The Pin-in-Paste (or AART) Process for Odd Form and Through Hole Printed Circuit Boards

Jay B. Hinerman[1], K. Srihari[2], Ph.D. & George R. Westby[3]
[1]DEK Inc
8 Bartles Corner Road, Flemington, New Jersey 08822, USA
[2]Professor, Department of Systems Science and Industrial Engineering
State University of New York, Binghamton, New York 13902, USA.
[3]Director - SMT Laboratory
Universal Instruments Corporation, Binghamton, NY 13902-0825.

ABSTRACT

Considerable interest exists in the process known as the pin-in-paste, or the Alternative Assembly and Reflow Technology (AART) process. The AART process allows for the simultaneous reflow of both odd-form and through hole devices as well as surface mount components. This process has several advantages over the *typical* mixed technology process sequence that includes wave soldering and/or hand soldering, often in addition to reflow soldering. The advantages include, but are not limited to, the following: multiple operations can be condensed into one comprehensive process; fewer pieces of equipment, materials, and personnel are necessary; a reduction in cost and cycle time can be realized; the high defect rate associated with wave soldering may be reduced; and it eliminates one or more thermal process steps thereby improving solderability, electronic component reliability, etc.

Numerous issues must be considered before implementing a high-yield AART process. A high-yield process requires a scientific understanding and a systematic examination of materials, design, and process related factors on proper interconnection formation and assembly reliability. For example, solder paste selection and component resin choice are important material characteristics. Component body design and stencil aperture design are critical parameters in the design area. Finally, the reflow profile and the placement method are important process parameters that must be understood and controlled. Many important aspects and issues for a successful AART process have been identified and examined in this paper.

KEYWORDS

AART, Reflow Soldering, Printed Circuit Boards.

INTRODUCTION

The manufacturing process steps used to assemble a Printed Circuit Board (PCB) depend primarily on the specific components used in the assembly. As more emphasis is put on smaller size, increased functionality, and increased component density, many single and double sided boards contain primarily Surface Mount Components (SMCs). However, due to the inherent strength, reliability, and availability, through hole devices are still chosen over SMCs, especially for edge connectors. Coupled with the fact that current automatic placement equipment can place through hole / odd form devices, a strong case can be made for these components. The drawback of choosing a through hole device on a largely surface mount board is the high cost-per-joint of additional processing steps such as wave soldering, hand soldering, or other selective soldering methods. For these assemblies, providing a robust process for simultaneously reflow soldering both through hole and surface mount components in a single comprehensive process is pivotal.

The Alternative Assembly and Reflow Technology (AART) process allows for the concurrent reflow of both SMCs and through hole devices in a single step. The cost savings involved in reducing additional process steps and material is only one benefit among many. The manufacturing process steps involved depend on the specific components used in the assembly. For example, a computer motherboard contains a variety of SMCs, which constitute the majority of the components used, and a limited mix of through hole devices: connectors, discrete components, switches, jacks, etc. Solder paste stencil printing and reflow are currently used to attach the SMCs to the PCB. A similar process may be used to form the interconnections for the through hole and odd form devices as well. In many cases, the use of AART processes may eliminate the need for subsequent soldering operations.

The focus of this paper is the identification of key material, design, and process related factors relating to the AART process. A flexible computer program has been developed that calculates the solder paste volume required to form a quality solder joint, based upon paste characteristics, printing parameters, board related factors, etc. Methods of solder deposition are discussed as well as options for component placement. In short, the key issues that need to be considered to successfully implement an AART process are discussed in detail.

OBJECTIVE

The objective of this research effort was to identify the factors that have a significant impact on the quality of the AART process as a whole. These factors were then categorized into either material, design, or process-related factors. The objective is to transfer knowledge over a broad range of topics, illustrating the key issues that must be well characterized before a high-yield AART process is undertaken.

FACTORS CONSIDERED

Given below is a listing of the primary factors that must be well understood and characterized in order to implement a high-yield AART process. Each of these key issues is discussed below in detail.

- Commonly accepted through hole solder joint criteria
- Solder paste volume calculation given an ideal solder joint.
- Solder paste related factors that can affect the volume required - AART volume model.
- Solder deposition methods including stencil printing, automatic dispensing, and solder preforms.
- Component design and material issues.
- Stencil thickness discussion, aperture design guidelines, hole fill, and overprinting.
- Placement issues and options.
- Reflow profile development and recommendations.
- The need for industry-wide quality standards and methods of evaluation.

