An Interesting Approach to Yield Improvement for the Solder Paste Printing and Reflow Process

Lauri Märtin and Kristjan Piir, Enics Eesti AS;
Keith Bryant, SMT Solutions, South Africa

ABSTRACT

Whilst many companies invest time, effort and cost into up front work to fix snags which would lead to issues with yield during production, this paper shows the efforts of the company who looks to take things further.

With increasing pressure on cost reduction within our industry, companies are looking ever more closely at their manufacturing process. In order to remain globally competitive and even to succeed in their local market every dollar saved here helps the bottom line. However, in many areas there is a danger that lower price equals lower quality and therefore actually results in higher costs in the end.

The approach here involves spending a little more money than normal at the start of a project but less than hundreds of dollars and the results show savings of many times more than this outlay. However, it is acknowledged that this does take a little more time to get the job onto the shop floor.

The key to this methodology is that it needs the time and effort of a skilled team and time on a production line before the job is started. But as the paper shows it really does improve yield, reduce cost, save the potential issues around repair and gives better reliability.

In essence the results of the solder paste printing process are analysed, after the components are placed, using X-ray and these results compared to the results after reflow soldering. The resultant pre-reflow solder paste shapes are impossible to see with the naked eye or by lifting the components, as the paste would not release evenly. This allows the engineer to determine how differences in printed paste shape and volume react when components are placed on them and how ultimately this affects product quality.

Post reflow problems including mid-chip solder balls were found to be common faults, as were issues under BGAs including insufficient solder and shorts.

The product is run on a "real line" and the results evaluated. Improvements are then made to the stencil design and other key process parameters to ensure that when in production the board is producing acceptable yields.

INTRODUCTION

The EMS company supplies customers in the various fields of transportation, building automation, energy, industrial automation and instrumentation working to optimize their value chains and improve their competitiveness.

The engineers responsible for Product Introduction always try to get the customer design to a standard that allows for high yield in production. However sometimes the design is fixed, the boards purchased already or there is another reason why the board has to be built as is.

In these cases, redesigning key stencil apertures or modifying other process parameters can make the difference between a low quality and expensive result and a high level first time pass rate and a cost effective build.

These engineers noticed that despite making improvements in many areas the production yields were often lower than anticipated. This led to some further investigation of root cause and a design of experiment to look at these issues, with a view to improving the manufacturing process still further. It was discovered that issues related to problems hidden from view were by far the biggest cause of the reduction in yield.

This started the use of the 2D off line X-ray system to check the paste after component placement on the pre-production runs, instead of using it as an AQL tool and for checking production 'first offs'.
This checking was in depth and required a high-resolution system with the ability to see angled views at maximum magnification.

By checking the solder paste dimensions and shape after component placement and then checking the finished boards for solder balls, solder joint quality and voiding levels were compared to the solder paste. Correlation was found between instances of poor quality and inconsistent paste shape and volume. Changes were made so that any problems could be addressed before putting the job into production. Quality was now built into the product at the start rather than bad boards inspected out and reworked to achieve acceptable results.

This has led to significant improvements in yield, reduced rework and scrap, proving that the methodology of this procedure offered significant process improvements.

METHODOLOGY

This paper does not investigate the differences between different stencil manufacturing technologies. Its findings are based on a consistent stencil manufacturing technology and using the same manufacturer to supply all the stencils evaluated. Figure 1 below is of part of a QFP aperture cut into a stencil.

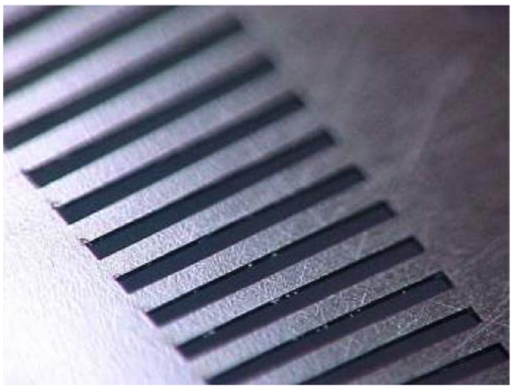


Figure 1: Part of QFP aperture cut into a stencil

It is a long established procedure to make a reduction in the solder pad size when cutting the aperture for the solder paste print in a stencil. There are many reasons and theories for this, from reducing the chance of smearing paste onto the board, to improving the gasket between the stencil and the printed circuit board and attempting to eliminate solder balling.

There is a general consensus that the x and y dimensions of the stencil aperture size should be reduced by 5% below the pad size to give a suitable aperture size for printing the paste . However, for smaller pads, the number often increases to up to a 10% reduction of the pad size.. Obviously as the volume of printed paste reduces a 5% reduction is in reality very little paste deposit difference, hence the need for a larger percentage to have an effect on the process of solder paste printing   This strategy does not take account of any issues arising due to problems on the board, incorrect pad size or shape for the component terminations, or other design constraints. Figure 2 shows a pad and overlaid aperture with a standard “global” reduction of 5%

Figure 2 A pad and overlaid aperture with a standard “global” reduction.

Many companies use this method of stencil modification and the process issues caused by this methodology are accepted as part of the assembly process.
Arguments in favour of this approach include: No time to make another stencil or too expensive or I do not know how to make it better.

Many companies will invest some time, effort and cost to overcome some of the failures which repeatedly appear during the assembly of the boards. Redesigning pads will help to reduce solder balls, repeatedly seen at the same components as in Figure 3 where a short circuit can be seen between the two right hand chip components. This is caused by a large solder ball actually connecting the two components during the reflow process. There is also a large single ball in the centre, which, if not removed, could move around the board causing a short circuit.

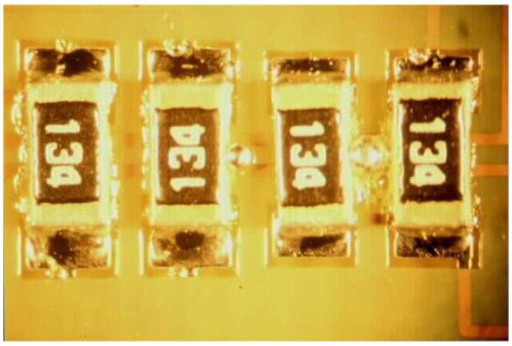


Figure 3 A short circuit seen between the two right hand chip components

Figure 4 gives a graphical representation of the whole process.

The left hand image shows the printed solder paste onto the two pads of a chip and the reduction in x and y are visible as the pad can be seen all around the edges of the solder paste deposit. The next left image shows the component placed into the solder paste deposit, excess solder paste is visible being squeezed off the pads on the inner edges. The next image shows solder balls moving under the component before reflow, with any kind of motion of the board resulting in this. Rapid machine movement during component placement is the most common cause of this. The right hand image shows the result after reflow soldering of the chip component, with joints formed on both pads and the excess squeezed out solder paste has formed into a ball and attached itself to the corner of the pad. This attachment is not strong and the ball can easily be detached and move to join others and/or form a short circuit.

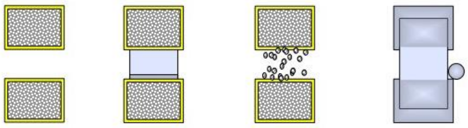


Figure 4: Graphical representation of the whole assembly process

An amount of pad reduction of at least 5% is needed in all cases except where over sized printing is required. For instance where a large component pad is attaching a heavy component to the board, so more paste is required to make the joint, the volume of paste is controlled by stencil thickness which is fixed, and the aperture dimensions, so over printing the pad is the only way to get enough paste to make a good joint.

Figure 5 below shows the stencil forming a gasket against the pad that ensures a good print with minimum bleed under the stencil. This is the main reason for using a reduction and as stated earlier many companies implement a blanket rule on pad reduction for this reason and do not investigate further.

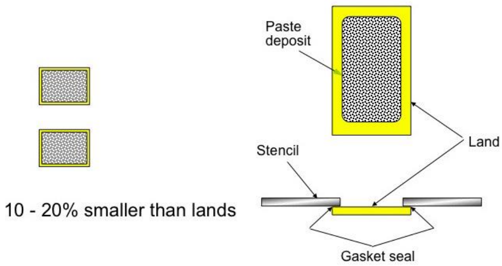


Figure 5: Stencil forming a gasket around the pad.

In addition to visible faults that can be found by AOI or skilled inspectors, there are some faults that need X-ray to find them. Figure 6 shows a case of severe solder balling under a BGA; it is easy to see a huge number of very small balls around the solder connections. This phenomenon is often referred to as spattering. There are several causes including too much solder paste on the pads. Effectively after the joint has coalesced and formed there is solder material left over which cannot flow into the joint. This material then forms very small balls and attach themselves to each other or the side of the pad.

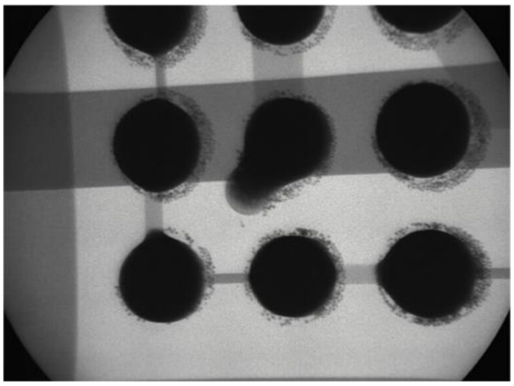


Figure 6: Severe solder balling under a BGA.

With a good X-ray system this fault is easy to see and with some diagnostic work can be fixed by reducing the apertures while ensuring that there is enough solder paste to allow a good joint to form.

Figure 7 , gives a good representation of some of the different reduction strategies used by engineers to improve yields. As pads get smaller then rounded corners are designed into the aperture shape to allow solder paste to release easier from the stencil. These changes are not an exact science, there are many different views on the correct radius etc., and many variables including solder ball size and flux type.

This is related to getting insufficient solder on finished joints due to poor paste release from the stencil. There are other potential causes that are outside the scope of this paper, including solder paste ball size, stencil finish etc.

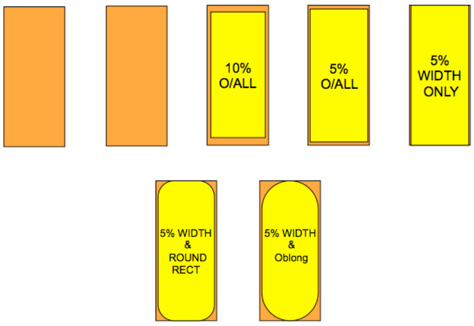


Figure 7: Different stencil aperture reduction strategies to improve yield.

For those companies wanting to improve yield further by investigating different aperture designs which may give better and more consistent results than a simple length and/or width reduction, there are many options. Figure 8 shows the normal reduction style
on the left, then the full arrowhead which is favoured by some, but the point can lead to solder balling and the reduction at the edges needs very accurate placement to ensure a full joint. For these reasons some engineers prefer the inverted arrow head next to it which reduces the chance of solder balls as there is no central point and also maintains a full length edge to help ensure a good joint, but it is possible for solder paste to squeeze out of the sides or back, leading to solder balls. The image on the right hand side requires very accurate and consistent placement, excellent board to stencil alignment and a good deal of confidence in the process.

