such as washing wet adhesive off a board which has been incorrectly processed, cleaning solder paste from a board which has exceeded the tack time of the paste, etc., cannot be done without removing the protective sealers and exposing the copper to oxidation. Adhesive cure temperatures may also have to be limited to protect the sealer. These sealers fortunately respond to over-temperature by changing color. The second type is the PCB with solder coated pads. These boards are usually fused or reflowed to increase the coverage of the solder to the edge faces of the copper and to prevent further chemical processes in the board manufacture from damaging the solderability by attacking the copper through the porous tin plate. This reflow can result in dome shaped pads to mount parts on. Hot air leveling is usually specified to minimize this feature. If you are going to use a "No-Clean" process, it may be important to specify the cleanliness of the PCB. Individual wrapping of the board will help keep it clean before use.

  Mounting pad designs and considerations are critical. Review of other standards (such as IPC, EIA, and KEMET's recommendations) are an excellent place to start. However, the best pad is the pad optimized for your set of conditions. Trial and retrial is important. Remember keep all pads, traces and parts away from the board edges. The amount will be determined by your board transportation methods, magazines, carriers, etc. Also, leave room between parts for visual inspection and for rework. Keep through hole vias out from under parts. Solder build up on the via, along with subsequent reflow during the soldering process, will tend to move the part at this critical time. Tenting the via with solder mask does not help this condition.

  Tight tolerance of PCB tooling holes to artwork is recommended. These will determine location of solder paste and components to the designed pads. Optical location during PCB manufacture is suggested.

- **Solder Paste:**

  Selection of a good solder paste is important. This is a subject unto itself. We suggest you work closely with your supplier and inform him of your process and profiles. Pastes have been developed which optimize the solder process, however, these have been developed for a specific set of profiles, and are not always applicable to your process. Some trial and inspection, and retrial, is important. Whichever paste you select should be inspected on a routine basis, and consistency from lot to lot is an important parameter. Two important variables are viscosity and slump tests. Understanding the effect of premixing and test temperature on viscosity is important to your incoming inspection program. Shelf life control and refrigeration is also important.

- **Rheology**

  Rheology of solder paste is one of the important characteristics as the flow and deformation behavior of solder paste directly affect the quality of paste deposition on the solder pads. Factors influencing the rheology of solder paste include:

  - Drastic differences in the density of the organic matrix and the metallic particles.
  - Relatively large particle sizes.
  - Suspension of metallic particles.
  - Spherical particle shapes.
  - High load of particles.

  The ideal rheology of solder paste should deliver the following properties:

  - Specified shelf life and stability.
  - No separation or settling.
  - No stringiness, but have enough tackiness to hold components after placement. A specified tack time of 8 to 12 hours is not unrealistic.
  - No slumping/sagging, but with a good

capability of leveling.
- Perfect transfer through the stencil.
- Clean release from the stencil.
- Clean breakoff of fine needles.
- Temperature insensitive in the manufacturing environment. (Temperature and humidity affect the tackiness and drying of the paste.)
- Smooth spherical particle shapes. (These result in least friction during screening, and best dimensional control for fine pitch soldering. Irregular shape particles are also more demanding of flux to clean the oxidized metal particles. For fine pitch soldering, the maximum solder particle size should be 1/3 the smallest screen opening.)
- Easy to clean, and not leave difficult to clean residues.
- Must minimize solder balls and solder splatter. (Especially important for "No Clean" processes.)

- **Purity**

Solvent purity and metal particle cleanliness (oxides, carbonates, etc.) are very important in minimizing solder balls and voids, and in improving solder wetting and shapes of solder fillets. These are obtained by specifying good quality pastes, and maintained by proper storage in closed containers below 75°F (refrigeration is best), as well as cleanliness of screening equipment. Shelf life controls are very important. If refrigerated storage is chosen, the container and paste should be brought to room temperature before opening. Failure to do so will result in condensation and contamination of the solder paste with water.

Metal percentages in solder paste are presented in weight. After reflow (the fluxes and rosins are removed) the reduction in volume results in 1/3 of the volume of the printed paste. This can guide you in the expected solder fillet versus thickness of solder paste laydown.