Commonly Accepted Solder Joint Quality Criteria

Commonly accepted solder joint quality standards include ANSI/J-STD-001B [1] (October 1996) and IPC-A-610. Depending on the classification (class 1, 2 or 3), minimum acceptable conditions are given for visual inspection. A company may choose to use such standards as a basis for quality evaluation, or make modifications to accommodate their process. For this research, the model solder joint is a completely filled Plated Through Hole (PTH) with a fillet on top and bottom of the PCB. This model is used when calculating the necessary volume of solder paste required. It is important to decide on quality metrics early in the implementation of this process.

Solder Paste Volume Calculation Given a Quality Solder Joint

The starting point for a solder paste volume calculation is to begin with the ideal solid metal solder joint. As described above, the ideal is considered as a completely filled PTH with a fillet on the top and bottom of the PCB. The exact fillet shape cannot be predicted exactly due to varying metallurgies, lead conditions, reflow characteristics, etc. Yet, a fillet as described by the radius of a circle is an adequate and simple approximation. This has been described in previous research [2, 3]. The fillet area is then rotated to determine the volume of the solid fillet. This solid volume is then multiplied by two for each fillet (top and bottom) and added to the solid volume in the PTH (minus the lead volume). The volume of solid metal for a quality solder joint is thus computed. The solder paste volume required is a function of the alloy type, the flux density, and the weight percent metal in the solder paste. The next section goes into detail on computing proper solder paste volume once a solid metal volume is computed.

Solder Paste Related Factors That Can Affect The Volume Required - The AART Volume Model

Solder paste, in a simple sense, is a combination of metallic spheres encased in a flux binder. Solder paste characteristics are modified by adding tackifiers, rheology enhancers, altering flux chemistries, etc. One main specification for solder paste is the weight percent metal. For stencil printing, a 90 percent by weight alloy is often specified, for viscosity reasons among others. For a typical eutectic 63Sn/37Pb alloy with a flux density of 1g/cc and a 90% metal by weight content, 1.92 times more solder paste must be deposited than the computed solid volume. The *volume* fraction of metal in the solder paste is 52%, and upon reflow, nearly half the paste volume is lost as flux vapor and residue.

If this reducing factor is compared with a typical dispensing grade paste (85% metal by weight) of the same composition and flux type, 2.46 times more solder paste than the solid solder computed must be dispensed. This increase in volume, as compared to the solder paste that is stencil printed, is due to the tradeoff that must be made to dispense solder paste: reduce the metal content in favor of flux to increase the lubrication power of the paste. The flexibility in the volume dispensed by an automatic syringe dispenser comes at the cost of an increased solder paste deposit volume and subsequent residue.

The AART volume model prompts the user through a pull down menu for alloy type, flux density and weight percent metal in the solder paste. The volume percent metal, density, and reducing factor of the solder paste are all automatically calculated. The model also has a section specifically for the stencil printing process and prompts the user for stencil thickness, print pressure, print speed, squeegee angle, etc. These parameters are all used, along with specific hole size and solder paste characteristics, to predict how much solder paste will fill the PTH. The importance of solder paste hole fill will be described later in the paper.

Solder Deposition Methods Including Stencil Printing, Automatic Dispensing, And Solder Preforms

Stencil printing is the preferred method to deliver solder paste to the PCB for the AART process. The thickness of the stencil is a critical parameter, since the solder deposit is a function of the aperture area and stencil thickness. This will be discussed in detail in the stencil design section of this paper. Steel squeegees are used to limit scooping of the relatively large stencil apertures and to enhance hole fill. Proper board support is essential for a repeatable process. The board support may be custom designed with routed holes or channels to accommodate more than 100% hole fill if necessary. Magnetic pillars are also an option. In any case, for a given solder paste, the amount of paste that fills the PTH must be well characterized. Paste should not squeeze out of the holes and contaminate the board supports and subsequent assemblies. This can easily occur when trying to solder the retention features on components. The retention holes are quite large and can easily fill to well over 100% of the board thickness. An alternate stencil design for soldering retention features is given in the stencil design section. The AART volume model, discussed above, is designed to provide process variables for the prediction and control of hole fill.

There are several process sequences that may be employed, depending on the specific assembly being produced. The easiest and most cost effective process is to design one stencil to accommodate both SMCs and odd form/through hole devices. For through hole devices with two rows and almost no space constraints on the location and size of the solder paste deposit, a range of stencil thicknesses could be used. For example, the stencil designer could choose the thickness that would be most appropriate for the SMCs on the board.