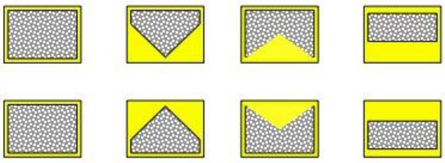


Figure 8: Different stencil reduction style strategies employed

Some of the images below are similar to Figure 8 but the dimensions are significantly different. The arrowhead in Figure 9 has a much blunter point and the paste volume is much greater than the previous one. This illustrates the complexities of stencil aperture design, as there is no agreed dimensions or even volume for any given named shape. The "Wendy House" aperture is preferred by many engineers as it does not end in a point which is viewed as a reason for solder balling. The "Horseshoe" aperture would seem the perfect solution as it follows the shape of the end cap of the chip component and does not have much volume or a point to cause mid chip solder balling. However, this shape is viewed to cause solder spattering due to the large volume of solder paste close to the edges of the pad.

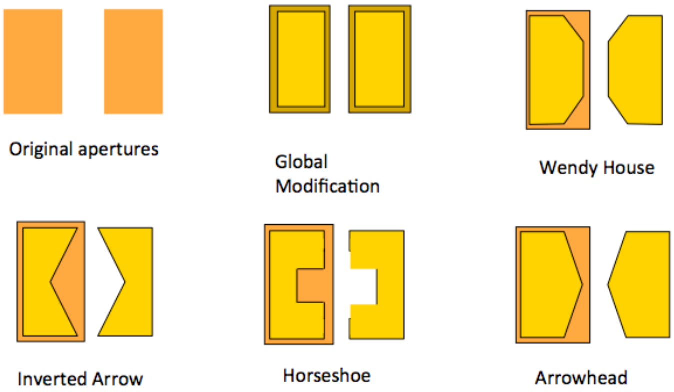


Figure 9: Additional stencil reduction style strategies

It would seem that there is no clean and simple solution and physical experimentation is the only way to find a solution for components that exhibit soldering problems in the field. This work is best done before the boards are placed into production to allow time for different shapes and dimensions to be tried. This will ensure a higher yield and a more reliable product when the production run is started and ultimately reduce the cost of manufacture due to the high first time passes and the low amount of rework or scrap.

EXPERIMENTS

0603, SOT89 and C1206 components were chosen for the experiments on aperture design as these were exhibiting the most issues related to soldering. The standard reductions in aperture sizes were giving problems in production; both mid chip solder balling and solder spatter. Figure 10, below, shows good examples of both faults. It should be noted that most of the solder spatter is under the chip and can only be seen by the X-ray image, as it is mostly between the pads not around the component which is the more common case.

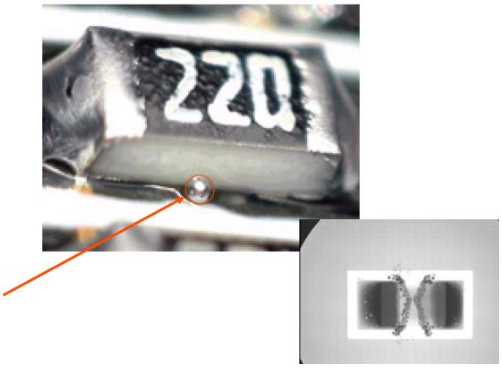


Figure 10: Solder balling and solder splatter.

Figure 11 shows some of the other challenges, which can be faced in assembly caused by a design that is less than perfect. The pad design allows for a very large area behind the terminations that can lead to a thin joint as the paste is spread over a larger area. An effort to cover the pad area may lead to excess solder and spatter, but if the pad is bare copper it needs to be covered with solder for long-term reliability.

The tracking on the left of the components is wider than the tracking on the right and this seems without reason. This means that heat is absorbed by the solder paste at differing rates on each side of the components and this can lead to tomb- stoning or component lift.

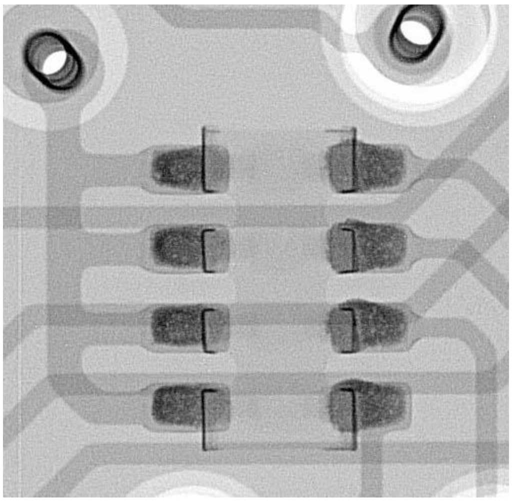


Figure 11: Incorrect design example

Figure 12 shows the resultant image after reflow and a large area of untinned pad is visible.

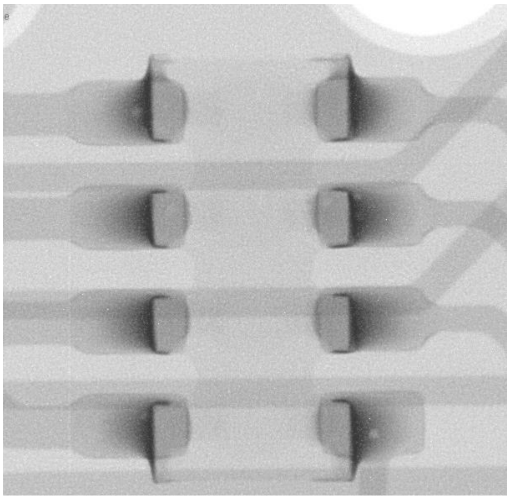


Figure 12: Reflowed component areas

The first amended aperture design conducted on a range of 0603 components (Figure 13) was an Arrowhead profile similar to the one in Figure 9 as can be seen in the X-ray images in Figure 13. This still allowed paste to spill over the edges between the pads when the components were placed and also over the sides of the pad where the components had been placed. When reflowed this exhibited mid chip solder balling and was not a success.

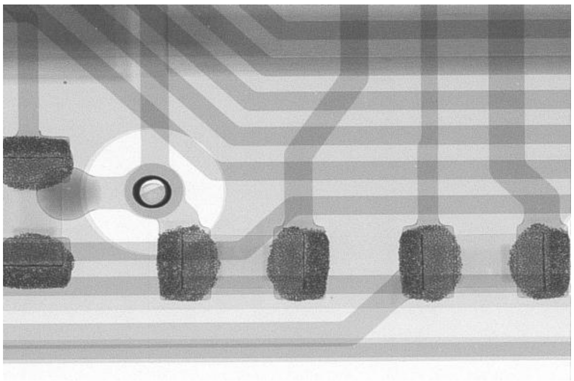


Figure 13: Paste printed using Arrowhead designed stencil

The next shape tested was the “Wendy House” or “Home Plate” as in Figure 9. This can offer advantages over the Arrowhead as it does not have a point on the internal edge and this can reduce the chance of mid chip balls. However, the volume of solder paste tends to be higher and can lead to solder spattering. It can be seen from Figure 14 that the solder paste has remained within the pad boundaries except under the component where there is a slight encroachment.

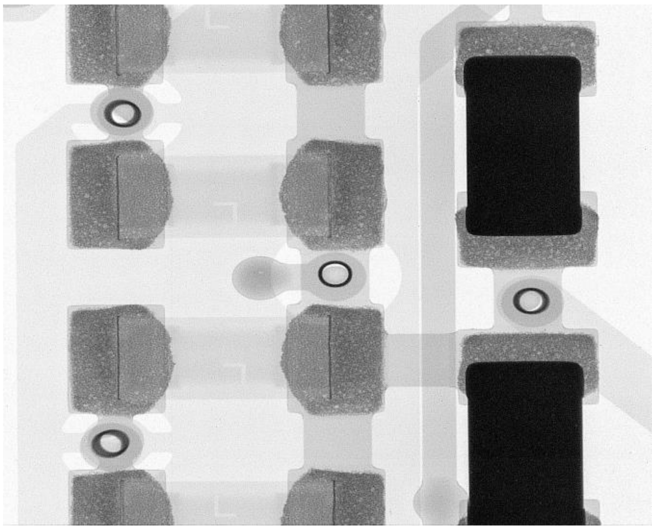


Figure 14: Wendy House/Home Plate stencil design for paste printing

The resultant X-ray image below in Figure 15-1 shows that the higher volume of solder paste did have a negative effect on the finished joint causing solder spatter outside of several of the components. This is an unacceptable process defect as the solder balls are free to move around the assembly where they can easily create shorts. Figure 15 shows one example of this failure mechanism and also mid chip solder balling which was also seen as a result of this aperture design.

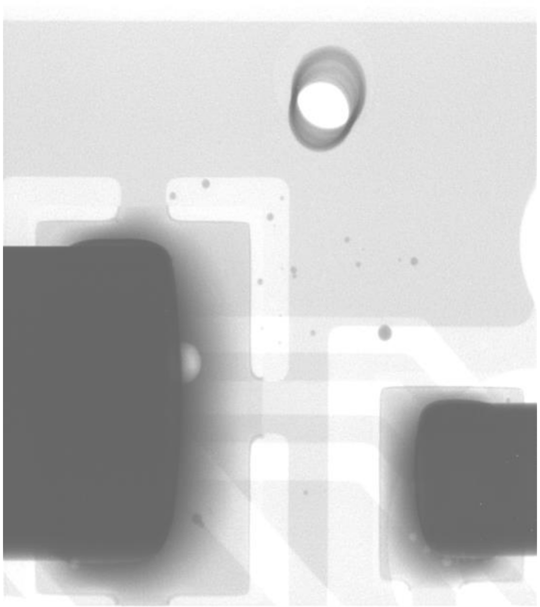


Figure 15-1: X-ray image showing reflowed solder with solder splatter.

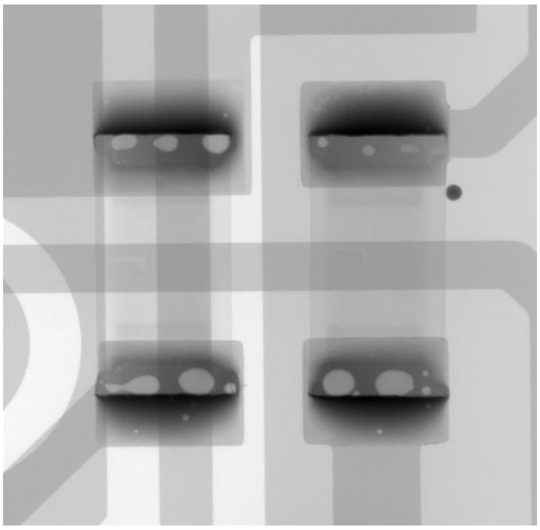


Figure 15-2: X-ray image showing solder balling.

