Comprehensive Guidelines for the Implementation of the AART Process

The Alternative Assembly and Reflow Technology (AART) process allows for the simultaneous reflow of both odd-form and through hole devices as well as surface mount components. It offers several advantages over the typical mixed technology process sequence that includes wave soldering and/or hand soldering, often in addition to reflow soldering. Implementing a high-yield AART process requires a scientific understanding and a systematic examination of materials, design, and process related factors on the formation of a 'proper' solder joint and on 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. This paper examines the entire AART process. It is designed to provide a process engineer with some of the knowledge necessary to implement a high-yield AART process. The AART principles illustrated in this paper can be designed into a product at the product design phase, or used as a 'drop-in' solution to an existing PCB assembly.

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 for certain applications, 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, it is pivotal to provide a robust process for simultaneously reflow soldering both through hole and surface mount components in a single comprehensive process.

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. The key material, design, and process related factors relating to the AART process are identified and discussed in this paper. This paper is based on extensive research into the AART process conducted at the Surface Mount Technology (SMT) laboratory at Universal Instruments Corporation. These research efforts encompassed the topics that related to the materials, design, processes, modeling, and reliability aspects of the AART process. This information on the AART process has been utilized to design and develop intelligent agents that can assist with solder volume calculations, solder paste hole fill predictions, printing parameters, and stencil aperture design. A stand alone cost model and a tool to evaluate possible failure modes have also been developed. These intelligent agents are not discussed in this paper.

Issues and Factors to be Considered

Prior to the implementation of the AART process in a manufacturing environment, several factors must be studied, understood, and characterized [3]. These are:

- Commonly accepted solder joint quality criteria
- Calculation of the required solder paste volume.
- Solder paste related factors that can affect the volume required.
- Solder deposition methods including stencil printing, automatic dispensing, and solder preforms.
- Component design and material issues.
- Stencil considerations, solder paste holefill, and overprinting.
- Placement - options and issues.
- Reflow profile development and recommendations.
- Strength of AART solder joints.
- Solder joint inspection and quality criteria.

Commonly Accepted Solder Joint Quality Criteria

Commonly accepted solder joint quality standards include ANSI/J-STD-001B (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 (discussed in this paper), the model solder joint is a completely filled Plated Through Hole (PTH) with a fillet on top and bottom of the PCB (*Figure 1*). 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.

Calculation of the Required Solder Paste Volume

Computation of the required solder paste volume needs to begin with the ideal solid metal solder joint. The ideal solder joint is 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. However, a fillet as described by the radius of a circle is an adequate and simple approximation. This has been described in previous research [1]. The

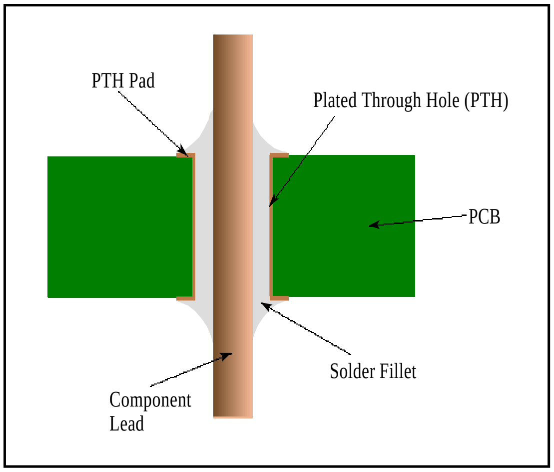

Figure 1

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.