The solder paste volume necessary is printed into the PTH and the remainder is overprinted onto the surface of the PCB. For SMCs that require a very thin stencil combined with multi-row odd form/through hole devices with a large volume requirement, a double stencil printing process may be necessary. Two in-line stencil printers must be used for this process. The first stencil prints solder paste onto the surface mount pads. The second (thicker) stencil is relieved on the bottom side to accommodate the print pattern for the SMCs. A third option is a step stencil with a relatively thicker area for the through hole devices. The process chosen varies with the technology mix on the specific assembly. Finally, a step soldering process has been implemented for the intrusive reflow process [4]. A combination of Sn63/Pb37 solder paste and the lower melting point (with a melting point of 165°C) Sn43/Pb43/Bi14 was used. The result quoted was less thermal shock to parts and lower operational costs while achieving adequate solder joint strength.

Automatic solder paste dispensing has been used successfully to deposit the correct volume of solder paste to through hole and odd form components. It offers flexibility and the ability to deposit large volumes of solder paste that may not be feasible through stencil printing. The speed of current automatic dispensing equipment makes it a very attractive option for many applications. This may become especially important when processing boards thicker than the standard 0.062" PCB thickness, for example backplanes. Automatic dispensing of solder paste is often used for niche and prototype applications. This technique can also apply solder paste to a board that is already partially populated. If dispensing over a bare PTH [5], use of a nozzle that is *slightly larger* that the diameter of the PTH is recommended. This forces the paste against the walls of the PTH during dispensing and extrudes the material slightly from the bottom of the PTH. The component should then be inserted *opposite* to the direction the solder paste was dispensed. If a nozzle smaller than the diameter of the PTH is used, the paste tends to eject from the hole and critical paste volume is lost. An alternate process sequence is to insert the component and dispense solder paste over the leads. The nozzle is designed to curve around the component lead. Solder paste is forced around the lead and into the PTH. Both positive displacement and Archimedes style pumps are available for dispensing solder paste. These two pumps are active research areas in the AART study.

Solder preforms offer yet another option for providing necessary solder volume to form a quality interconnection. Companies offer components with solder-bearing-fluxed leads. Flux-filled solder clips reside on the leads. The process sequence is to place the component and reflow solder. Upon reflow, the clip melts and the solder flows into the PTH and wets to the component lead. These components are currently being evaluated as an alternative to the classical pin-in-paste processes. The success of this technology would offer a very attractive option where space is critical, since the metal is already in solid form. In addition, the pre-calculated solder volume offers a secondary benefit to those who do not have a suitable characterization of the degree of solder paste hole fill.

Component Design And Material Issues

Since the through hole and odd form components are going to pass through a reflow profile, they must be able to withstand the temperature excursion. The component should be made of a resin that does not degrade under a 60-90 second period above 183° C (220°C for 60 seconds to be safe) and a peak temperature of 240° C. UL 94 V-O flammability and other plastics industry resin standards help manufacturers produce reliable components. Component manufacturers also have demanding standards for warp, dimensional stability, shrinkage and dielectric properties [6].

Temperatures at which the components change their dimensions are never well defined. The orientation of the component in the reflow oven should also be taken into consideration, especially for long, thin components such as memory modules. Given below is a listing of resins that can withstand the reflow temperatures [3, 6].

- Liquid crystal polymer (LCP)- relatively expensive, yet retains tight tolerances in thin-wall moldings and has good thin wall stiffness (important for applications such as memory modules).
- Polyphenylene sulfide (PPS)- has good flow characteristics.
- Polydimethylene cyclohexylene terephtthalate (PCT).
- Polyphthalamide (PPA).

A great deal of component manufacturers offer through hole components in reflow compatible resins. This high-temperature compatibility is one of the basic necessities of the process for obvious cosmetic and reliability concerns.

The next component requirement is adequate and correctly positioned standoffs from the surface of the PCB. A standoff allows the molten solder paste to flow freely from its printed position to the PTH. The standoff should not sit in and/or block the print deposit, nor should any other part of the component. Solder balling and/or bridging are secondary defects caused by incorrect component body designs. The component design must be taken into account when designing the stencil apertures. This requirement is extremely important and is another basic requirement for a successful process, backed by experimental data and experience.

Some components, such as mini-DIN connectors (which a PC mouse plugs into), are shielded. The metallic shielding is a solderable surface. If the solder paste deposit contacts this material, there is a chance that the solder will wet to the component body rather than the PTH and lead. It must be noted that an over-printed paste deposit, upon reflow, gets shorter and *taller* as it pulls back to the PTH. This increase in height can cause the paste deposit to come into contact with the solderable shielding. Again, the components design must be taken into account when deciding where to place paste deposits.