The best result was achieved by reducing the width of the “Wendy House”/ “Home-Plate” shape by a further 5%. This gave consistently good results, a good joint shape with no solder balling or spattering. For all 0603 components this aperture design was adopted and Figure 16 shows a typical result. There is evidence of solid dark coloured joints, very little voiding, good pad coverage with no solder balling.

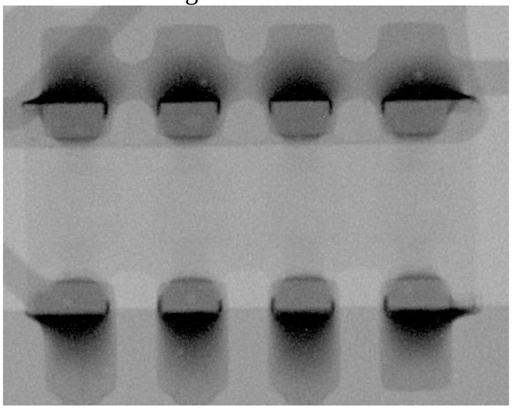


Figure 16: Improved reflow solder result with modified stencil design.

When investigating the larger 1206 components the situation was very similar. Simple aperture reduction produced very poor results both with mid chip solder balls and spattering. Figure 17 shows the extent of this solder paste spread when the components are placed into the solder paste. The pad design here is also quite tight which adds to the problems for the manufacturer.

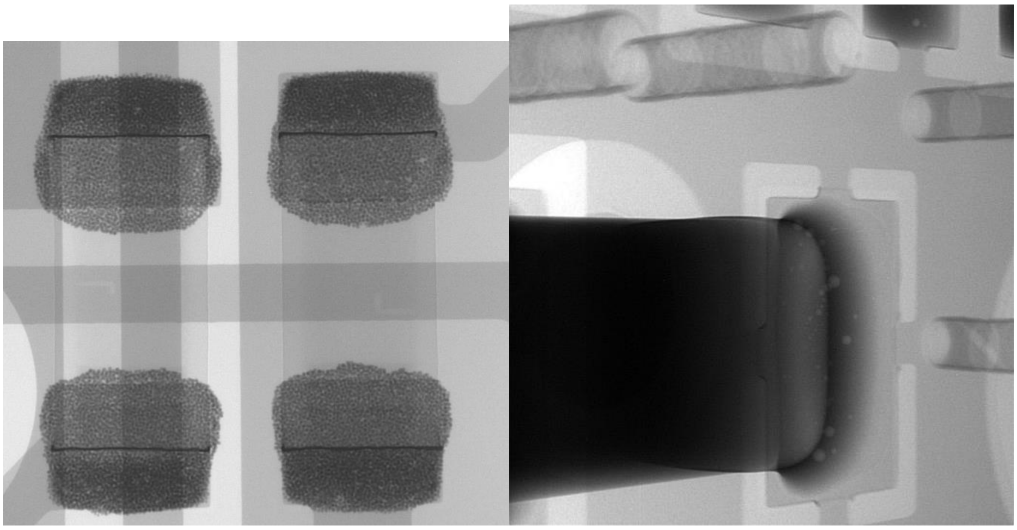


Figure 17: Simple stencil aperture reduction leading to solder reflow issues.

Using the “Home Plate” or “Wendy House” aperture shape on this pad gave some improvement, as can be seen in Figure 18. This shape is better suited to the two components on the right hand side (the 1206 component) rather than the 0603 components on the left hand side of the image. Pad design and the volume of solder required to make a good joint have a major effect on this. The larger components needing more solder to achieve a good solder joint fillet than the smaller 0603. Therefore, there is no excess solder to form solder balls or splatter around the components.

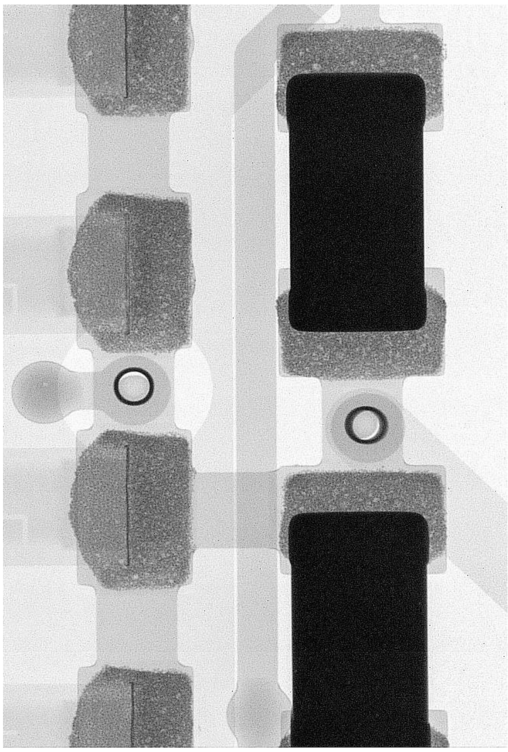


Figure 18: “Wendy House/Home Plate” stencil aperture design for 0603 (left) and 1206 (right) components

This shape and volume on the 1206 components produced consistently good results with joints as seen in Figure 19. Once more, there was a good joint, a well-covered pad and no evidence of solder balls.

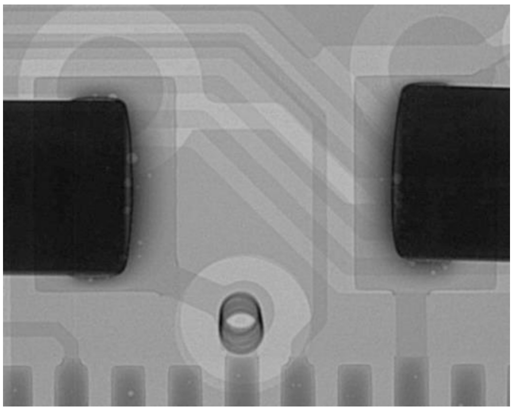


Figure 19: Reflowed 1206 chip components

The SOT 89 components presented more complex challenges due to the number, shape and size of the legs and potential issues of co-planarity. This would normally be countered by having extra solder paste volume to allow for any differences in height, which would be caused by the three smaller component legs not being parallel to each other or in line with the larger thermal pad, allowing them to rest flat onto the four pads of the device. It can be easily seen from the top X-ray picture in Figure 20 that the solder paste has spread well outside the large pad of the device.

This led to the formation of the large solder ball visible in the next picture down in Figure 20. To overcome this issue the pad width was reduced still further and a “Wendy House”/ “Home-Plate” shape was used at the front of the pad. This achieved the satisfactory result seen in the bottom picture of Figure 20 with no failures were seen after this shape was adopted for the SOT89 devices.

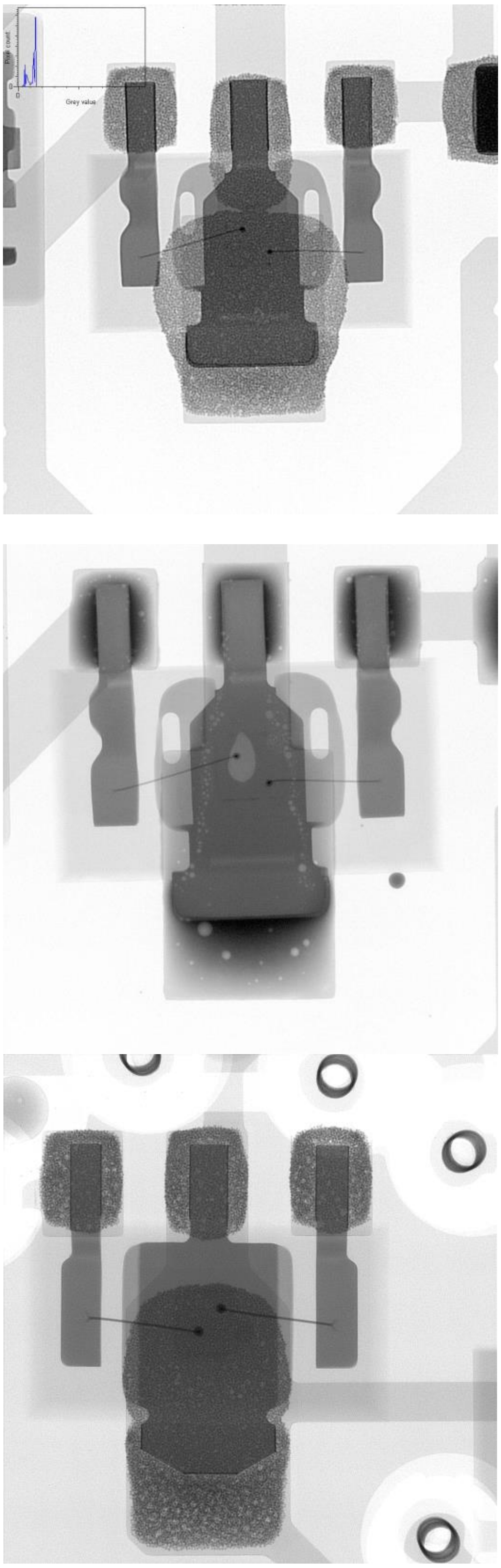


Figure 20: SOT 89 X-ray Images

RESULTS

The most accurate way of measuring the success or failure of this type of experiment is to monitor the change in first time pass results on a production line. Table 1 shows the results over a 16 week period while the improved design apertures were implemented into main line production.

The best unit of measurement of this type of result is DPMO, Defects per Million Opportunities. This is a tougher and more realistic methodology than Parts per Million.

$$DPMO = \frac{1,000,000 \times \text{number of defects}}{\text{number of units} \times \text{number of opportunities per unit}}$$

A defect is defined as a nonconformance of a quality characteristic to its specification. DPMO is stated in opportunities per million units for convenience: Processes that are considered highly capable (e.g., processes of Six Sigma quality) are those that experience only a handful of defects per million units produced.

Note that DPMO differs from reporting defective parts per million (PPM) in that it comprehends the possibility that a unit under inspection may be found to have multiple defects of the same type or may have multiple types of defects. Identifying specific opportunities for defects (and therefore how to count and categorize defects) is an art, but generally organizations consider the following when defining the number of opportunities per unit:

- Knowledge of the process under study
- Industry standards

-
- When studying multiple types of defects, knowledge of the relative importance of each defect type helps in determining customer satisfaction.
-

The time, effort, and cost to count and categorize defects in process output

The work contrasted those results before the changes in aperture design were implemented with the results seen after the new stencil with the improved apertures put into the printer and used for the production run.

The changes in aperture design were done in two stages. Stage 1 in week number 42 involved changing the solder paste stencil to one with the modifications to the apertures relating to chip components. As there are large quantities of these on the printed circuit board assembly a large shift in results was seen as these improvements fed through the production system.

Against a target of 150 DMPO the line had been running at around 750 per week average or 5 times the target. After implementation of the improved stencil the DMPO figures were much closer to the target.

