A Flexible Automated Solder-Paste Dispensing System

Jon R. Plummer, Barry M. Lunt
Brigham Young University

Abstract
This paper describes a flexible automated solder-paste dispensing system which does away with the need for a dispensing screen, thus avoiding its cost and changeover time. The system described is composed of a 4-axis robot, a solder-paste dispensing tool and controller, a conveyor system, and a processing computer. The processing computer inputs the Gerber file describing the PCB and creates the program to drive the robot with the solder-paste dispensing tool. The PCB is then brought under the robot and the dispensing takes place.

This system has been developed and demonstrated in Brigham Young University's Electronics Assembly and Automation Laboratory. Due to the absence of a solder-paste screen, this solder-paste dispensing system is capable of lots as small as 1, yet provides the quality of a screen-printing system. The disadvantage is a lower throughput.

Introduction

In the process of assembly of electronic printed circuit boards, the solder paste is usually deposited on the printed circuit board (PCB) by means of an automated screen-printing machine. These machines feature good reliability, high throughput, and high quality, but their major weakness is that there must be a screen built for each PCB manufactured. These screens, typically manufactured by very precise etching or laser milling machines, can be quite expensive. Additionally, for each different PCB, the screen-printing machine must be manually set up with the corresponding screen. There are additional disadvantages to the screen-printing method, as discussed in the next section. In general, the present method of screen-printing the solder paste is viable only for large volumes of a given PCB design.

When producing prototypes or small lots of PCBs, it is valuable to have a flexible manufacturing system that can change between PCB shapes, sizes, and layouts with ease. Setup time and changeovers can be costly, wasting valuable time and preventing the PCB from being tested and ultimately sent on to mass production.

In such flexible manufacturing environments, the solder paste dispensing step of manufacturing a PCB is done using a dispensing syringe because the cost of making a dedicated stencil as is typically done in mass production can range from an additional $250 to $500 depending on the requirements of the stencil.[1] Often times the design of the PCB will change before mass production thus making the production of a stencil even more costly if changes occur multiple times.

Currently the setup time of the equipment for syringe type solder paste dispensing between PCBs is tedious and time consuming taking away from the flexibility of the system. The current method of programming the solder paste dispensing robot is simply trial and error programming. The centroids of the solder pads are measured or taken from the CAD drawings of the PCB and manually programmed into the robot controller's software. The syringe must also be given information regarding the amount of solder to dispense. The dimensions of the pads are used to calculate the surface area of the pads. The surface area then corresponds to an amount of solder to be placed on each pad. This data must also be manually programmed into the robot controller's software. This process is not only tedious, but time consuming and expensive. Human error is a large factor in this type of programming, requiring the system to be monitored even after programming to find and debug any errors in the system. The time needed to program the solder paste dispensing hardware takes away from the ability of a system to have the "flexible manufacturing" title because of time-consuming changeovers between different PCBs. Automating the programming step of controlling the solder paste dispensing equipment would eliminate human error and provide for "flexible" automated solder paste dispensing.

Existing Technology

Currently in most mass production settings, solder paste is placed on a PCB using a "stencil printing" or "screen printing" process. This process is similar to silk-screening a t-shirt; by pushing solder paste over a metal stencil using a squeegee, forcing solder paste into the apertures of the stencil onto the pads of the PCB. Stencil printing is one of the most cost effective processes in electronics manufacturing for solder paste deposition. The process can provide very high throughput and is widely used in industry. Although stencil printing is popular and is common in industry, it is apparent that industry does not fully understand the stencil printing process indicated by the fact that industry reports 52 – 71% of fine pitch SMT defects are directly related to the stencil printing process.[2] There are over 45 factors that contribute to successful screen printing with the most influential of those being stencil thickness, aperture size, aperture shape, board finish, solder paste type, and print speed.[2]

Some of the disadvantages to screen printing include:[3]

1. Costly step stencils are required for placing different thicknesses of paste on a PCB.
2. The storage space and capital of storing possibly hundreds of stencils in a manufacturing facility is costly and uses up real estate.
3. In squeegee-based printing up to 40% of the solder paste placed in the stencil printing machine ends up being thrown away because of material degradation from the paste being left exposed in large open areas on the stencil over time.
4. Cleaning the under side of the stencil between PCBs to prevent contamination of future prints adds to the cost in solvents and paper.

Pressure dispensing, the focus of this study, has several advantages over the screen printing methods of placing solder paste. The main advantage over screen printing is that pressure dispensing eliminates the need of a stencil, thus eliminating an entire process and equipment in the assembly of a PCB, ultimately reducing cost. Cleaning of the dispensing cartridge is negligible when compared to cleaning a stencil because there are little to no cleaning supplies needed, especially in the case of disposable syringe cartridges.[3] Pressure dispensing uses a closed reservoir of solder paste whereas screen printers have open reservoirs, leaving them susceptible to contamination of foreign objects. Pressure dispensing is an inherently flexible process because of its ability to apply variable amounts of solder paste in any X,Y,Z location. Pressure dispensing finds its place particularly in high product mix environments because all that is required at a product changeover is to change a pre-stored dispense program. In many cases, solder paste dispensing is the only option to applying solder to a PCB. Such cases are found in modern 3D circuits in cell phones and in automotive applications. Other R&D situations prefer dispensing because each time a change is made to the design it is as simple as changing the line of data to update the dispensing instructions.[3]

Automating syringe type dispensing

To eliminate the need of manually programming the hardware in syringe type dispensing, a software program was developed that reads and interprets a Gerber file into a series of commands to automatically control the solder paste dispensing equipment. This software program was written in C++ using Microsoft Visual C++ and compiled to run on an X86 Intel-based platform. The software is capable of reading a Gerber file of the RS-274X format as defined by Barco Graphics. The RS-274X format is the industry standard Gerber file format with aperture parameters built into the data file. The Gerber file is in ASCII text format and can be read by any basic text file reader.