The introduction of “No-Clean” and Aqueous Cleaning processes have greatly changed the character and constituents of solder pastes. Some require higher preheat temperatures in the reflow profile. Make sure to consult the solder paste manufacture in determination of the solder profile.

**Stencil**

Selection of the optimum solder paste stencil is a trial and retrial situation. Each process engineer develops his own tricks. Some tips which have merit are:

- Stencil opening should be about 5% less than the size of the pad. This helps minimize solder balls and splatter.
- Stencil thicknesses should result in solder paste heights of .006 to.010 inches. This assists in minimizing the effects of coplanarity and in minimizing placement errors.
- Control the amount of solder for specific pads by controlling the size of the window as opposed to stepping or reducing the thickness of the screen.
- Keep the stencil clean. Develop a routine and follow it. (Do not introduce fibers from overuse or misuse of wipe materials.)
- Inspect the stencil and squeege routinely for damage.
- Do not return used paste to the jar.

**Solder Masks**

The best solder mask is no solder mask. PCB’s, with no solder mask, have only mounting pads on the outside layers. These have been referred to as “POOL” (Pad Only Outer Layers) boards. “POOL” boards may have initial higher costs, but these are typically offset by much lower repair rates.

However, when”POOL” boards cannot be used, solder masks will be beneficial. These are used to:

- Prevent solder bridges.
- Tailor solder fillets.
- Reduce tombstoning.
- Assist in cleaning the board.
- Protect fine traces from damage.
- Reduce the effects of metal electromigration.

Types of solder mask and some limitations to be considered are as follows:

- Wet Film

These are screened on wet materials. Problems with these include flow of the mask onto solder pads or testing pads, accuracy and repeatability of screening, reflow of solder plated traces under the solder mask, and flaking off of solder mask after the solder is reflowed. The new thixotropic materials have reduced bleed problems substantially. This is one of the most economical methods, and with proper controls and for less complex designs, can work quite well.

- Dry Film Solder Masks

Tighter tolerance location of the mask to the artwork is available with dry film. These are thicker than the wet film and can result in locations where solder balls and solder flash can be trapped. If the dry film is not properly attached, blistering and entrapment of moisture can result. If the dry film is not properly cured, solder balls and splatter may attach or embed themselves.

The cost of dry film solder mask is also not insignificant. With proper processes and designs dry film solder masks are excellent.

- Wet Film Photo Imagable Solder Masks

  This appears to be the solder mask of choice. It has the advantages of dry film tolerance and the low cost of wet film screening.

**Visual Criteria**

Define the optimum solder fillet for each style termination being used. This should be used in developing the process, training solder machine operators and visual inspectors, and as criteria for visual inspection and repair personnel. Graphical (photos and sketches) details make the best criteria.

*Key items to be considered (tied to our goals) are:*

- Repair is to be minimized. Define criteria so that only necessary repairs are made. Repairs should be linked to corrective action steps.
- Minimum solder fillets are preferred in SMT.
- Good solder fillets are best described by wetting angles (as opposed to amount or height of fillets).
- Solder splash and solder balls are very difficult to inspect and remove. Preference should be given to minimize them in selection of pastes and profiles. This has become even more important with "No-Clean" processes. The cleaning process is no longer available to wash away the solder balls.
- Shiny joints are not necessary. The luster of the solder surface is a function of the metal type, the solder atmosphere and the cooling profile. Shiny surfaces have been thought to indicate better durability, however little data is available to prove this contention.
- Size of fillets needed may be determined by strength (push-off) tests. These are destructive and should be done only to verify visual criteria requirements. A good portion of solder strength is contributed by solder under the part, thus minimum fillets with positive wetting angles are usually sufficient.

**Equipment Consideration**

All systems utilize conveyors or transporters of some type. Problems can occur during transportation or at interfaces of conveyors (i.e, to cleaners, or from in-line queueing systems). Board bounce can displace components in solder paste, and board corners can get caught in wide mesh conveyors, results in excess twist of the board.

## TYPICAL REFLOW SOLDER PROFILE

A typical reflow solder profile is shown in Figure 1. Each of the labeled points on the profile is discussed in the following:

(A) This is the start of the process. The PCB with the parts placed in solder paste begins to enter the first preheat zone.