Table 1: Target DPMO and Actual DPMO by production work week (

Week	Target	DPMO
W36	150	611
W37	150	693
W38	150	738
W39	150	953
W40	150	742
W41	150	794
W42	150	175
W43	150	363
W44	150	203
W45	150	223
W46	150	102
W47	100	149
W48	100	75
W49	100	67
W50	100	63
W51	100	62
W52	100	83

When the second stage of improvements was implemented in Week number 46 a further drop in end of line failures was seen. This stencil incorporated the improved aperture design for the SOT devices in addition to the improved chip aperture designs. This was a success such that the target DPMO figure was reduced to 100 from Week 47 a reduction of 33%. This meant that the target performance of the line was now 100 failures per Million Opportunities and a dramatic improvement over the average of around 750 DPMO seen previously. However as can be seen from the results in Table 1 the production line is now consistently performing better than the revised target. The improvement from a DPMO average of around 750 to double digits shows that these experiments were a success.

CONCLUSIONS

The paper shows that there are improvements in quality and reliability of the products produced due to the lack of rework and repair. Also there was an overall improvement in product quality and the reduced potential for faults to escape the inspection procedures.

This paper was not focussed on commercial aspects but it can be said that there is a high amount of saving in the cost of manufacturing this product, easily covering the costs of experimentation and new stencils, in a matter of days.

The experiments have shown that spending time working with X-ray images of placed components before reflow soldering allows improvements to be made to aperture design which can eliminate mid-chip solder balling and solder spatter caused by excess solder paste or solder paste which is squeezed off pads by the placement of the component.

There is no simple redesign of all stencil apertures which overcomes these issues but by incorporating X-ray technology to examine what is happening under specific component types or in areas where solder balling is an issue, it is possible to make significant improvements to first time yields.

This is a strong argument for increased effort at the front end of a production job, with an increase in engineering time leading to potentially significant improvements in yield and reduced cost of manufacturing.

However, the real benefit may be in the longevity of the product as it does not undergo rework and repair which have been shown to have a detrimental effect on the products life cycle and sometimes also performance in the field.

ACKNOWLEDGEMENTS
The authors would like to thank the following for their support in producing this paper:
P Zarrow, ITM Consulting
R Willis, EPS Services
A Weldon, TW Consulting

An Interesting Approach to Yield Improvement

Kristjan Pir and Lauri Martin, Enics Eesti AS, Estonia

Keith Bryant, SMT Solutions, South Africa

Introduction

- The EMS company supplies a wide range of customers from Transportation to Medical
- DfM is key to their success strategy
- But sometimes the board design cannot be changed for improved yields

Background

- Product Introduction Engineers often have to work with poor board designs
- This leads to low production yields
- Unexpected and unplanned costs are incurred fixing these faults
- Leading to low margins for the company

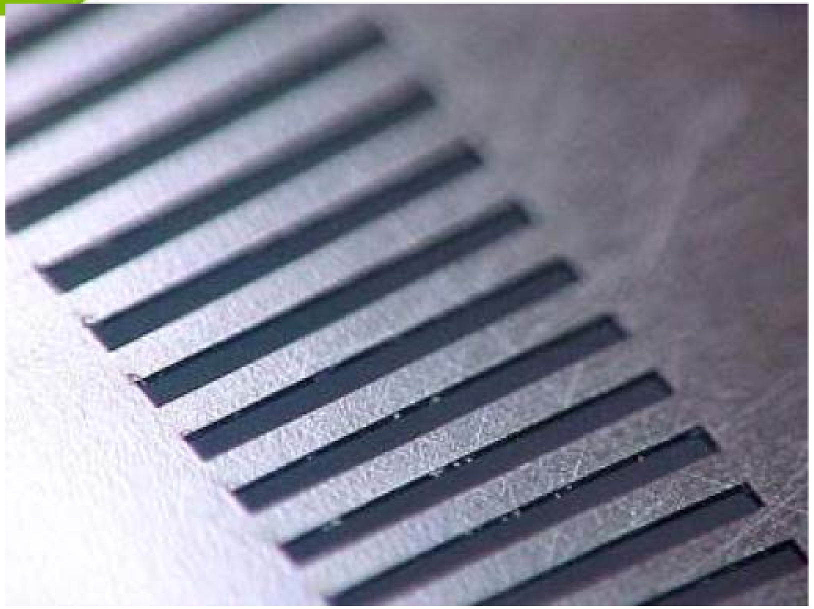
Root Cause

- The Engineers investigated the reasons for low yields
- Many assembly faults could be traced back to poor circuit board pad layouts
- This happens at the design stage, the inception of the board and can rarely be changed due to:
  - Boards already manufactured
  - Design approved by external authority
  - Cost of making the design changes

First Improvements

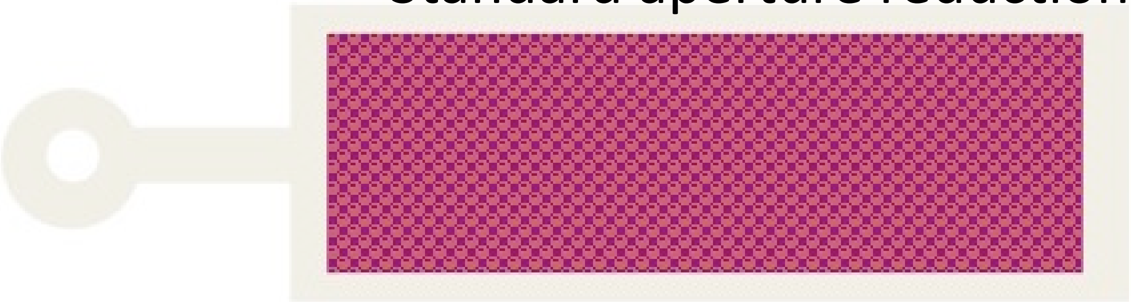
- Redesign of stencil apertures is the standard method of improving yields in this situation
- There are many theories on these design improvements and many differing views
- Also amending process parameters, including reflow profile can show improvements
- But even after doing these things, the team found production yields were still below expectations