A Gerber file can contain a plethora of information containing artwork for photoplotters, CNC machines, and/or part placement machines in the world of electronics manufacturing. A Gerber file will typically have information defining where a PCB image starts, what shape it will take, and where it will end. Within the borders of the PCB, the Gerber file will define aperture information including the shapes and sizes of holes, traces, pads, and other features if needed.[4] In the case of solder paste dispensing, the information essentially needed from the Gerber file are the fiducials for PCB calibration and the aperture sizes and shapes of the pads on the PCB.

Upon reading the Gerber file, the program connects to the software and hardware controlling the solder paste dispensing equipment. This software is the workhorse of the system removing almost all user intervention creating the automated system.

Included in the software is the ability to read a Gerber file and pull out the important information essential to placing solder paste on a PCB. This includes reading the initialization parameters of the Gerber file which sets the mode (inch or millimeter units) and the decimal format of the data within the Gerber file. The parameters determine whether the Gerber file was written in incremental or absolute format. Also the aperture sizes are given setting the exact dimensions and shape of each of the pads on the PCB. The software has the ability to read the various D Code and G Code instruction sets used in CAM software to instruct the robot how and where to move. Finally the software finishes its commands by reading the end of file instruction from the Gerber file and finishes all actions with the hardware by returning the equipment to its proper locations and presenting a PCB complete with solder paste. The software controls all of the following hardware through a ProfiBus interface in processing the PCB through the system: an audible and visual warning system, a conveyor belt, pallet stops, pneumatic actuators, a Seiko XM3000X robot, a vision system, and a solder paste dispensing controller. The software interfaces with Windows NT 4.0 for message output by sending modal window messages to the user. The software also incorporates many of the CODE™ CimControl server commands for signal command throughput and uses VisionBlox® software for camera and image evaluation.

Determining solder paste volume to dispense
The volume of solder paste dispensed on each pad on a PCB is critical in order to provide for a good solder joint between the leads of the components and the pads on the PCB. An algorithm determining how much solder paste to dispense on a pad depending on its surface area and shape was needed. After researching and talking to professionals and PCB assembly companies, it was determined that there is not a set standard on how much solder paste to dispense on a pad of a PCB. There were simply too many factors to take into account when determining how much solder paste to place on a particular pad. Those factors include the surface tension of the substrate, surface tension of the solder paste, pad dimensions, component shape and size, density of solder paste, reflow profile of solder paste, rheology characteristics of the solder paste, and so forth. Some general standards used in stencil screen printing were found. It was found that for fine pitch assembly (< 25 mils in width) the stencil thickness for screen printing solder paste onto a PCB was usually between 4-6 mils. For course pitch (> 25 mils in width) the stencils ranged from 6-10 mils thick. In cases where both fine pitch and course pitch components were required, a stepped stencil was used adding to the cost of the stencil. Because an exact standard for the amount of solder paste required for a specific area could not be found, a PCB made by Universal that contained every type of SMT components available for mass production was examined. Figure 1 shows the sample PCB made by Universal (this PCB was given out at a tradeshow as a demonstration of Universal's equipment capabilities in SMT printed circuit board manufacturing).

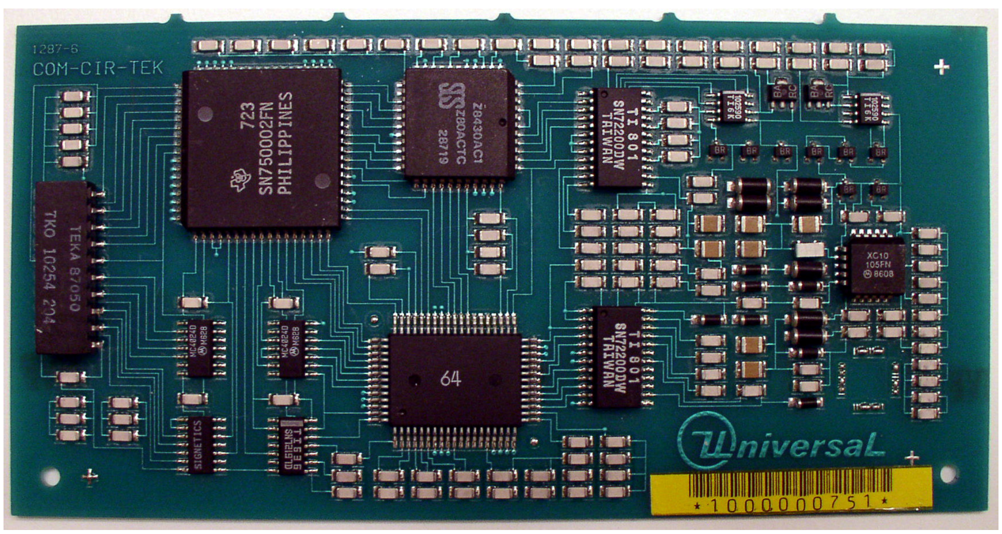

Figure 1 Sample PCB made by Universal to demonstrate capabilities in mass production.

Using the Universal PCB as an example, a defacto standard was created for the volume of solder deposited on the PCB with respect to the given area of the pad and the type of component. This was found by measuring the height of the solder joint attaching the component to the PCB along with the length and width dimensions of each pad. The data in Table 1 shows measurements of length, width, and height of each of the solder joints corresponding to the various types of components on the PCB. The measurements were taken using a set of calipers with a 1 mil resolution.