For most applications, the PCB's are carried on a conveyor of wire mesh, and are transported through the system at a regular rate. Since the PCB's act as a load on the reflow system sections, it is important to profile the equipment with the manufacturing load. Board spacing (between boards in the direction of flow as well as side-by-side, if that is the plan), is important and manufacturing should be run as profiled. The use of hot gas convection additions to IR heating eases these considerations. See the later section on hints for profiling.

(B) Between "A" and "B", the temperature is increased at a rate of about 1°C / sec. until reaching 100°C. During this time, the volatiles and solvents in the solder paste are evaporated. The temperature is held at this point to insure all are out of the paste. Time depends on the mass of solder paste involved. This temperature may also vary depending on the type of solder paste selected.

A gradual preheat cycle is needed to minimize "skinning" of the top surface of the paste and entrapping volatiles and fluxes, which may later lead to voids and blowholes in the solder joint. Preheating also reduces the tendency of the vehicle in the paste to spread on exposure to reflow temperatures. Spread (and "skinning") can often lead to splattering and unwanted solder balls.

(C) Between "B" and "C", the temperature is further increased at a rate of about 1°C / sec. until reaching 150°C at point "C".

This preheat drying process initiates the activator, and cleaning of the surfaces to be soldered and of the solder particles. It is important that this function is allowed to happen before the burn-off of the organic activator happens.

The gradual preheat again is important to insure materials of unlike heat capacities adjust as equally as possible to the temperature excursions. A hold periodmay be included to insure all parts have reached the same level, and that the activators have completed their function.

(E) The temperature is again increased. Most systems do this by a more rapid increase of temperature with a rate approaching 4°C / sec. The solder melts at point "D" (about 183°C), and the rosin is further activated at about 200°C.

**To insure adequate soldering conditions, the minimum peak temperature should be 205°C. The maximum recommended peak soldering temperature should be 220°C.**

Many reflow solder profiles have been established with profiles as high as 240 to 260°C. This is not recommended as the time at or above critical temperatures (molten solder, glass transition temperatures, etc.) can be excessive, and degradation of material properties can result. In addition, higher than needed solder temperatures can lead to flux residues baked on the PCB and make it difficult to clean. High temperatures for long time also lead to excessive board discolorization, flux charring, excessive board warp and twist, and damage to some of the parts being soldered (see later discussions).

(G) Between “E” and “G”, the temperature is decreased. Controlled temperature zones are the desired way of cooling with cooling by the room ambient conditions as a secondary means. Cooling by forced air (fans) or by immersion in cooler liquids is not recommended.

In addition to the thermal shock potential to all components, analysis has indicated forced cooling can result in lead (Pb) rich dendrites in tin (Sn) rich matrix in the solder fillet. Natural cooling rates of 4 to 6°C per second are acceptable.

At point “F” the solder has reached the solidification temperature. The total time the solder is molten (“D” to “F”) should be minimized as this, along with the peak temperature, will determine the grain size and the strength of the solder fillet.

At point “G”, the PCB assembly enters an inline cleaning system which is generally heated in the range of 50°C to 100°C (dependent on the aqueous system used), and the solder reflow process is complete. If “No-Clean” system processes are used, the boards may be air cooled from point “G” to room temperature.

## PROCESS SPECIFICS

The main differences in reflow soldering processes lie in heat transfer methods. Heat transfer for reflow soldering is of many types, and in practice, some methods are combined. Radiation, conduction, convection and condensation are all used. The heat is applied from above, below, or all directions dependent on the equipment being used. The methods most prevalent, in early reflow processes, were the heated belts or convection tunnels used in the hybrid industry. With growth of SMT, the reflow industry concentrated on infrared (IR) and vapor phase (VP) reflow soldering methods. The application of heat through lasers and heated bars have been developed to solder large pin count IC’s. Recent combinations of IR heating coupled with hot gas (nitrogen or air) convection heating have become the predominant choice. These have helped to eliminate vapor phase reflow systems.

IR systems are the most prevalent production systems with variations of these left to the process engineer. Vapor phase systems have lost a lot of favor, with the push to eliminate CFCs playing a major role. Each of these reflow methods have positives and negatives, which can be overcome if they are understood. Once the methods and systems are understood, it is up to the process engineer to determine which method best fits the manufacturing process and mix of his products.