Aperture reductions

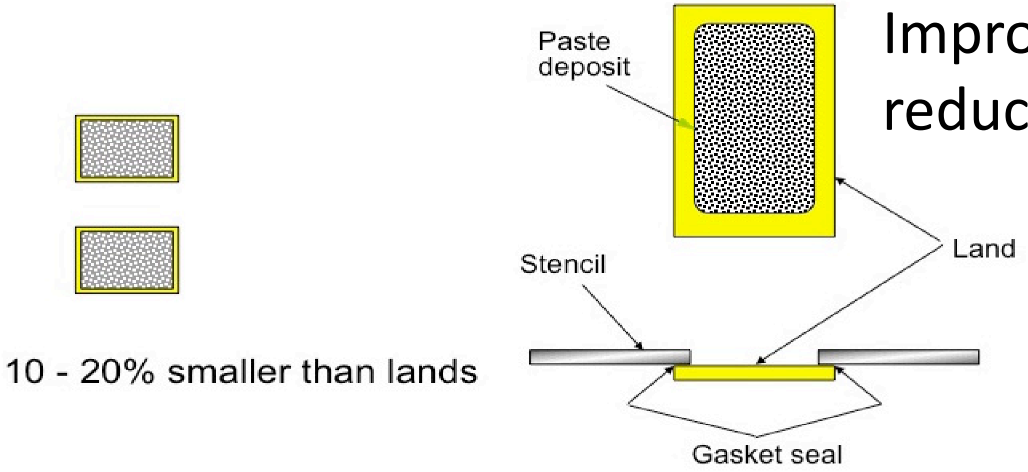


QFN stencil apertures

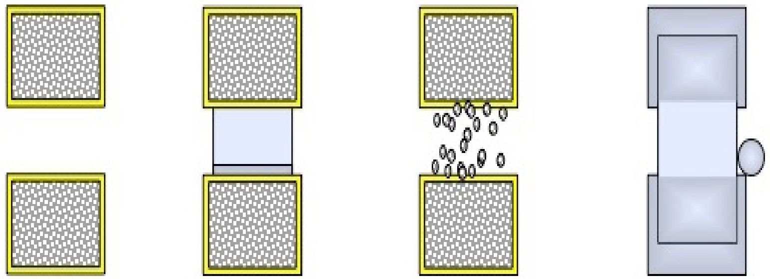
Standard aperture reduction



Aperture reductions

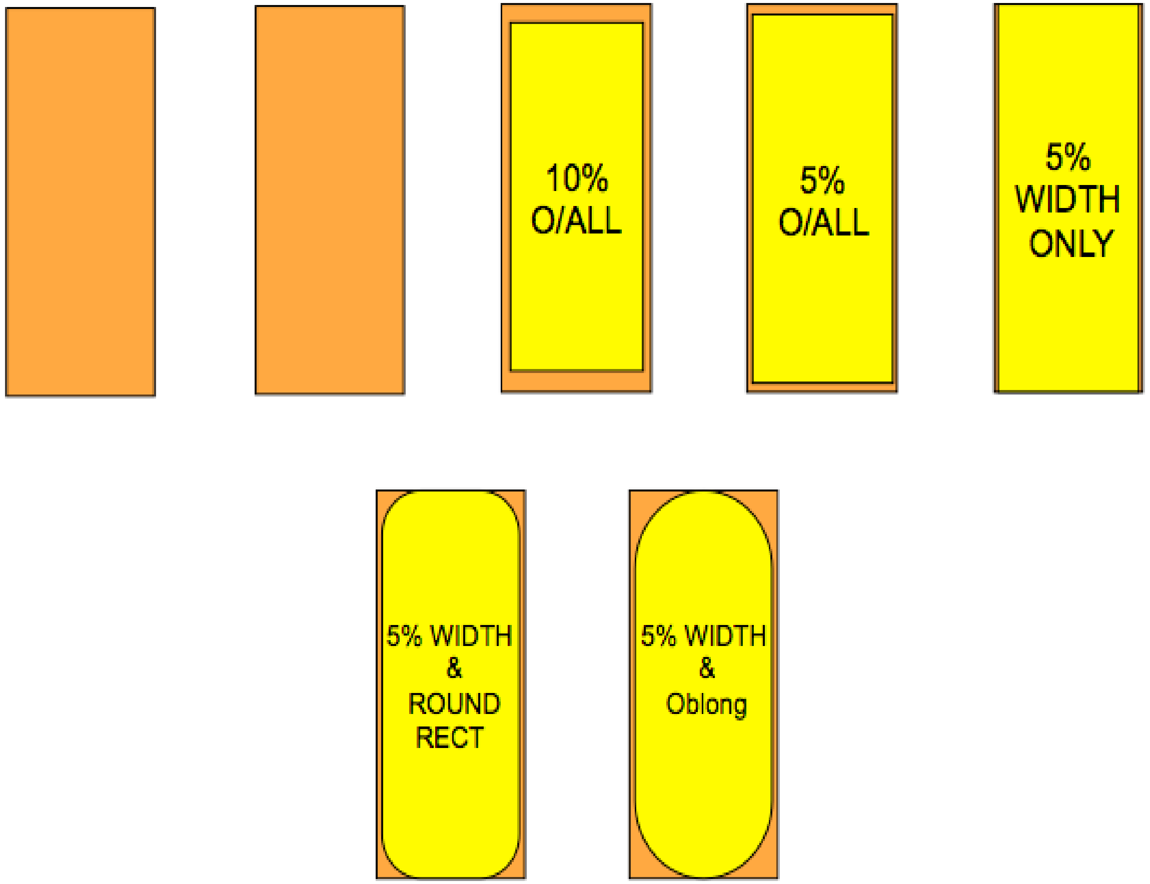


Improved gasketing, reduces solder balling

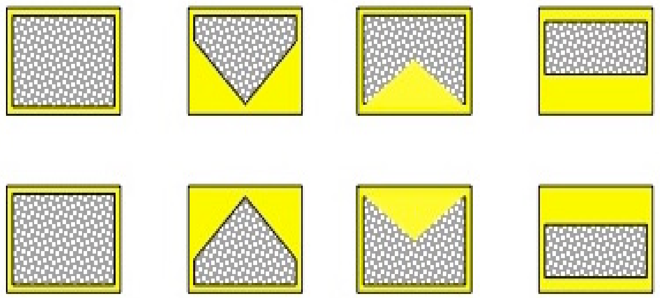


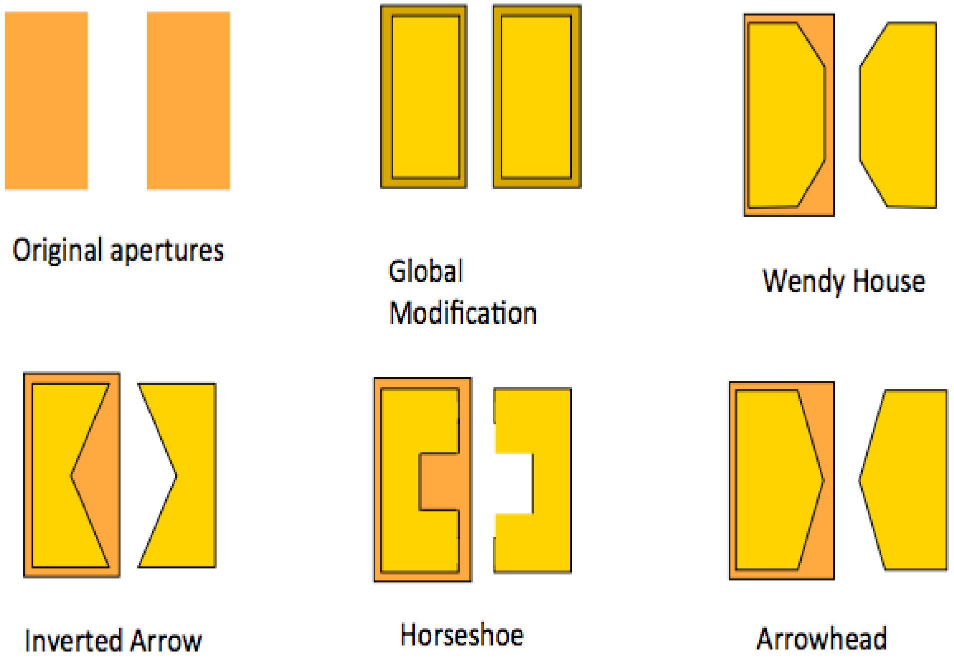
Under chip and mid chip balling
With reduced apertures

Modified Aperture Designs



Advanced Aperture Designs

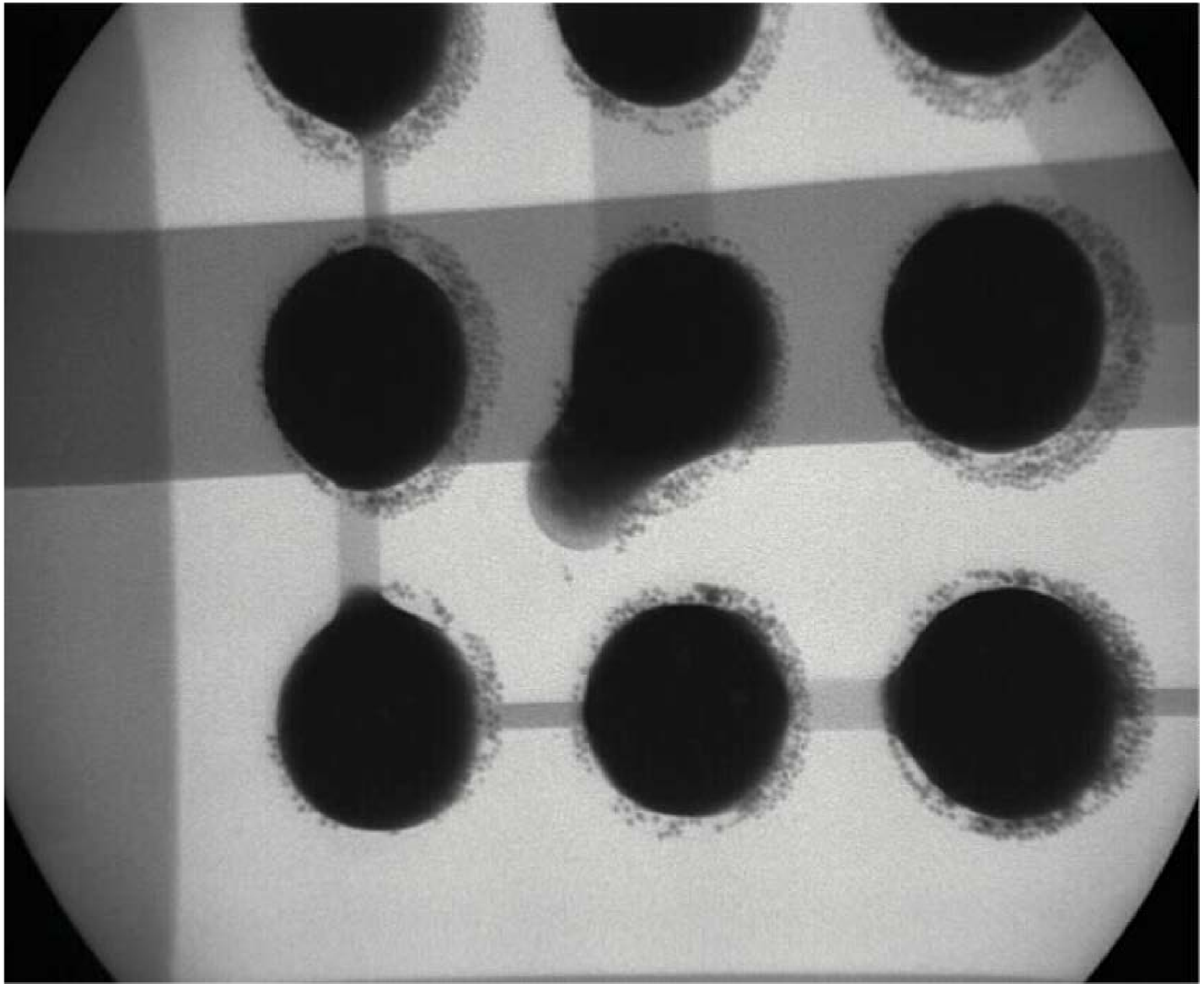




But….

- Results need board assembly to take place
  - So time and cost issues
- Many options and opinions to be tested, each with a new stencil
- Solder Related Faults still appeared at final test, x-ray or optical inspection at end of line
- A better solution was needed to get to acceptable yields

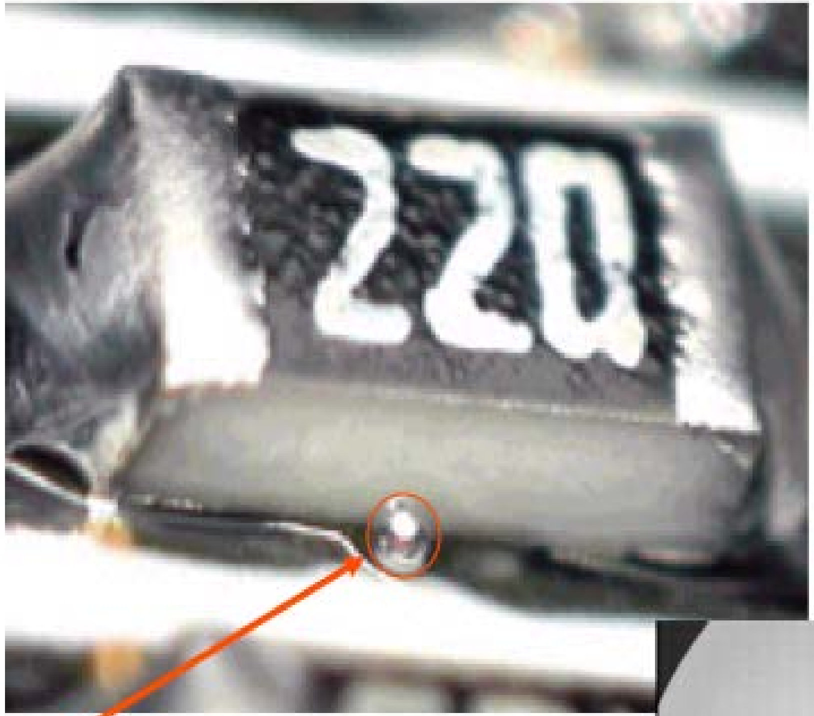
Spatter under BGA



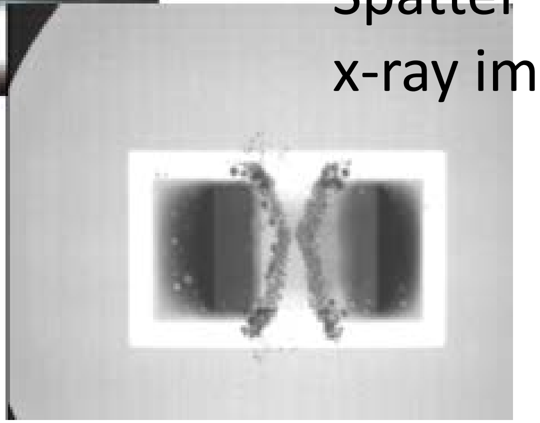
Innovative Approach

- Use At-Line x-ray to check solder paste positioning after placement
- This inspection technique is normally used on finished assemblies
- But with care it can be a great way of seeing what is happening to paste under the components
- This gives an indication of what will happen to the paste after reflow or wave soldering