Component	Pad Length	Pad Width	Fillet Height
Chip Type 1	70 mil	60 mil	40 mil
Chip Type 2	100 mil	60 mil	40 mil
MELF 1	50 mil	50 mil	40 mil
MELF 2	60 mil	60 mil	40 mil
J-Lead	90 mil	25 mil	30 mil
Gull wing 1	80 mil	25 mil	30 mil
Gull wing 2	50 mil	40 mil	30 mil
SOT	30 mil	30 mil	30 mil

Table 1 Measurements taken of pad and solder joint geometry on a sample PCB made by Universal.

Using the measurements of the solder joints, it was necessary to calculate the volume of solder on each of the pads with their respective components. This would enable us to form a defacto standard that could be used generally for calculating how much solder paste needs to be placed on a pad depending on its area or dimensional characteristics.

Solving for Solder Fillet Volume

First the surface area of the pad was found by multiplying the length and width of the pad. These values are shown in the fourth column of Table 2.

$$A_{Pad} = L \times W$$

The area was then multiplied by the height of the solder joint to find the volume of the cube containing the solder joint with its base being the dimensions of the pad.

$$V_{Pad} = A_{Pad} \times H = L \times W \times H$$

Because the component uses up approximately 50% of the real estate on the pad, the volume was divided in half leaving only half of the pad that could hold solder paste.

$$V_{Pad}\ w/o\ component = \frac{L \times W \times H}{2}$$

This leaves us with a cube of solder paste on the remaining 50% of the pad. However, the solder joint is not in the shape of a cube, instead it is a fillet that extends between the pad on the PCB and the component as shown in Figure 2.

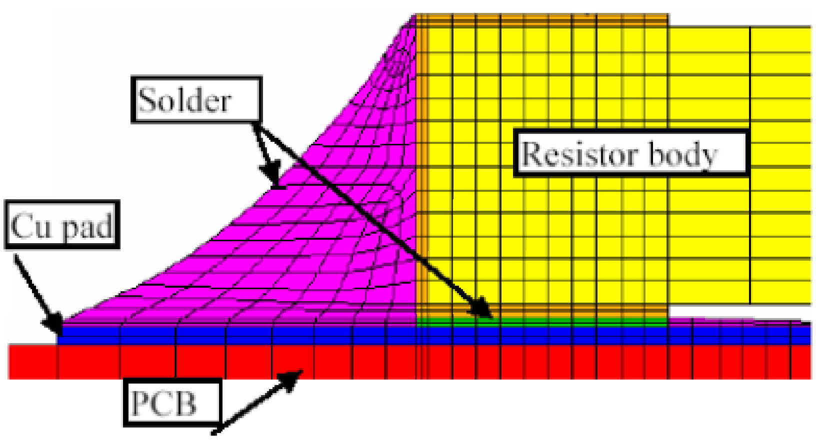

Figure 2 Profile of a PCB, resistor component, and solder fillet.[5]

As can be seen in Figure 2, the fillet is slightly concave. The question now arises, What is the correct method of predicting the volume of solder paste for a fillet that is "slightly" concave? In an article on solder joint reliability, the UK National Measurement Laboratory stated, "It is not known what is the optimum solder fillet shape and volume. Too much solder can reduce the joint's compliance but a thin joint will be weak. At present there are various rules applied to pad geometries but there is little scientific justification for these."[6] At the University of Greenwich, research done on the fatigue life of solder joints states the following, "The solder fillet shape, its curvature, and resultant standoff height of the component above the PCB is dependent on solder volume used and the material properties of the solder, for example surface tension. The surface tension of tin-lead and lead-free solders can be quite different resulting in different fillet shapes for the two materials."[5]

The Harbin Institute of Technology in China has developed a method of predicting the exact size and shape of a solder fillet by taking into account the pad size, component height standoff from the PCB, solder material properties, and component shape and size. In order to solve for the fillet volume they have set up a complex integral that is dependent upon the parameters of solder material selected including its density, surface tension, and wetting angle on a metal surface. This is done by accessing databases of solder material properties and component specifications for standard components.[7] This method of predicting the "perfect" solder fillet dimensions for any type of solder material and components seems to work well for the ideal situation, but in the case of a flexible manufacturing system that needs to be able to predict a solder fillet on the fly without access to complex databases, this is not very practical.

In order to effectively predict the volume of solder paste needed for a fillet, a simpler approach is taken by assuming the solder fillet is a prism in shape as shown in Figure 3. By assuming the fillet is a prism we are calculating more solder paste than is needed to create a "slightly" concave fillet. But we also need to consider that a small amount of solder will remain on the pad beneath the component. Also the solder paste is approximately 85% metal due to the fact that flux activators and other additives have been added to the metal to form the paste. Therefore approximately 15% of the paste will burn off resulting in a concave shape of the solder fillet.

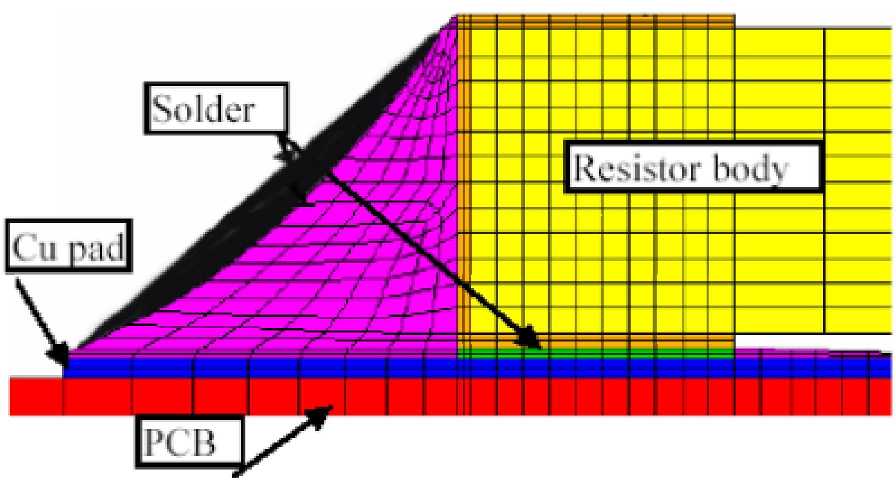

Figure 3 Solder fillet volume estimation represented by a prism (*Black line added to original image*)[5]

In estimating the needed solder volume for through-hole dispensing; a similar method was used for calculating volume. Mitch Holtzer, the business manager at Cookson Electronics Assembly Materials Group found that the majority of his customers wanted 100 to 110% of the required solder volume in the through holes with 360 degree coverage. In order to calculate the needed volume for a fillet he estimated the volume by calculating the volume of a cone with similar dimensions. Figure 4 shows a cone used in estimating the required volume around a through-hole component lead.