### Infrared Reflow Solder Process

When IR energy encounters a target, some of it is absorbed, some reflected and some transmitted through the target. The absorbed energy heats the targeted material, the reflected energy is returned to the source, and the transmitted energy is lost. Generally with most objects, a combination of the above takes place as some of the object’s charactertistics are reflective, some transparent, some opaque and some resistant. The chemistry of the material will partially determine the absorptive quality with respect to particular wave lengths. Physical characteristics, such as mass, length, shape, width, specific heat, cell sizes, etc., also come into play. At some wave lengths (in the visible range) color also comes into play.

There are two basic types of IR reflow systems, near wave length IR and medium wave length IR systems. As these systems have evolved and been improved, the wave length is becoming less descriptive and the wave length source is becoming more descriptive. We will use the source of the heat, and refer to the wave lengths in the description.

### Infrared Reflow, Area Source Emitters

Systems using area source emitters or panel emitters are of the medium to far IR (long wave lengths of 3.5 to 7.0um) type. There are a few different versions of panel emitter systems, but they basically operate in a similar way. (Some hot wire emitters also operate at these wave lengths).

The heat transfer results from a combination of radiation, convection and conduction. The radiation bandwidth is such that it heats the surrounding air, due to its high absorption capacity, and the heated air supports the heating of the parts and PCB. The radiation bandwidth is also adjusted to the absorption coefficient of the PCB material. This heats the PCB, and by conduction, heats the pad and the solder paste. The combination of these two results in about 60% of the total heat transfer. The balance of the heat transfer takes place through radiating energy directly to the materials. The radiation bandwidth is such that absorption of energy (and heat transfer) is not color selective. However, the time to heat different parts to the same temperature is dependent on their thermal capacities. This effect is minimized in this system by the three methods of heat transfer and by gradual heating and cooling rates. With the rates of the optimum profile (Figure 1), this effect is not noticeable.

**Positives**

- This is becoming the system of "choice", for various reasons. A large application history is being developed and the manufacturers are fine tuning the equipment to optimize the profile.
- Solder wicking occurs, but is preferential to the PCB (for additional explanation of wicking see Vapor Phase Systems). Wicking is minimized by the gradual heating ramps, and by the use of hot gas convection additions.
- Double sided reflow solder profiles can be developed (with care and proper fixturing).
- Precise and optimum solder profiles can be developed for each application. These are capable of computer storage, recollection, and verification.

**Criticisms**

These are points the users and competitors are making in discussing this reflow method. Methods for improving or negating these are suggested:

- Long Change Over Time
  The time to change from one product profile to another product profile may be significant. Optimum reduction of this time can be made by storing the product profile settings in a computerized control system (available with the equipment), and by scheduling product to be run in inverse order to total heat transfer needs (somewhat difficult to do). The use of hot gas (nitrogen or air) has greatly minimized the need for numerous profiles.
- Uneven Heating
  Because the component with the highest heat capacity determines the profile, some parts may see excessive heat or excessive heating ramps. Mass differences alone can account for 50°C differences in temperature during the preheat ramp. This effect can be minimized by using gradual heat ramps, with proper hold zones to insure even heating.
  Once again, the hot gas convection systems have minimized this effect by providing uniform heating across the entire board.
- Non-Shiny Solder Joints
  Shiny solder joints have long been recognized as representing a high quality solder joint. The shiny joints represent a fine grain microstructure. Non-shiny solder joints also indicate some oxidation of the metallic solder particles. The presence of oxygen (21%) in air promotes this oxidation.
  Another approach to minimizing oxidation is to reflow solder in a non-oxygen environment. Most IR equipment is supplied with the capability of a nitrogen or air environment. These atmospheres have other benefits, such as eliminating discolorization of the PCB, minimizing spreading of flux and minimizing flux char. Indications are - large improvement in repair rates can be expected. Costs of the nitrogen are minimal with usage about 1,000 cubic feet per hour. Extra venting is not needed. Care must be taken to insure a minimum moisture level is maintained. There are many technical papers on this subject.

**Infrared Reflow, Lamp Emitters**

These systems usually use quartz tungsten lamps as the IR energy source. They have typically been characterized as near IR (short wave lengths of .3 to 3um) systems.

The heat transfer in these systems is primarily through radiation and conduction. Some heat transfer through convection occurs as the wave length approaches the higher end.