For double-sided reflow, and for dispensing solder paste over component leads, the components should have retention features to firmly hold the component in place during processing. It has been found, however, that retention features that are not absolutely necessary for processing should be drilled oversize if an automatic insertion machine is used.

Component lead length beyond the bottom of the PCB should be as short as quality standards will allow. This distance should not be more that 0.040" to 0.050". If the leads reveal length is too large, the solder paste on the end of the pin will fall off or form a ball on the end of the lead upon reflow. The best scenario would be close to zero protrusion length. In this case, no paste would be lost from the plated through hole, and volume calculations would be near perfect. It must be noted that paste viscosity drops very fast with an increase in temperature. As a note, if there is a glob of paste on the end of the pin, chances are it will drop off in the preheat zone, and that volume will be lost. One option, if lead reveal length cannot be controlled, is to build a safety factor into the volume model.

Stencil Thickness Design, Aperture Design Guidelines, Holefill And Overprinting

Careful consideration must be given before a stencil thickness is chosen. Typically, stencil thicknesses of 0.006" or 0.008" are used, since these are a drop-in for most surface mount processes. It must be realized that the area of an aperture is a function of the component pitch, number of rows, and deposit-to-deposit spacing. A two-row component with a lead pitch of 0.100", such as a 25-pin DSUB connector, is easily processed with nearly any reasonable stencil thickness. Four row, 2 mm pitch memory modules begin to become more of a challenge as far as stencil thickness is concerned. In these cases, thicker stencils or other processes that have been previously discussed may have to be implemented, or quality standards altered.

The placement of the aperture over the PTH is very important with respect to the variability in hole fill. If the apertures are offset to the sides of the PTH, variability in hole fill of up to 20% has been observed. It is important to take the squeegees print direction into account when designing stencil apertures. This effect is more pronounced for smaller diameter PTHs and is attributed to how the apertures are unevenly spaced over the two rows of PTHs. The solder paste fills the apertures differently and thus variably in adjacent PTHs. This effect is eliminated if the print direction is rotated 90°. In this print direction, the apertures are now evenly spaced over the PTHs.

The deposit-to-deposit spacing is important to retain separate solder paste deposits upon reflow and to avoid solder robbing. Separate deposits are another basic process requirement. If the paste deposits run together, the hottest spot will rob solder from the other areas. This will not occur for separate deposits. Solder paste tends to slump, or spread out when it is heated and the viscosity is lowered. The amount of slump is largely a function of the specific solder paste. Statistically designed experiments can be performed to correlate deposit area, height, and solder paste formulation to deposit-to-deposit spacing. A modified profile will be developed to bring the paste patterns up to pre-reflow temperature when it exits the reflow oven. The patterns will be examined for degree of slump and stencil aperture. Stencil spacing guidelines will then be developed.

Depending on several variables included in the AART volume model, PTHs will fill to a varying degree with solder paste if the stencil apertures are located over the holes. The balance of solder volume (amount necessary - amount in PTH) must be overprinted onto the surface of the PCB. Solder paste deposits in excess of 0.350" long have been overprinted on the board surface and successfully reflowed to form interconnections.

The importance of understanding solder paste hole fill is best described by a plot of solder volume deposited versus hole fill percentage. The example shows that the solder volume can go from deficit to excess, depending on the degree of solder paste hole fill. The calculations were made for one component lead size and two hole sizes, 0.042 and 0.046, respectively. The overprint aperture in this example is designed to provide 65% of the required paste volume, which is common for multi-row connectors. It can be seen from the figure that around 50% hole fill in this case is the optimum. If the PTHs fill less than this, the solder joints may be smaller than ideal. If the holes fill more than this amount, the interconnections may be excessive.

Hole fill is especially important when overprint area is limited, or a thin stencil is being used. Multi-row components impose constraints on the overprint aperture area. What further reduces this area is the row-to-row spacing that must be included as well. The moral is that it is very important to characterize hole fill.

Very large holes, such as plated retention holes, should not have complete stencil openings over their entire diameter. If retention features must be soldered, an exploded pie shape should be used. The circular area should be broken into four pie-shaped pieces with the tips over the edge of the hole. Or, if space permits, the retention hole should be completely blocked and the entire paste deposit overprinted.