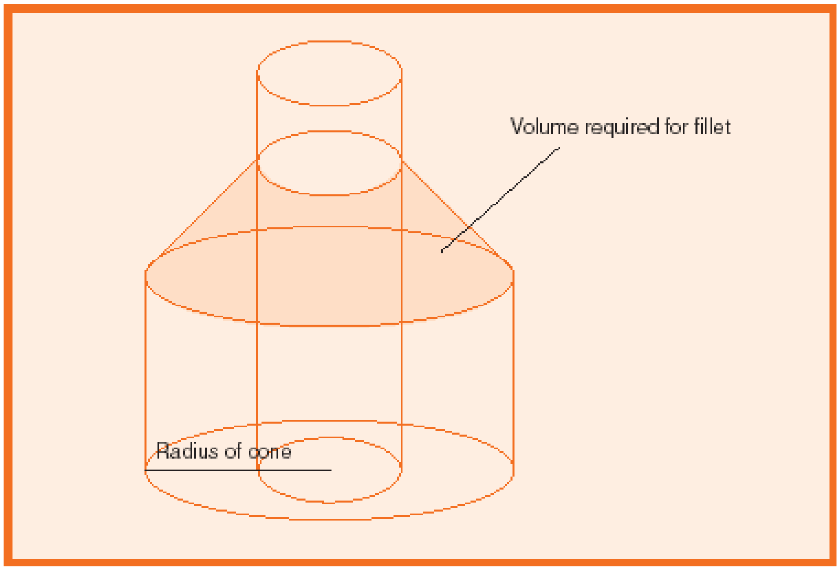

Figure 4 Cone used to determine solder volume for fillet.[8]

When calculating the volume of solder in the fillet, it can be assumed that it will resemble the shape of a cone for through-hole assembly.[8] If a cone represents the shape for a round pad in through-hole assembly then a prism should resemble the volume of solder paste needed for a rectangular pad in surface mount assembly. This allows us to divide the volume of the solder paste in half one more time giving us an approximation of the needed volume of solder to form the fillet.

$$V_{\textit{Fillet (prism shape)}} = \frac{L \times W \times H}{4}$$

This final volume of the prism was then divided by its length and width components leaving only the height. This leaves the average value under the curve, or simply the needed stencil height in order to dispense the correct volume of solder paste if this process was done using a screen printing stencil. This estimated stencil thickness became the defacto standard and was placed into the software program written to control the dispensing equipment. The values for the stencil thickness are used when reading the Gerber data and a request is made to place solder paste on a pad. Depending on the area of the pad, the software will choose the best stencil thickness for the job and thus calculate the volume of solder paste needed for that pad. Note in Table 2 the coarse-pitch components with sizes larger than 25 mils in both length and width require a stencil thickness of 10 mils in order to achieve the proper amount of solder, whereas the fine-pitch components on this sample printed circuit board require a thinner stencil of 7.5 mils to achieve the correct volume of solder on the pads of the PCB. The only exception to this was the gull-wing 2 component listed in Table 2. This component had larger pads, but little solder was found on them. It was decided that this was an anomaly and that it was actually a little short on solder; therefore if a pad of its size were to be dispensed using the automated system it would receive an equivalent volume of solder paste to a stencil thickness of 10 mils.

Component	Length	Width	Area	Height	Volume	Stencil Thickness
Chip Type 1	70 mil	60 mil	4200 mil^2	40 mil	42000 mil^3	10 mil
Chip Type 2	100 mil	60 mil	6000 mil^2	40 mil	60000 mil^3	10 mil
MELF 1	50 mil	50 mil	2500 mil^2	40 mil	25000 mil^3	10 mil
MELF 2	60 mil	60 mil	3600 mil^2	40 mil	36000 mil^3	10 mil
J-Lead	90 mil	25 mil	2250 mil^2	30 mil	16875 mil^3	7.5 mil
Gull wing 1	80 mil	25 mil	2000 mil^2	30 mil	15000 mil^3	7.5 mil
Gull wing 2	50 mil	40 mil	2000 mil^2	30 mil	15000 mil^3	7.5 mil
SOT	30 mil	30 mil	900 mil^2	30 mil	6750 mil^3	7.5 mil

Table 2 Measurements & calculations for creating defacto standard for solder paste volume.

Determining Dispense Flow Rate

The next task in automating the solder paste dispensing system was to study the Kester solder paste used in this study and its characteristics at room temperature (68° F). Several variables can be altered to change the dispensing characteristics: time dispensed, air pressure, speed of the rotary screw dispenser, needle size, and the rheology characteristics of the solder paste. These are critical factors in determining the effectiveness of the automated dispensing system. These dispensing factors are controlled in order to achieve consistent results.

The Techon Systems TS5000 rotary microvalve dispenser used in this study has a controller that regulates the amount of air pressure applied to the reservoir of paste. The air pressure was determined to work sufficiently at approximately 12 psi. The air pressure simply pushes material through the feed shaft to the Archimedes screw until it is dispensed by the rotary microvalve.