Chip faults after standard aperture modifications



Mid Chip Ball



Spatter visible in x-ray image

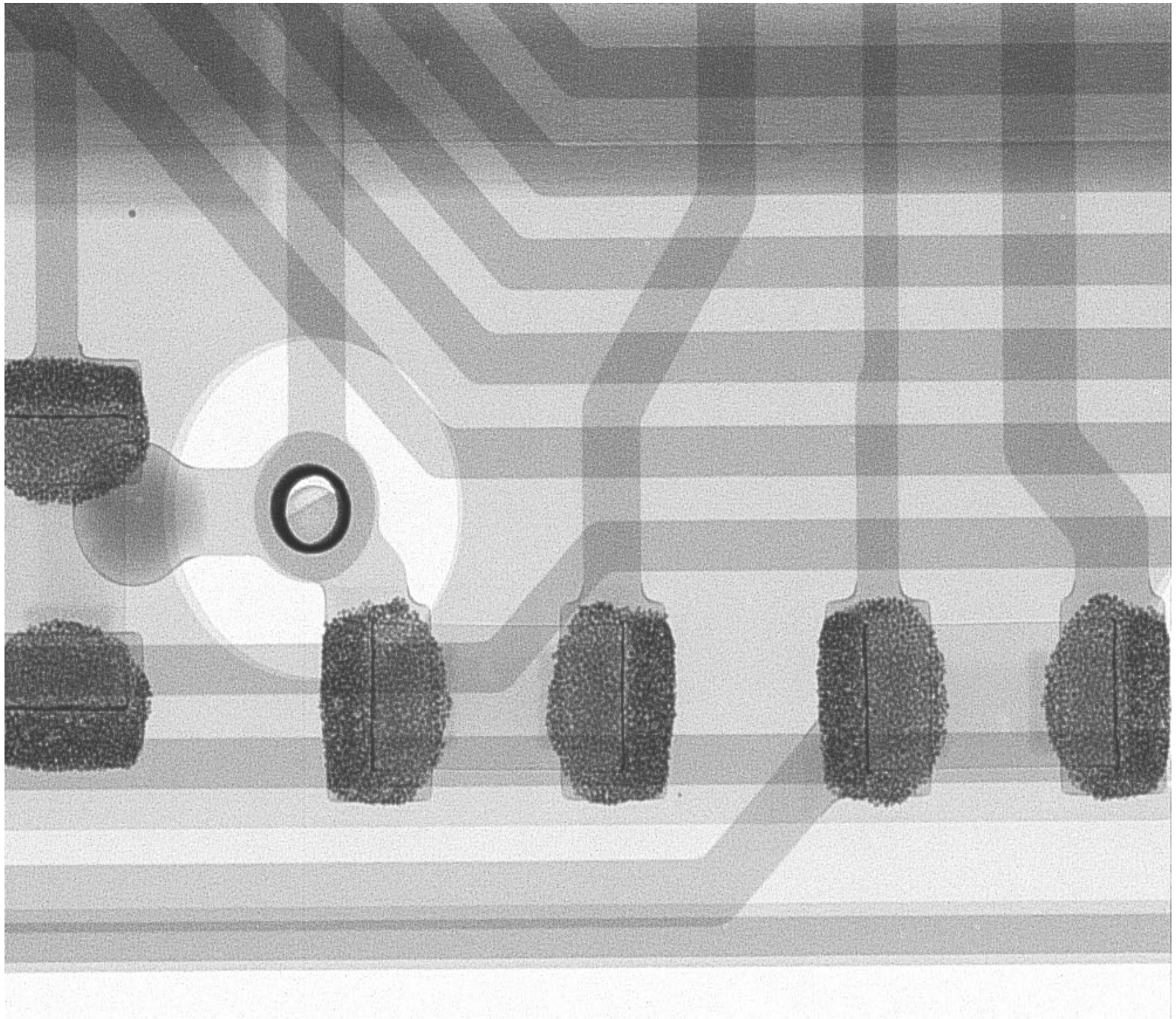
The Experiments

- 0603, SOT89 and C1206 components were chosen as they exhibited the most issues
  - Standard aperture reductions exhibited both mid chip balling and spatter
  - This lead to many issues post the assembly process
- A batch of boards was processed up to reflow and then all the suspect components were x-ray inspected, the images saved and results after reflow compared

0603 Arrowhead Design

- X-ray images show paste over the pad area after component placement
- Also paste between pads
- This could lead to both spatter and mid chip balling
- For this component and pad design the Arrowhead modified stencil aperture is not a suitable solution

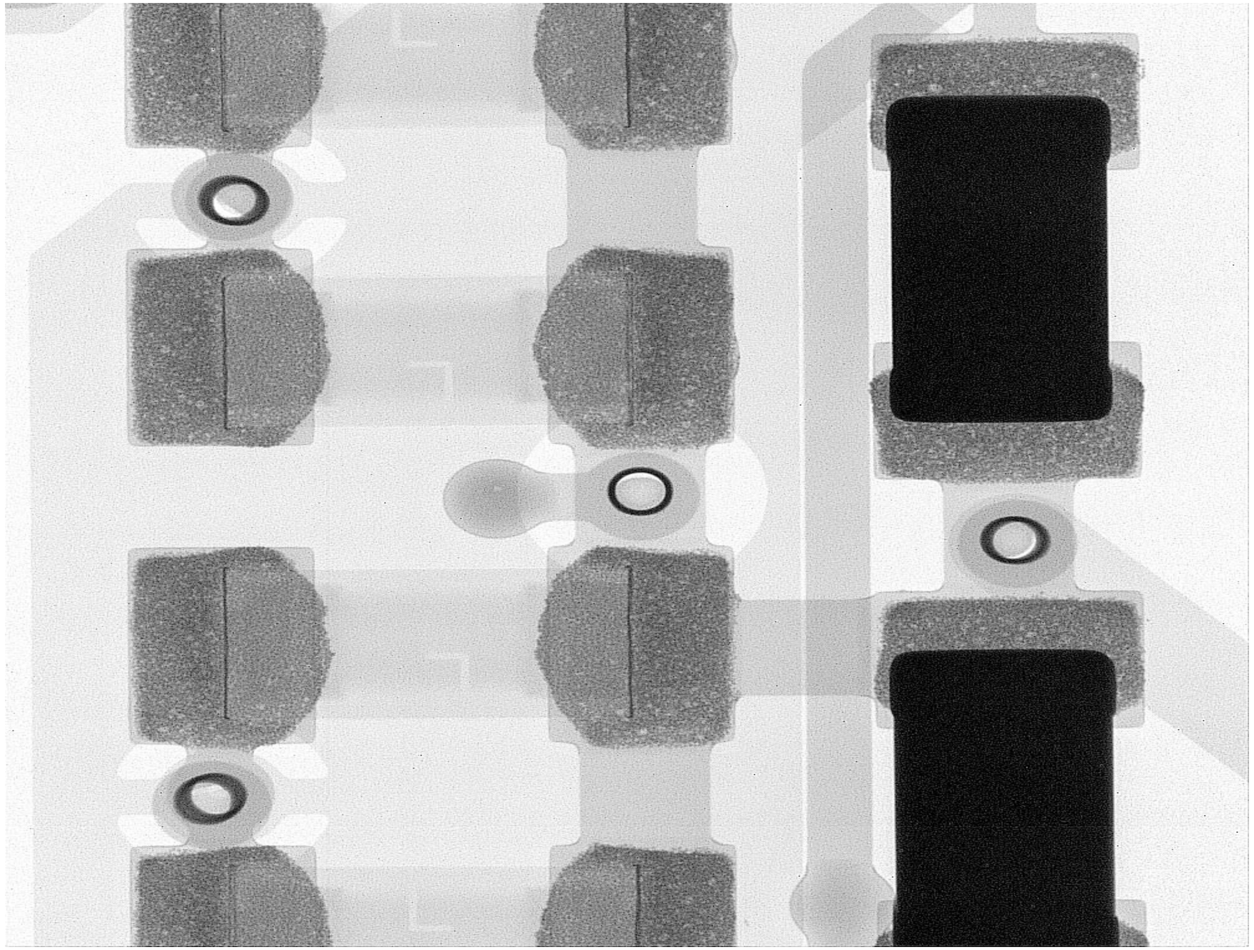
0603 Arrowhead Design



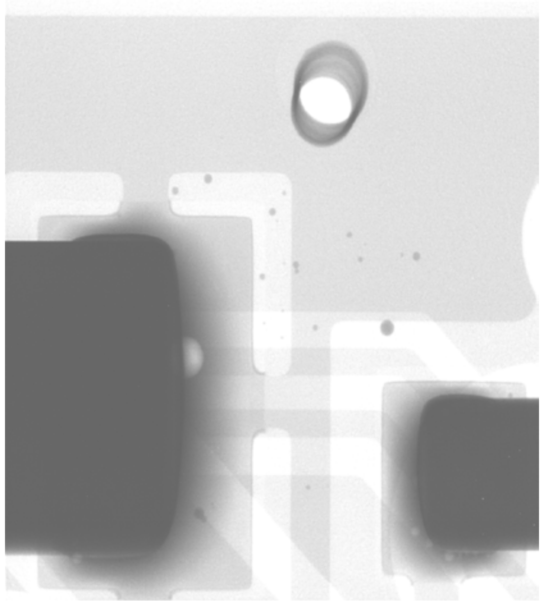
Wendy House or Home Plate

- Better shape as the lack of point on the internal edge reduced the change of solder balling
- However the paste volume is higher with this design and can lead to spattering
- The previous image shows the paste has remained mainly inside the pad geometry
  - Except under the component where there is a slight spread