In practice, the IR lamp reflow systems are set up similar to the other types of reflow solder systems. That is, they have various preheat zones and a soldering zone, followed by a cooling zone. In the preheat zone the lamp emitters are run at much less than rated wattage, and the biggest ratio of energy is transmitted in the 2 to 3um wave length. In this range the radiation heat transfer is supplemented by conduction and some convection. Actual percentages have been calculated, but experience indicates these vary somewhat dependent on the PCB assembly. As the PCB travels to higher preheat zones and eventally to the soldering zone, the wattage is increased on the lamps and the wave length decreases. This increases heat transfer through radiation, and decreases the convection component. The energy transferred by conduction through the PCB depends on the speed of PCB transfer. Much of the heat transfer is done from within the PCB, solder paste or component. The majority of heat transfer is through radiation in the near IR wave lengths. This heat transfer penetrates the material, thus heating from within.

**Positives**

- Near IR reflow systems have much faster response times than the middle IR systems. Profiles can be changed by changes in applied wattage, and since convection plays less of a part in heat transfer, the response time is much less.
- The heat transfer to the solder paste is done from within by the penetration of the nearer IR

wave length. This minimizes skinning over and oxidation of metal particles. The amount of penetration depends to a large degree on the absorptive properties of the target.

**Criticisms**

- Uneven Heating

  The selectivity of parts and PCB's are accentuated in this system. The PCB, or part with the highest heat capacity, sets the profile and the part with the lowest heat capacity can get very warm, very fast. The heating rates can be excessive and cause extensive damage. Some portions of the PCB can experience temperatures in excess of the glass transition temperature of the epoxy material, which leads to warp, twist, delamination, and discoloriza-tion. This selectivity is a function of color, mass, shape, shadowing and other factors.

  The near IR industry is attempting to correct some of the above by assisting inprofile selection and attempting to improve the ratio's of convection heating to radiation heating.

  The positive feature of fast response times also results in fast temperature rises. As seen earlier, a gradual temperature rise if more desired. Near IR rise times are advertised at 5°C / sec., while an optimum of 1 to 2°C / sec. is desired. Faster rise times increase the likelihood of solder balls, flux splatter, and thermal coefficient mismatches.

- Difficult Profiles

  This reflow system is the most difficult to profile and result in even temperature distribu-tion across the PCB, and to result in a smooth, gradual temperature transition.

- Long Change Over Time

  The response of the system to wattage change is fast, however, the preciseness and difficulty in profiling probably minimizes the amount of profile adjustment which can be done by changes in conveyor speed alone.

- Non-Shiny Joints

  The same considerations mentioned for area emitters apply to the lamp emitter systems.

- Profile Stability

  The aging of lamp emitters is of concern. Adjustment of profiles may be necessary as the lamps age. This should not be of major concern as periodic profiling is recommended regard-less of the system used.

## VAPOR PHASE REFLOW SOLDER PROCESS

Vapor phas reflow systems are disappearing from use. The ban on CFCs and other conditions have made them impractical. The following is pre-sented to describe this system and its advantages and disadvantages. Vapor phase reflow soldering is a direct contact heat transfer soldering method. Heat is transferred when a hot, saturated vapor condenses and releases its latent heat of vaporization. A vapor zone is gen-erated by heating a perfluorinated liquid to its boil-ing point. At that temperature, a saturated vapor forms over the surface of the boiling liquid, creating an essentially oxygen-free zone suitable for melting solder paste. The temperature of the vapor is the same as the boiling point of the liquid, and is con-trolled by the selection of the liquid. For example, with standard solders, such as SN63 or SN60, a liquid with a boiling point of +215°C is selected.

The vapor condenses on the colder parts of the part or PCB and gives off its latent heat until the temperature rises to the boiling point of the fluid. The heat transfer is independent of size, shape or color, etc. The speed of heating is again determined by the heat capacity of the termination or PCB. The process was originally developed in 1975 by the Engineering Research Center of Western Electric.

The original systems were batch systems and preheat was not a part of the system. Preheat in batch ovens to bake the flux was soon recognized as impor-tant. Batch systems are disappearing from use. This is being accelerated by the ban of CFCs.