The needle size chosen for this research was a 22 gauge needle with an inside diameter of 16 mils. The screw speed on the dispense controller can be set from 0 to 24 volts DC. For the needle size chosen a safe screw speed that did not overload the capacity of the needle was 13.0 V. Higher voltages were tested but created undesirable results when the solder paste solidified within the needle head. Because the paste could not leave the needle at higher rates, it was forced to rotate within the needle head creating friction and thus heating the paste enough for it to solidify within the needle. When this occurred the flux was forced out of the needle tip leaving only a clear substance on the PCB surface.

In order to determine the flow rate of the solder paste an experiment was set up that placed 12 dots of solder paste on a substrate while increasing the dispense times of each dot by equal amounts. The diameters of the dots were measured using calipers and the volumes of the dispensed dots were estimated. The dispense controller only accepted times in .0001 second increments therefore a dispense time of 1 second would be given as the value 10,000 to the dispense controller. Table 3 shows the times, the measured diameters of the dots in mils and the estimated volume of the dispensed dot.

Dot #	Time (ms)	Trial #1 (mils)	Trial #2 (mils)	Avg. (mils)	Volume ($mils^3$)
1	0	0	0	0	0
2	0.3	38	43.0	40.5	17391
3	0.6	50	56	53	38976
4	0.9	68	73	70.5	91735
5	1.2	75	80	77.5	121863
6	1.5	80	84	82	144348
7	1.8	90	90	90	190852
8	2.1	100	95	97.5	242651
9	2.4	105	102	103.5	290261
10	2.7	107	109	108	329792
11	3.0	110	112	111	358045
12	3.3	115	114	114.5	392993
13	3.6	113	120	116.5	413948

Table 3 Solder flow rate estimations from time dispensed and size of dot.

Figure 5 shows a graph of the time vs. volume dispensed from the experiment on the previous page. By adding a trendline to the data points plotted on the graph we find an average amount of solder paste dispensed per unit of time. The trendline on the graph shows that an average of 11.58 mil^3 of solder paste were dispensed per unit of time.

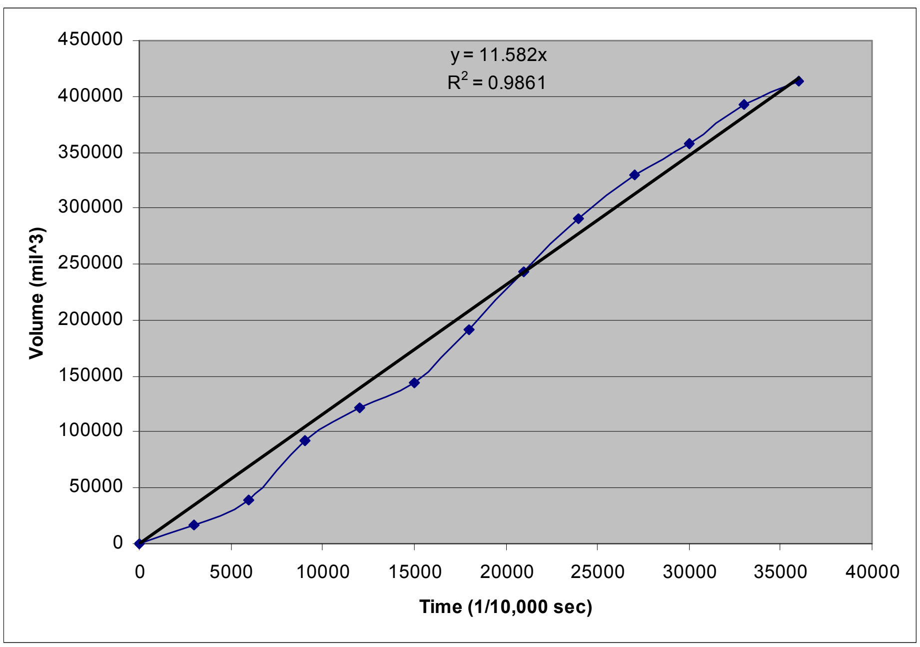

Figure 5 Graph of solder paste volume vs. time.

By dividing the needed volume of solder paste by 11.58, we find the value in time that needs to be sent to the dispense controller to dispense the correct volume of solder paste for any circumstance. The processing computer that controls the robots and dispensing hardware calculates these values on demand as it reads the aperture sizes from a Gerber file. Therefore, in summary to this point, the processing computer first reads the Gerber file looking for pad information. From the pad information it determines what thickness of stencil is needed for the dispense. The computer then calculates the volume to be dispensed depending on the stencil thickness. And finally the needed volume is then used to calculate how much time will be sent to the dispense controller.

Determining Needle Height

Finally the last critical item that needed to be addressed for a successful automated dispensing system was to determine the needle height off the PCB for each dispense in order to achieve effective and consistent dots. The dispense height of the needle must change dynamically depending on the volume of solder paste to be dispensed as the dispenser moves from one pad to the next. The volume of solder paste to be dispensed has been previously calculated therefore using the characteristics of the dot to be dispensed the needle height can be solved for. Mark Norris of Camelot Systems wrote in an article on dispensing technology that the most important factor in dispensing is the proportion of a dispensed dot's diameter to height ratio. "Low viscosity materials should have a 3:1 ratio and high viscosity materials should have 2:1 ratio or better."[9] After the needle has finished dispensing and the needle begins to move upward, there is an important relationship of surface tensions that come into play. In a good dispense the surface tension between the PCB and the solder paste will be greater than the surface tension between the needle and the paste. If this is the case the solder paste will snap off clean from the needle and the needle will be ready for the next dispense. If the needle was too low during the dispense, then solder paste will accumulate around the needle tip creating a greater surface tension on the needle. This may cause stringing or tailing to result. If the needle was too high in the dispense, then the material may not make a good contact with the PCB, leaving incorrect amounts of solder paste on the pad. The following dispense will most likely have an excess of paste and subsequent dispenses will not be consistent.[9]

Dispensing through a needle usually requires a material with a lower viscosity; therefore the 3:1 ratio for the diameter to height ratio was implemented into the program's software. Figure 6 shows a geometrical representation of a spherical cap. A spherical cap is the general shape of what a dot represents after being dispensed on a PCB.