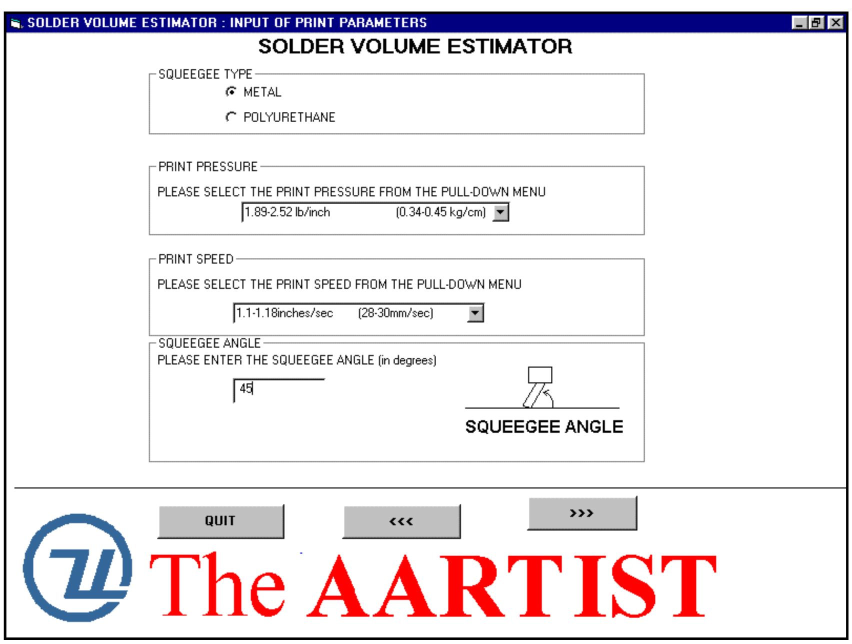

Figure 2

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

Solder paste, in a simple sense, is a combination of metallic spheres encased in a flux binder. Methods that are used to modify solder paste characteristics include adding tackifiers and rheology enhancers, and altering flux chemistries. A primary 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.

An intelligent agent that can help a user estimate the required solder volume was designed and developed. This AART volume model is an integral part of the AARTIST, a comprehensive decision support system for the AART process. This agent 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. This software-based 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 (*Figure 2*). These parameters are all used, along with specific hole size and solder paste characteristics, to predict how much solder paste will fill the PTH (or can be deposited in the PTH). This is called 'solder paste hole fill' (*Figure 3*) and its importance is described later in this paper.

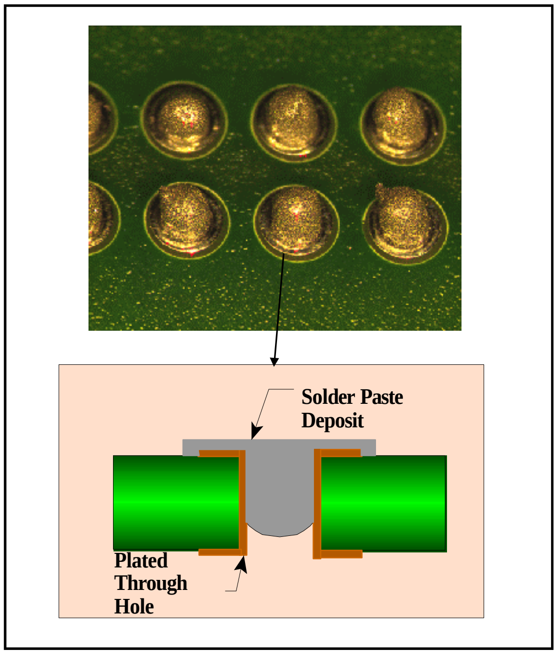

Figure 3

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. 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 necessary solder paste volume is printed into the PTH and the remainder is overprinted onto the surface of the PCB. Another 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.

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 than 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 (*Figure 4*) are available for dispensing solder paste. These two pumps were actively studied in this research effort.

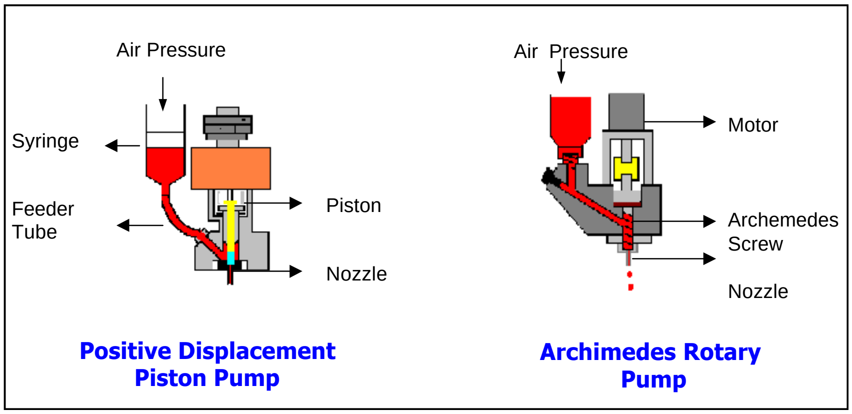

Figure 4

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 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 that are to be soldered onto a PCB using the AART process are going to pass through a reflow profile, they must be able to withstand the temperature excursion [2,4]. The component should be made of a resin that does not degrade when subjected to a temperature greater than 183º C for a 60-90 second period (220°C for 60 seconds to be safe) and a peak temperature of 240º 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. Several component manufacturers offer through hole components in reflow compatible resins. This high-temperature compatibility is one of the basic necessities of the process for both reliability and cosmetic concerns.

The next component requirement is adequate and correctly positioned 'standoffs' from the surface of the PCB [4]. 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. Solderballing 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. This 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 component's 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 [4].

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 lead's reveal length is excessive, 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 to have a reveal length that is close to zero. 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 Considerations, Solder Paste Holefill, and Overprinting

The thickness of the stencil must be carefully selected. Typically, stencil thicknesses between of 0.005” and 0.008” are used since these are a ‘drop-in’ for most surface mount processes. The area of an aperture is a function of the component pitch, number of rows, and the 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 memory modules with a lead pitch of 0.080” begin to become more of a challenge as far as the stencil thickness is concerned.