Many improvement, including new chemical systems (PFPE's), in-line systems, preheat sections, etc. have been introduced which have improved applicability to SMT applications.

**Positives**

- Adaptability

  Vapor phase, especially in-line systems with good preheat, are also the system of choice. Experienced process engineers recognize the virtues and adaptability of this system. Large experience histories have proven its gentle profile very worthwhile.

- Even Heating

  Heat transfer is independent of parts and PCB characteristics other than thermal capacity (see discussion of solder wicking under criticisms). Board sizes, side-by-side loading, and board spacing are all not critical (other than for preheat considerations). This process can result in the lowest thermal gradients across the PCB.

- Shiny Solder Joints

  Shinier (finer grain size) solder joints are possible. With adequate preheat, the actual time the solder is molten can be reduced. Less chance of oxidation is likely as the solder is flowed in an oxygen free environment.

- Double Side Reflow

  Vapor phase reflow soldering has proven to be a good system to reflow parts on both sides of the PCB. The cleaning of the first side should be postponed, and the board cleaned after second side soldering. This minimizes moisture in the board during soldering.

- Temperature Control

  The maximum temperature is established by the chemistry of the boiling fluid. It is always controlled, and is always +215°C. Stresses to temperature-sensitive items are under control.

- Fast Change Over
  The only change that needs to be made in the profile is slight adjustments to the preheat cycle. Some very high density PCB assemblies may also require belt speed adjustments. These are very fast to make, and the system is most adaptable to applications which require many product changes and minimum downtime.
- Easier Cleaning
  The potential of overheating the flux is present with IR reflow systems. This charring and baking on of the flux makes it very difficult to clean. No charring will take place with the vapor phase system.

**Criticisms**

- Preheat
  Most of the criticisms for this method are related to the preheat portion of the process. It is important that the PCB, fluxes and parts be preheated evenly and gradually and that the **temperature be maintained at 150°C or above** until the assembly reaches the vapor. Items which fall under this area are:
  - Equipment
    Batch systems have little or no provision for preheat systems. Solder paste bakeout has traditionally been done in stand-alone ovens. The preheat can only be controlled by the speed of lowering the PCB through the secondary vapor blanket (47°C). This is marginal at best. Very fast thermal rise times and large temperature differences can occur. These detract from the quality of soldering and impact the quality of the parts. First generation in line vapor phase systems also had little provision for preheat. Some preheat was observed as the vapor zone tended to heat the incoming air in the throat of the system. In-line preheat systems, prior to the vapor phase unit, sometimes are used to preheat and bake the PCB flux and parts, but it is difficult to maintain the heat in the assembly. Newer in-line vapor phase systems have in-throat preheat sections which allow closer to optimum profiles. Both types of in-line systems have produced excellent results. (See Figure 2 for comparison of typical vapor phase profiles). The preheat zone for vapor phase should receive the same considerations as infrared systems.
  - Wicking
    In vapor phase reflow systems, the tendency is that the parts and the part terminations will reach soldering temperatures prior to the solder pad on the PCB. This will result in solder wicking with a preference to the part termination. The solder paste wicks up the surface of the part termination, and later reflows to the pad on the PCB. This can result in open solder joints if the amount of solder available is insufficient to wet to the pad or the pad solderability is not optimum. This can be overcome with adequate preheat. Experimental data indicates that with minimal preheat, the temperature of the termination reaches soldering temperature 16 seconds ahead of the pad. With preheat to raise the board to +115°C, this delta is reduced to 10 seconds. Additional preheat to +150°C will reduce this to near zero. Adding silver to the solder paste will also reduce wicking by increasing the surface tension of the molten solder. A dual alloy solder paste is available which delays the fluxing and wetting of the paste, and results in a 60-40 alloy after reflow. These extremes are not needed if proper preheat profiles are employed.
  - Voiding
    Greater amounts of voids have been reported in vapor phase reflow solder joints. These are contributed to by inadequate preheat times, resulting in skinning over of the solder paste and flux entrapment. Adequate preheat profiles will allow an orderly vaporization of the flux volatiles.
  - Intermetalics
    Greater amounts of intermetalic layers have been reported with vapor phase solders. These are results of extensive times above the solder reflow temperatures. If adequate preheat profiles are maintained, the time above reflow temperature can be reduced, and the intermetalic layers will be comparable to other reflow methods. In addition, a method of using plenum air to reduce time above liquidous has been developed. This reduces intermetallics further. (Forced air cooling is not recommended).
  - Tombstoning, Drawbridging,etc.
    Whatever you call it, the result is the same - one termination soldered. The biggest reasons for this lies in part solderability, pad design, and uneven paste deposition. Excessive wicking or uneven wetting can also contribute especially with very small parts. Fast rise time profiles, resulting in large amount of flux solvent activity, can also lead to this condition. Slow ramps and adequate profiles can all but eliminate this condition.
- Chemistry
  As the heat transfer media is a chemical that