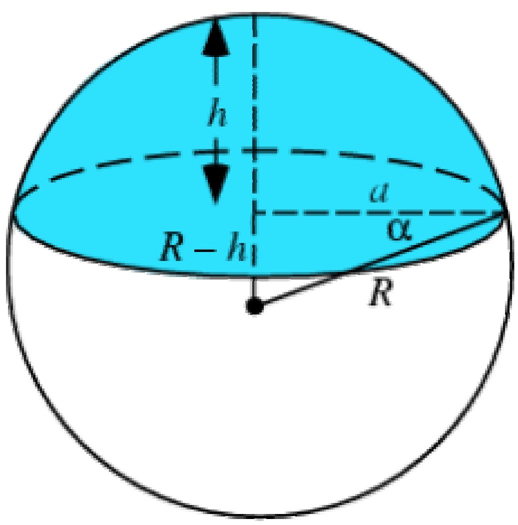

Figure 6 Spherical cap representing the shape of solder paste dispensed on PCB.[10]

The following equation solves for the volume of the spherical cap when given *a* as the radius of the base of the spherical cap and *h* is the height of the cap:

$$V_{cap} = \frac{1}{6}\pi\, h\,(3a^2 + h^2)$$

Because volume has been previously calculated from the pad dimensions as explained previously, the equation for the spherical cap needs to be turned around to solve for the height of the spherical cap given the volume and assuming the 3:1 diameter to height ratio. The result of the derivation is as follows:

$$height_{cap} = \sqrt[3]{V_{cap}\frac{24}{31\pi}}$$

This equation has been placed into the source code of the software controlling the dispense controller. The software will use the equation to determine the needle height distance on demand as the pad shape and size changes throughout the Gerber data file for the PCB being dispensed.

Results

The Electronics Automation & Assembly Laboratory (EAAL) at Brigham Young University was used to automate the setup of a PCB for solder paste dispensing in flexible manufacturing. The EAAL has an a electronics manufacturing system consisting of a Cartesian 4-axis robot, a solder paste dispensing tool and controller, a conveyor system and a computer to process data. The software interface for converting a Gerber file to dispensing instructions for the robot and dispensing controller was implemented on this hardware automating the PCB setup process providing for a more flexible manufacturing system. This software reduces the setup time down to a matter of seconds versus the hours of painstaking labor it took to manually program each pad on the PCB previously. The human error was also removed from the setup step, adding to the value of the software. The Gerber translation software was tested using standard Gerber files of the RS-274X industry format. The software then used the given Gerber data to apply solder paste to a PCB containing many of the SMT components used in industry.

The completed PCBs built in this study were compared to an industry-built PCB made using screen printing techniques. The quality of the solder joints were examined on each of the PCBs and the PCBs built in this study were equivalent to the results of the industry-built PCB. The solder joints on the PCB built in this study even had cleaner joints than those of the one made in industry. The solder joints were shiny, had the correct volume of solder, and the fillets were slightly concave showing that the algorithms written into the program's software were correctly calculating proper amounts of solder paste for each of the joints in both coarse and fine pitch assembly. Also the quality of the solder joints were visually evaluated to compare with the standard PCB made with screen printing of the solder paste; no quality issues were noted.

In order to effectively judge the solder joints made using the automated flexible manufacturing system studied in this research the following characteristics that make up good and poor quality solder joints were examined:

1. Solder should cover the surface area of the pad.[11]
2. Solder should create a slightly concave fillet between the component or lead of the component and the PCB. [11]
3. Solder should be bright and shiny in appearance. [11]
4. There should not be any cracks in the solder. [12]

5. There should not be any pits or porosities on the surface of the solder joints.
6. There should not be excess solder around the joint. [13]
7. Flux residues should not remain on the PCB. [12]

The following images show some of the results of a PCB (the tan PCB) built with the automated solder paste dispensing system described in this paper along side a sample PCB built in industry (the green PCB) using the screen printing process.

When examining these images, note that the fillets reach the top of components in the chip resistors and capacitors while covering the complete area of the pads. The solder joints are bright and shiny in appearance without any pits or porosities on the surfaces and there is sufficient solder to create a good fillet making a solid solder joint.

Chip Resistors & Capacitors

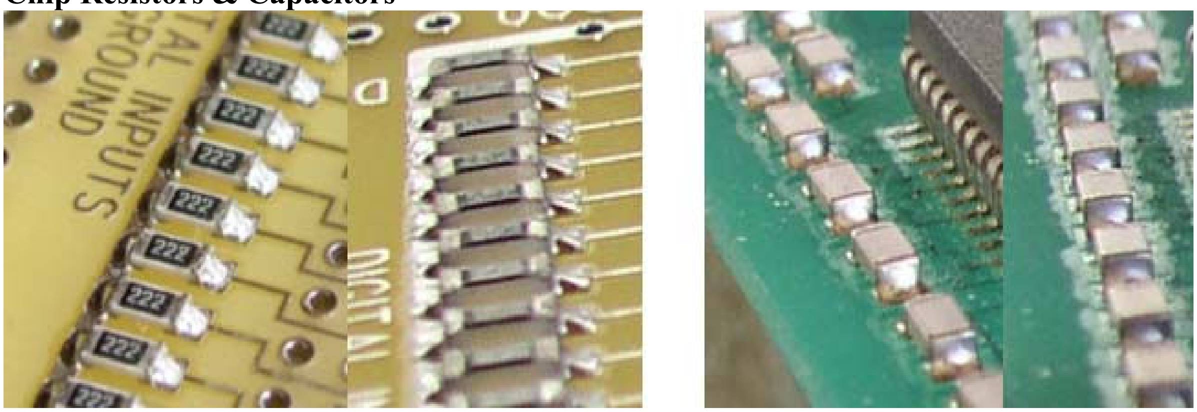

Figure 7 Chip resistors soldered to the test PCB and chip capacitors & resistors on the Universal PCB.