Placement Options And Issues

One reason for the resurgence of interest in through hole technology is the ability of automatic placement equipment to place odd form and through hole components. Components can be shipped in tubes, reels, trays, etc. and the feeders are placed directly on the placement machines. The machines use either mechanical search routines or visual inspection for placement location. Visual methods are preferred for solder paste-printed PCBs, since mechanical search routines can disturb paste deposits. Automatic placement offers benefits such as accuracy, repeatability, and speed. The variety of components that can be automatically placed is increasing daily.

Hand placement is a second option for component placement. Component artwork on the PCB, as well as locating features such as retention clips, may help in alignment. These become increasingly important for components with high pin counts. Two benefits of hand placement are zero setup time and no setup cost. The drawbacks of hand placement include a lack of speed and variable accuracy.

Reflow Profile Development And Recommendations

The reflow oven used must be able to provide adequate heat (temperature) over the entire assembly at all lead locations. Many of the odd form/through hole devices are tall and/or have a high thermal mass when compared to other SMCs populating an assembly. It is generally accepted that a forced convection system is superior to IR for these AART applications. Separate top and bottom heating control can also be a benefit to lower the ΔT seen on a PCB assembly. On one computer motherboard with a tall stacked 25-pin DSUB connector (1.5" in height), the component body temperature was unacceptably high. This problem was solved by increasing the bottom side temperature and lowering the topside temperature. The time above liquidous should be long enough to allow the flux to volatilize from the PTHs, possibly longer than a standard profile. Cross-sectional analysis may be important to confirm if the reflow profile is correct. The peak temperatures as well as the thermal gradient on an assembly must be carefully measured and controlled.

The Need For Industry-wide Quality Standards And Methods of Evaluation

One of the key issues that no one person can answer is the following: what constitutes an acceptable solder joint? In what case(s) should an AART interconnection be reworked? (which can, by the way, cause more harm than good). There must be an industry-wide consensus, backed by joint strength and reliability data for this process. What percentage of deviation from the ideal solder joint is acceptable Current designed experimentation (in progress) will investigate the following situations to correlate strength and reliability for AART solder joints versus wave soldered joints.

For 20%, 60%, and 100% of ideal solder volume, the shape, strength, and reliability of the resulting interconnection will be assessed. Accelerated-life testing will be performed for assemblies built with no clean, water soluble, and wave soldered interconnections. Weibull analysis will be utilized to compare lifetimes. Finite element analysis will be used to study various lead sizes and solder volumes. Visual inspection criteria will be developed to help in the identification of good solder joints. The intention of these studies will be to add to the knowledge and experience database in order to create rational and statistically valid quality criteria. It would be useful for others to do the same in order to create industry-wide quality standards for AART interconnections.

SUMMARY

This paper identifies the critical issues that need to be considered prior to and during the implementation of AART processes. The AART process sequence offers certain definite advantages. These include the ability to simultaneously reflow solder THCs and SMCs, enhance throughput due a shorter process sequence (no hand and/or wave soldering), reduced floor space needs, etc. The process engineer needs to consider a variety of issues that span materials, process, design, and reliability. Materials related factors could pertain to the board, component, and the solder paste. Process related issues would relate to solder paste deposition through printing and/or dispensing, component insertion, and reflow soldering. Design related issues include stencil design and component characteristics. Reliability can be assessed through traditional mechanisms such as accelerated life cycle testing. The AART process can be applied to a wide array of applications that cover automotive, communication, consumer electronics, computers, etc.

REFERENCES

[1] ANSI/J-STD-001B, Requirements for Soldered Electrical and Electronic Assemblies, October 1996.

[2] Rupert, M., Determining Solder Volume for Surface-Mount-Process Compatible Through Hole Connectors, AMP Incorporated Document, Harrisburg, Pennsylvania.

[3] Gervascio, T., Developing the Paste-in-Hole Process, Proceedings- Surface Mount International Conference, San Jose, California, August 1994, pp. 333-340.

[4] Seelig, K., & Peek, J., Step Soldering Aids Intrusive Reflow, Surface Mount Technology Magazine, September 1996, pp. 88, 92.

[5] Hinerman, J. & Srihari, K. An Initial Study of Solder Paste Dispensing and Reflow for SMT-Compatible Through Hole Headers, Technical Report, Universal Instruments Corporation, Binghamton, New York, June 1997.

[6] McChesney, C., Bowers, D., & Kaslusky, A., Resin Selection for High Performance Connectors, Surface Mount Technology Magazine, May 1996, pp. 70-74.