Wendy House or Home Plate

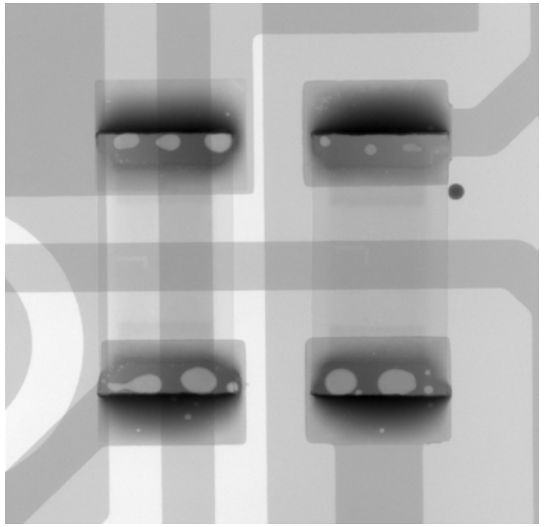


Wendy House after Reflow

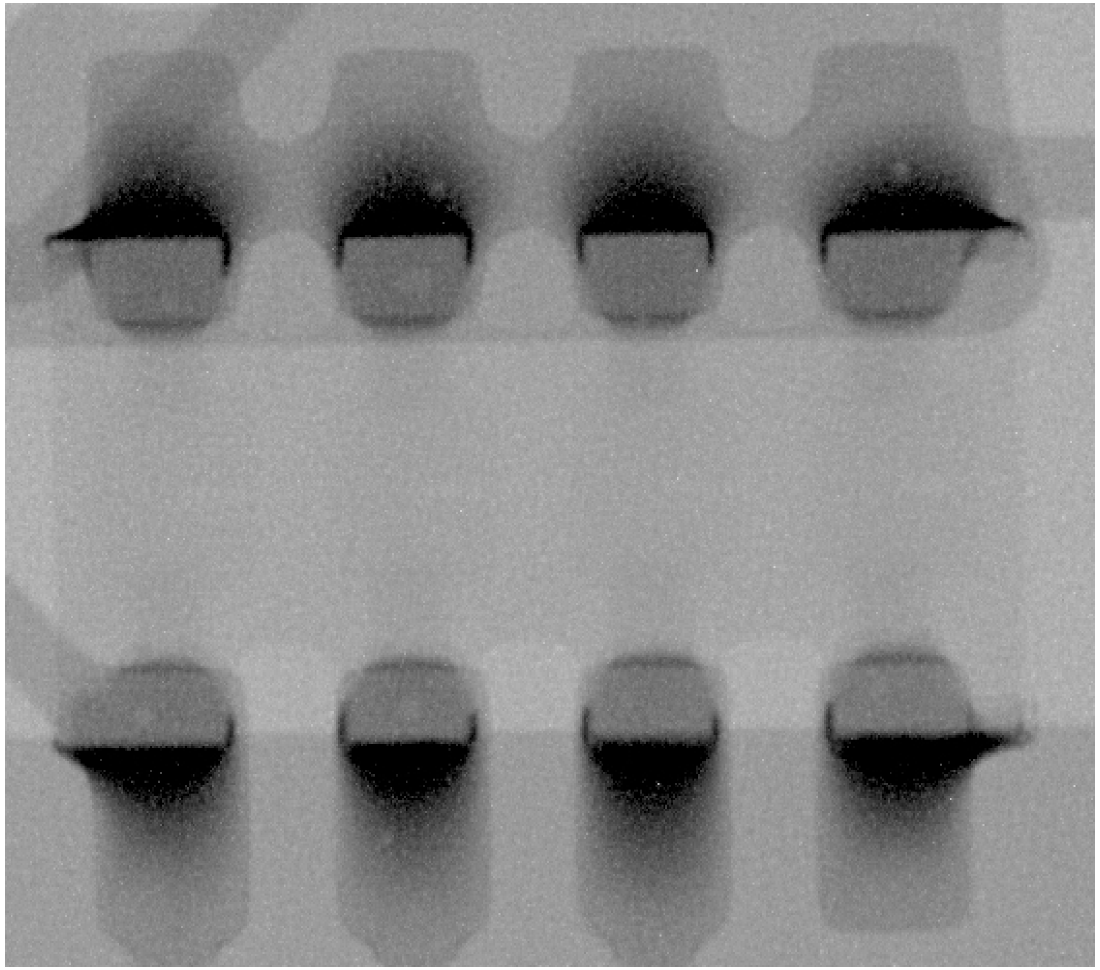


End cap x-ray view showing spatter

X-ray view showing mid chip
Ball and voiding under chips



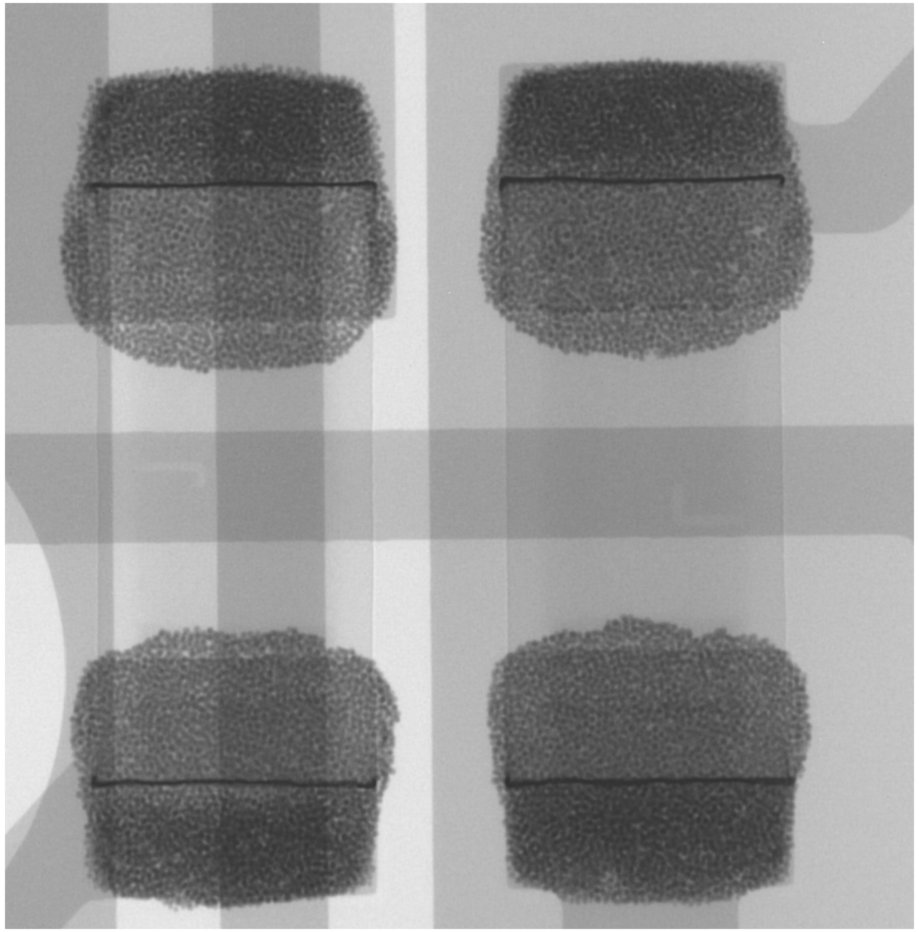
And the winner is….



Reducing the Wendy House design by 5% to reduce paste volume gives the result shown, no solder related issues

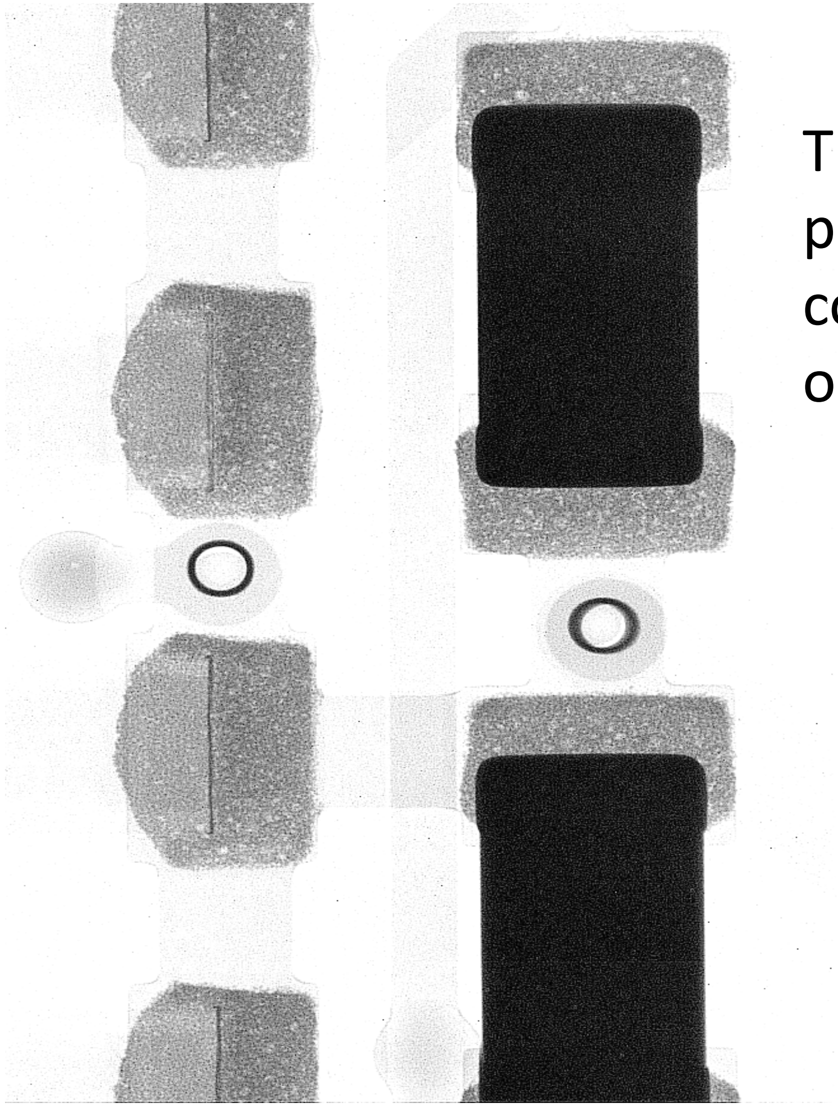
C1206 Components

Again simple pad reduction produced poor results



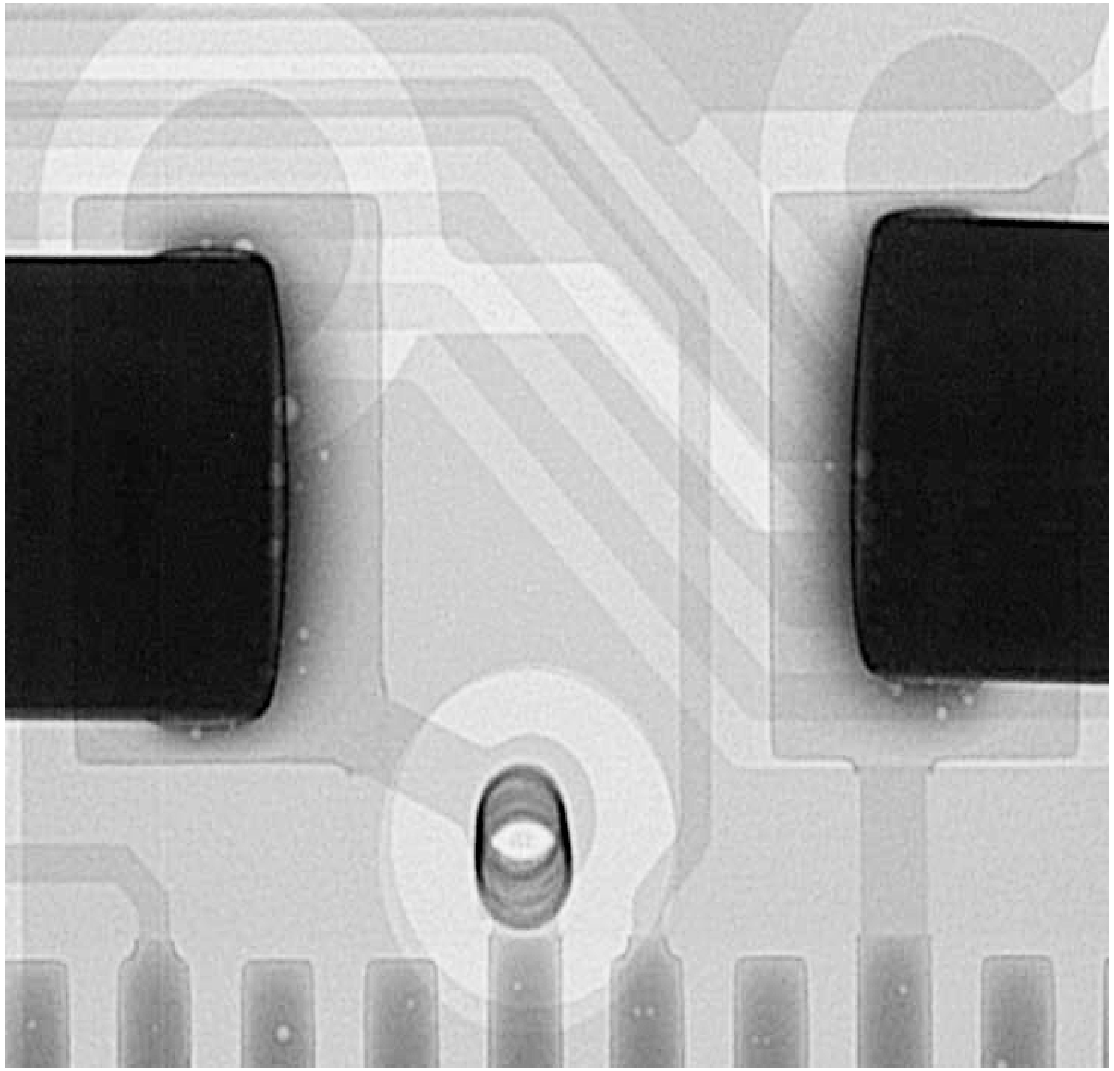
Solder paste spread shown here, plus close pad design makes the situation worse