MELF Components

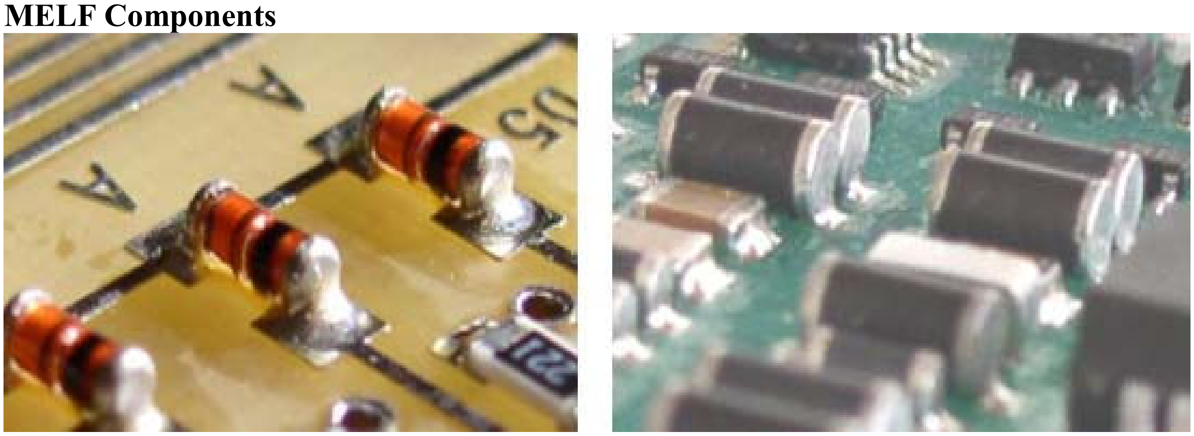

Figure 8 Cylindrical diode components on the test PCB and MELF components on the Universal PCB.

Gull-Wing Components

The gull-wing components were the most challenging components to dispense solder for because of their fine pitch leads of 24 mils. Instead of placing one dot of solder in the center of the pad, two dots of solder were placed on the centers of the pad after it was divided into two equal sections. Also, in order to prevent solder bridging, the amount of solder was decreased down to the equivalent thickness of an 8 mil stencil as if the process was done using a step stencil with a screen printing process. As can be seen in Figure 9, the results were excellent.

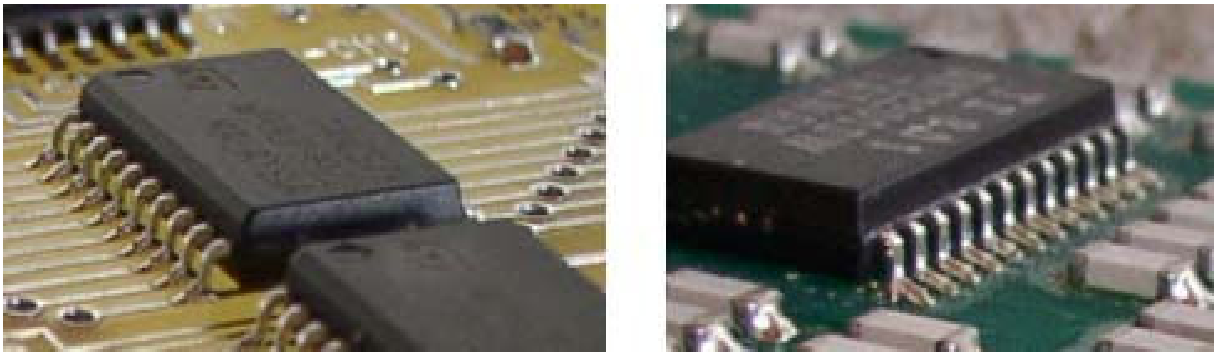

Figure 9 Gull-wing components on the test PCB and on the Universal PCB.

CONCLUSIONS

Flexible manufacturing systems are not necessarily known for their automation standards or achievements; instead they are better known for being slower and less productive than dedicated systems. This study has not brought flexible manufacturing up to the speed of a dedicated mass production system, but has made significant improvements to the changeover times for equipment setup of dispensing solder paste on a PCB. This decrease in time is a 1000% improvement when compared to being done by hand and a 133% improvement when compared to being dispensed using a manual foot pedal. If performed by a production dispensing robot at 60K dots/hr, the 398 dots on the PCB built in this study would be dispensed in 23.9 seconds. This compares favorably to the average cycle time of 49 seconds for screen printing as was measured from the actual cycle time of the screen printing equipment at Wolf Electronix in Orem, Utah.

Additional conclusions as a result of this research include:

- It is possible to automate solder paste dispensing in flexible electronics manufacturing while still maintaining the integrity of the end product.
- Syringe-type dispensing is more flexible than a single-thickness stencil often used in mass production. Syringe-type dispensing can dispense both fine-pitch and course-pitch components without added cost.
- The same robot that places the components on a PCB can also deposit solder paste on the PCB.
- Time and money is saved in prototype or small lots of PCB manufacturing by automating syringe-type dispensing
- The speed of solder paste dispensing is nearing the speed of stencil printing therefore making it a viable alternative for low-medium volume manufacturing.
- Machines are more consistent and accurate than work done by hand, thus making automated flexible manufacturing ideal for hand-made prototype boards.
- Costly human errors are eliminated by having a computer calculate the proper amounts of solder paste for each pad on the PCB from the CAD data.