also acts as a cleaning solvent, the media will become enriched with flux residues and other contaminants. This requires a cleaning process (usually distillation) and additional equipment to accomplish this. This equipment adds to the cost, but is readily available and not difficult to install and use. The perfluorinated liquids are not CFC's. However, the secondary vapor in the batch system is a CFC. However, the name alone has helped reduce the use of vapor phase systems.

• Operating Costs

One of the largest criticisms of the vapor phase reflow systems is its perceived operating costs. The initial bulk expense of the perfluorinated liquids are not cheap. Some items to consider are:

- Ventilation balance to insure a wind tunnel through the in-line system does not exist. Contact your machine supplier for the latest ventilation stack air rates. These have been greatly reduced from earlier recommendations.
- Balancing production runs to minimize start-ups and shutdowns.
- Proper use of condensate cooling coils to minimize drag out.
- Use of distillation equipment to reclaim the perfluorinated liquid.
- Recovery of aerosols in the ventilation system.
- Use of computerized controls to ramp heating, up and down, in anticipation of load demands.

Many of the advantages of vapor phase systems have been duplicated in IR/convection systems, and vapor phase systems have all but disappeared.

## TEMPERATURE PROFILES

The thread that runs through our discussion of reflow soldering (and for that matter wave soldering as well) is thermal profiles. On the surface this seems a simple matter. However, there are a few key points to be considered.

### Method

The optimum method of obtaining a thermal profile is through the use of thermocouples. Since multiple thermocouples are needed, a data collection system is needed. One system that appears to be used frequently is the M.O.L.E. profiler. This is Multi-Channel Occurrent Logger Evaluator available from Electronics Control Design. Other systems, such as the K.I.C. profiler are also also available. The type of thermocouple used and its mounting/location on the board is important. The thermocouple should be a low mass unit; 40 gauge works well, but is somewhat fragile. The user needs to determine what gives the proper response. To mount the thermocouple to the board it is recommended that it be lightly coated with epoxy. Excessive epoxy will delay the response and indicate a lower temperature. Mounting thermocouples in solder has long been recommended (including this author). The solder provides an electrical path reducing the voltage and indicates a lower temperature than it should. The solder material also provides additional intermetallic surfaces with the thermocouple and this provides a lower reading. The bottom line is the true profile could be higher by 30 - 50 degrees C than the indicated profile. Mounting the thermocouple in a through hole (via), or in a part, will attach the sensor to the thermal mass of either. This will suppress the thermal ramps and maximums, and not result in the proper indication.

The desired temperature control point is at the location of the solder fillet and part termination. The thermocouple can be attached either to the pad or the part termination. It is at this point that all the effects of various heat transfer (radiation, convection, conduction) are accumulated and is the point to be controlled.

### Longitudinal Profiles

This is the most commonly used profile. As the PCB travels the length of the machine, a thermocouple measures the temperature variation. To measure this profile correctly, the thermal response of the system with one board, and with multiple boards at production spacing, should be measured.

### Lateral Profiles

This is the profile across the width of the PCB as it travels through the process. This measures end effects and shadowing of the heat transfer. Most reflow systems have a flat response in the center of the transfer conveyor with some fall off at the edges. This fall off will control the total width of side-by-side PCB soldering. Hot gas (nitrogen or air) greatly minimizes this affect.