Wendy House shape



This image contrasts the 0603 paste spread with the large 1206 components, less spread is visible on the 1206 pads

Wendy House 5% reduction

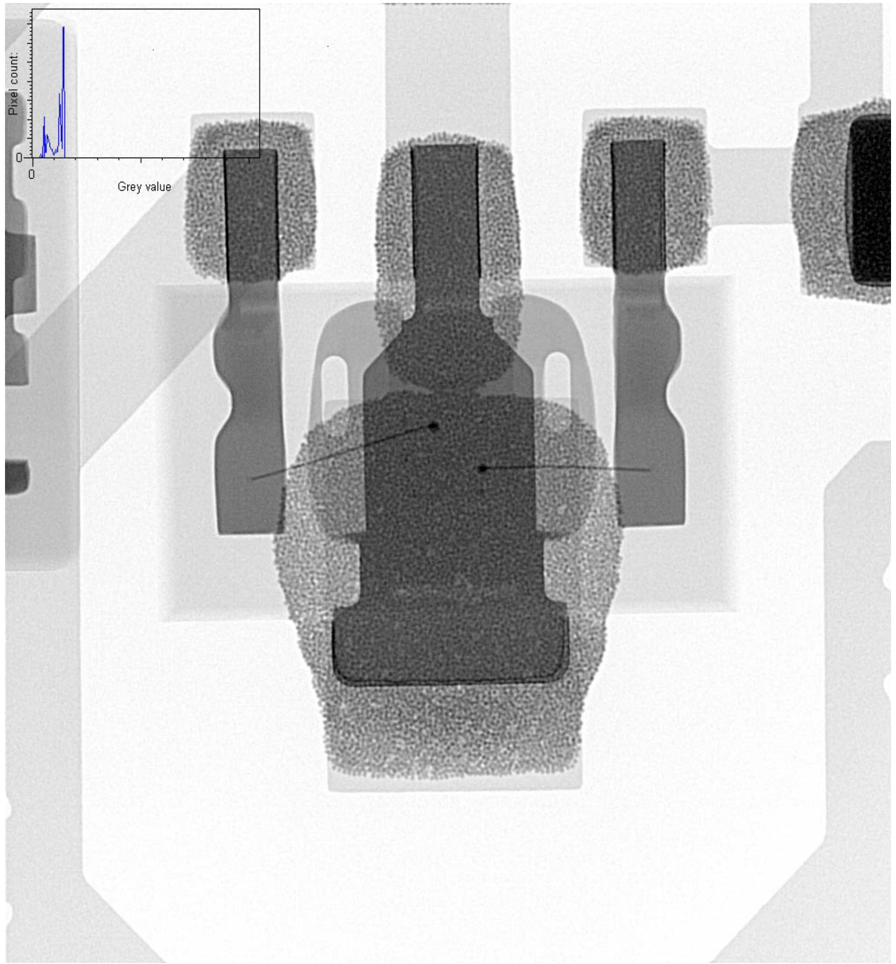


X-Ray image after reflow showing good joint shape and no balling issues

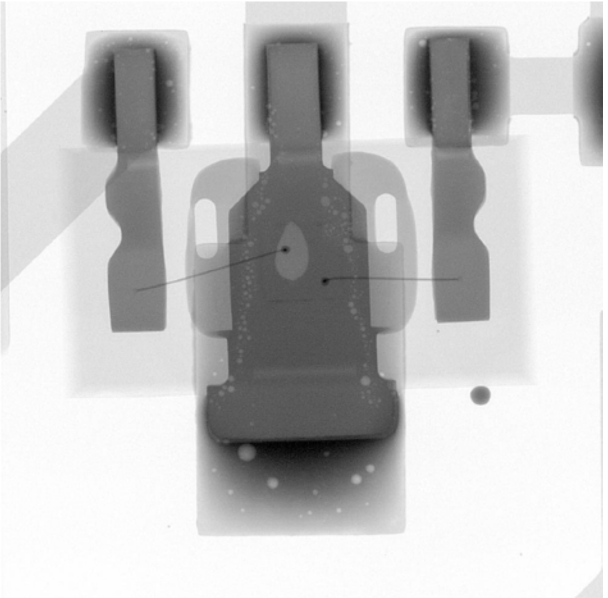
SOT89

- These component were a more complex challenge
  - More legs and larger pads
  - Co-planarity issues across smaller legs
- Flatness issues are normally overcome by increasing paste volume
  - But this leads to solder ball and splatter issues

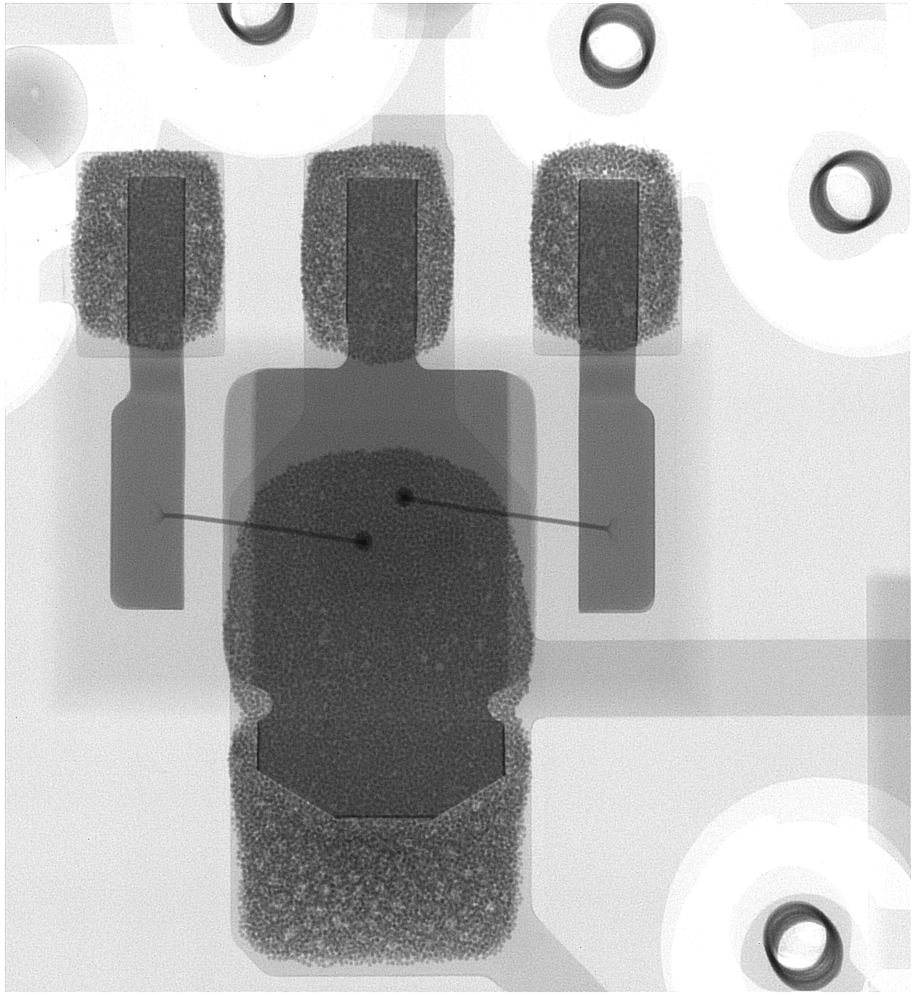
Standard Design



Large paste spread outside the large pad this lead to the solder ball below in the after Reflow image



Final solution



This image was produced of paste using a 10% global reduction on the 3 smaller pads and a Wendy House pad on the Large pad, with the reduced volume under the component.

Results

- Change in First Time Pass Results measured
- Test time was 16 weeks
- Units were Defects per Million Opportunities
  - DPMO is a better metric than PPM as it can measure more than one of the same defect per unit or more than one type of defect
- Changes were implemented in two stages over the 16 weeks

Week	Target	DPMO
W36	150	611
W37	150	693
W38	150	738
W39	150	953
W40	150	742
W41	150	794
W42	150	175
W43	150	363
W44	150	203
W45	150	223
W46	150	102
W47	100	149
W48	100	75
W49	100	67
W50	100	63
W51	100	62
W52	100	83

Stage 1

- Implemented in week 42
  - Modifications for Chip Components implemented
- Results dropped from 794 DPMO in week 41 to 175 DPMO in week 42
- Against a weekly target of 150 DPMO
- So the process in now much better but still below plan

Week	Target	DPMO
W36	150	611
W37	150	693
W38	150	738
W39	150	953
W40	150	742
W41	150	794
W42	150	175
W43	150	363
W44	150	203
W45	150	223
W46	150	102
W47	100	149
W48	100	75
W49	100	67
W50	100	63
W51	100	62
W52	100	83

Stage 2

- Implemented in week 46
- This stencil incorporated the SOT improvements
- DPMO now dropped another 50% to 102
- This is well below the target set
- The target was then reduced to 100 DPMO and the process is delivering double digit results consistently

Week	Target	DPMO
W36	150	611
W37	150	693
W38	150	738
W39	150	953
W40	150	742
W41	150	794
W42	150	175
W43	150	363
W44	150	203
W45	150	223
W46	150	102
W47	100	149
W48	100	75
W49	100	67
W50	100	63
W51	100	62
W52	100	83

Conclusions

- The experiment reduced DPMO for the production line from around 750 to double digits
- Product quality has improved dramatically
- Manufacturing and test costs have been reduced
- Using In-Line X-ray in this innovative way has lead to huge First Time Pass improvements

Acknowledgements

- Yxlon FeinFocus team for the x-ray images
- Company Senior Management Team for supporting and sharing this work
- P. Zarrow, ITM Consulting
- R Willis, EPS Services
- A Weldon, TW Consulting

The Bottom Line!

- Increasing the amount of work done pre production can produce a much more reliable product as high First Time Pass results mean less rework and repair, so less stress is put on the assembly
- Yield improvement is also a major cost reduction and therefore a significant reason for investing this effort ‘up front’

Thank you For your Kind Attention

Keith Bryant, SMT Solutions