Proceedings of the 2004 American Society for Engineering Education Annual Conference & Exposition

Within the university setting the system is ideal where space, finances, and use of the equipment is limited. The system has proven to save students hours of hand soldering time on the data acquisition board tested in this study. Time was saved by automating the system and placing the solder paste using a robot. Up until this point in the Electronics Automation & Assembly Laboratory (EAAL) at Brigham Young University, dispensing was controlled using a foot pedal. The automated controller has replaced the foot pedal and the requirement of someone having to manually control the solder paste dispenser has been eliminated. The system is also a good teaching tool for demonstrating methods done in electronics manufacturing and, of course, more specifically the advantages of having automation within a flexible manufacturing environment.

Recommendations for Future Work

One of the main issues that this study pointed out was the need to measure the vertical height of the PCB during the dispensing process. This study did not take into account variances in PCB height as the needle traveled to different parts of the PCB. This proved to be an issue especially when a new PCB was placed into the system. Each PCB tested had varying heights, causing the needle to be either too high or too low on initial start up. Initially we thought each board could be calibrated to the needle height and then run through the system. This method worked well in some cases but not so well in others. Two of the five boards tested were concave, causing the needle to be too distant in the center of the board if calibrated using the edges of the board. If the needle height was calibrated in the center of the board then the needle came in contact with the PCB near the edges of the PCB, either bending the needle or simply not allowing the solder paste to dispense from the needle tip. A needle height distance sensor that is capable of updating the needle height as the needle moves to each pad on the PCB needs to be investigated.

A time study of the amount of time wasted would be useful and reductions in the time required per dispense could be made to the system. Even though the system is significantly faster than hand soldering or using the foot pedal, more than half of the time used in motion could be reduced by using curvilinear motion in the robot movements, less Z height change between dispenses, less lead and lag time before and after each dispense and the order in which the pads are dispensed on the PCB. Some of the wasted time is inherent to the system, but could be improved if the hardware and/or the firmware were updated.

REFERENCES

1. "Screen Printing." Damaged Stencil-Process Defect Guide. 2003. PPM Monitoring. 27 October 2003 <http://www.thepdfshop.co.uk/ppm/defects/Screen/asp/bw0118.asp>.
2. Pan, Jianbiao, G.L Tonkay, R.H. Storer, R.M. Sallade & D.J. Leandri, "Critical Variables of Solder Paste Stencil Printing for Micro-BGA and Fine Pitch QFP." Proceedings from Electronics Manufacturing Technology Symposium, 1999. Twenty-Fourth IEEE/CPMT , 18-19 Oct. 1999 Pages: 94-101.
3. Speedline Technologies. "Should I Dispense or Screen Print Solder onto my PCBs?" Inimar. 1999. Inimar Process Technology in Electronics. 5 August 2003 <http://www.inimar.dk/pdf/Should%20I%20dispence.pdf>.
4. Barco Graphics. Gerber RS-274X Format User's Guide. Barco Graphics N.V., Gent, Belgium, 1998.
5. Lu, H., C. Bailey, M. Dusek, C. Hunt, & J. Nottay. "Modeling the Fatigue Life of Solder Joints for Surface Mount Resistors." EMAP. 2000. MicroSystems Modelling Group - The University of Greenwich. 5 August 2003 <http://msp.cms.gre.ac.uk/papers/hl_emap2000.pdf>.

6. Hunt, Chris. “Processing of printed circuit board assemblies.” Measurements for electronics assemblies. 2003. National Physical Laboratory. 5 August 2003 <http://www.npl.co.uk/npl/cmmt/programmes/mmp/mfea.html>.
7. Zhao, Xiujuan & Chunqing Wang, Guozhong Wang, Guanqun Zheng, and Shiqin Yang. “An Integrated System for Prediction and Analysis of Solder Interconnection Shapes.” IEEE Transactions on Electronics Packaging Manufacturing., 23.2: 87-92.
8. Holtzer, Mitch. “Solder Preforms in Electronic Packaging” Circuits Assembly April 2002: 42-44.
9. Norris, Mark. “Dispensing Technology.” SMT. October 1996: 56-58.
10. Weisstein, Eric W. “Spherical Cap.” Math World. 1999. Wolfram Research. 2 September 2003 <http://mathworld.wolfram.com/SphericalCap.html>.
11. “Soldering Materials.” U.S. Army Signal Center, Fort Gordon. 27 October 2003 <http://www.gordon.army.mil/stt/31C/b07lp1sa2.html>.
12. “Wave Soldering.” PCB-Process Defect Guide. 2003. PPM Monitoring. 27 October 2003 <http://www.thepdfshop.co.uk/ppm/asp/wave.asp>.
13. “Solder Connections.” U of U ECE Secondary Server. 2003 Electrical Engineering Department University of Utah. 27 October 2003 <http://www2.elen.utah.edu/~ee1010/EE1010_Lab1.txt>.

JON PLUMMER
Jonathan R. Plummer is a design engineer at Sumitomo Electric Wiring Systems Inc. He received a B.S. and an M.S. degree in Electronics Engineering Technology from Brigham Young University.

BARRY LUNT
Barry M. Lunt is an Associate Professor of EET at BYU in Provo, UT. Dr. Lunt received a B.S. and an M.S. degree in EET from BYU, and a Ph.D. in Occupational and Adult Education from USU in Logan, UT. He has spent seven years in industry as a design engineer. His present research emphases are the physical design of electronic circuits and systems, and engineering education.