### Other Considerations

*Profile development is a key factor.*

From our previous discussions, we know that all reflow soldering profiles of individual parts will most likely be different. Choosing numerous high thermal capacity and low thermal capacity parts, scattered in leading edge, trailing edge, middle and sides of the PCB will give the various representative profiles. A profile that results in nearly uniform elemental profiles will be the best. During this development, the process engineer may also wish to find the temperature being reached in the PCB, across the surface of the PCB, in some key components (such as IC packages), etc. This is the time to evaluate the profile and its affects. Once the profile is established, a few key points for process control should be established, and these points measured periodically to insure the process is being maintained. Some equipment types will age with use (lamps, combustion product buildup on emitter surfaces, etc.) and the profile may change. This process lends itself to statistical process control, comparing number of repairs required with profile drift. Non-contact temperature measuring equipment is available. These are useful as a process monitoring tool, and as a means of insuring the process remains in control. These should be checked periodically by using the thermocouple method.

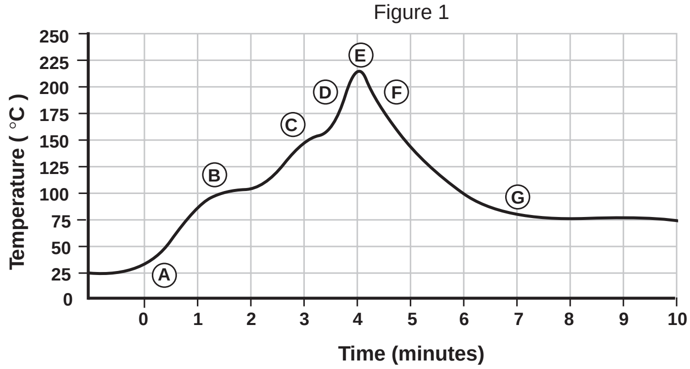
Time spent optimizing profile measurement techniques and understanding the measurement readings will be rewarding.

## SUMMARY

The ideal reflow solder profile exists. The conscientious SMT manufacturer will work to approach the ideal profile for each product. Using the ideal profile will result in the maximum attainment of all goals, and a nearly trouble-free process. The solder process used depends on its ability to consistently deliver the ideal profile for each product, and the amount of effort you want to donate to improving the system you choose. ***It is worth the effort!!!***

**Reflow Solder Profile**
**( Typical )**

Figure 1




## REFERENCES


1. "Infrared Reflow Soldering Re-examined" Philip Zarrow - SMTA '88 Proceedings
2. "Solder Pastes", Alpha Metals
3. "Fluorinert Liquids, Vapor Phase Condensation Heating Reference" 3M, Industrial Chemical Products Division
4. "Temperature Profiles: The Key for Proper Surface Mount Assembly Process Control" John Maxwell - CARTS '88 Proceedings
5. "Desiccant Packaging of Moisture Sensitive Electronic Devices" Joseph M. Williams - SMART IV Proceedings
6. "Moisture Induced Cracking in Plastic Encapsulated Surface Mount Components during Solder Reflow Process" R. Lin, E. Blackshear, P. Serisky
7. "Moisture Effects on Susceptibility to Package Cracking in Plastic Surface Mount Components" S.F. Gordon, W. Huffman, K. Yee, et al
8. "Selectron Criteria for Infrared ReflowSoldering Systems" P. Zarrow - SMART IV Proceedings
9. "Concerns in Wave and Infrared Soldering of Surface Mount Assemblies" Don Elliott, SMTA '88 Proceedings
10. "Moisture Induced Plastic IC Package Cracking" Surface Mount Council, Package Cracking Task Force
11. "Wave Solder Process for Surface Mount Applications" KEMET F-2101 J. Bergenthal
12. "Surface Mount Mounting Pad Dimensions and Considerations" KEMET F-2100 J. Bergenthal
13. "Soldering in Electronics" R.J. Klein Wassink; Electrochemical Publications
14. "Vapor Phase curing of Polymer Thick Film Resistors and Conductors" A. Schultheiss, R.S. Briggs, Dr. P. Srinvasan; Expo SMT '88 Proceedings
15. "What's New In Vapor Phase" D. Linmon; Expo SMT '88 Proceedings
16. "Processing Considerations for Surface Mount Connectors" M. Jackson; Expo SMT '88 Proceedings
17. "Defects and Failure Phenomena of Solder Joints" Dr. J. Hwang; Expo SMT '88 Proceedings


**Vapor Phase Solder Profile**
**( Typical )**

Figure 2