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 print direction of the squeegee 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 90º. In this print direction, the apertures are now evenly spaced over the PTHs.

The spacing between deposits 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 should be performed to correlate deposit area, height, and solder paste formulation to the spacing between deposits.

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 with a length in excess of 0.350” have been overprinted on the board surface and successfully reflowed to form interconnections.

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 key 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 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 another 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 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 infrared for these AART applications. Separate top and bottom heating control can also be a benefit to lower the temperature differential 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. Increasing the bottom side temperature and lowering the topside temperature solved this problem. 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.

Strength of AART Solder Joints

The segment of this research effort that compared the AART solder joint's strength versus those produced by wave soldering had two objectives [5]. First, it benchmarked the strength of solder joints formed via the AART process against the traditional wave soldered joint. Second, it characterized the strength of solder joints that result when a less than 'optimal' amount of solder was used. The failure mechanisms that were observed were cataloged for different solder volumes and solder paste types. It was found that the strength of the joint decreases when the solder paste volume is less than 80% of the 'ideal' volume. A transition in the failure mechanism was also observed as the solder volume decreased. Furthermore, this study showed that the solder joint strength and the failure mechanism did not depend upon the type of solder paste (water soluble or no clean). This investigation reiterates the influence of the solder volume on the mechanical strength of the soldered joints and the predominant failure modes. It may also be used to support modified acceptance criteria for situations where it is impossible to deposit the 'optimal' volume of solder. Truly, it is clear that the bulk of solder from a wave soldering process does not further enhance the strength of a near 'ideal' AART solder joint, when PTH barrel failure is considered. Therefore, the acceptance criteria for AART solder joints should be carefully considered before the process is undertaken. The data produced are valuable for the quality and process engineer to make sound engineering decisions, not simply failing a solder joint because it does not 'look right' by common industry standards.

Solder Joint Inspection and Quality Criteria

Quality decisions for AART solder joints are usually based on the individual manufacturer's requirement [3]. There are, however, joint industry standards for typical through-hole component processes that may be used unmodified or slightly revised. These standards include "Requirements for Soldered Electrical and Electronic Assemblies" [ANSI/J-STD-001B, 1996] and "Acceptability of Electronic Assemblies" [IPC-A-610, 1996]. The J-STD-001 describes final product requirements for PCB assemblies depicting minimum end product acceptable characteristics, as well as test methods for evaluation. 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. It is important for a company to decide on quality metrics early in the implementation of this process.

It has also been shown that the human inspection of solder joints can be very subjective [4], especially if the inspectors are only familiar with typical bulky solder joints produced in a wave or hand soldering process. By design, AART solder joints are produced to have small fillets on top and bottom of the PCB. Some retraining of quality inspectors may be necessary to familiarize inspectors with proper AART solder joints.

Summary/Conclusion

The vast majority of current PCB designs that contain through hole components are compatible with the AART process. In many instances, the same parameters that are used to assemble the SMCs on a PCB may be used for the through hole devices as well. The goal is to provide a 'drop in' solution to assemble the through hole devices in the same step as the surface mount components. The AART process can be applied to a wide array of applications that cover automotive, communication, consumer electronics, computers, etc.

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 several 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 several issues prior to the implementation of the AART process. They include 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. The strength and reliability of AART solder joints was assessed and compared favorably to those obtained through wave soldering. This research effort proved that the AART process was more repeatable than wave soldering while concurrently producing quality solder joints.

References

1. *Gervascio, T., Developing the Paste-in-Hole Process, Proceedings - Surface Mount International Conference, San Jose, California, August 1994, pp. 333-340.*
2. *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.*
3. *Hinerman, J.B., Comprehensive Guidelines for the Alternative Assembly and Reflow Technology process, Masters Thesis, State University of New York at Binghamton, New York, February 1998.*
4. *McChesney, C., Bowers, D., & Kaslusky, A., Resin Selection for High Performance Connectors, Surface Mount Technology, May 1996, pp. 70-74.*
5. *Seeniraj, R., Study of Strength, Reliability and Other Issues Concerning AART Solder Joints, Masters Thesis, State University of New York at Binghamton, May 1998.*

Authors

Jay B. Hinerman[1], K. Srihari, Ph.D. [2], & George R. Westby[3]

[1] *Senior Applications Engineer*
DEK, 8 Bartles Corner Road,
Flemington, New Jersey 08822.

[2] *Professor - Department of Systems Science and Industrial Engineering*
State University of New York,
Binghamton, New York 13902.

[3] *Director - Surface Mount Technology Laboratory*
Universal Instruments Corporation,
Binghamton, New York 13902-0825.