APEX EXPO IPC 2017 — TECHNOLOGY'S TURNING POINT — TURN ELECTRONICS MANUFACTURING INSPIRATION INTO INNOVATION

# Improve SMT Assembly Yields Using Root Cause Analysis in Stencil Design

Greg Smith

FCT Assembly, Inc.

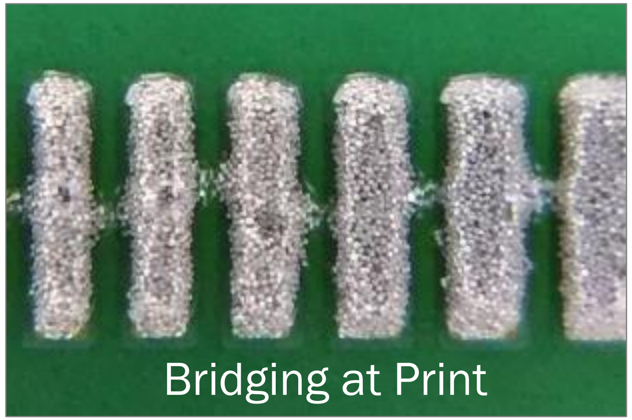
gsmith@fctassembly.com

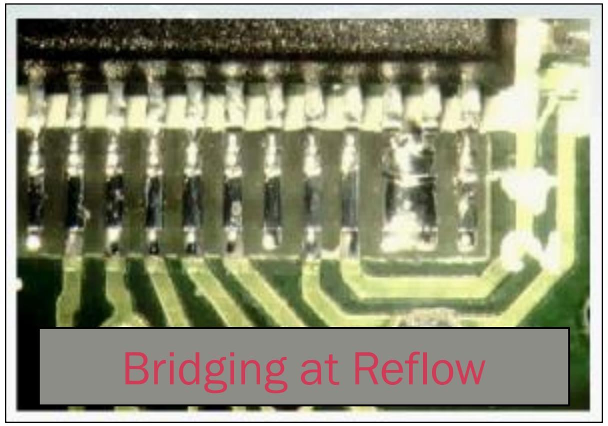
*This paper and presentation was first presented at the 2017 IPC Apex Expo Technical Conference and published in the 2017 Technical Conference Proceedings.*

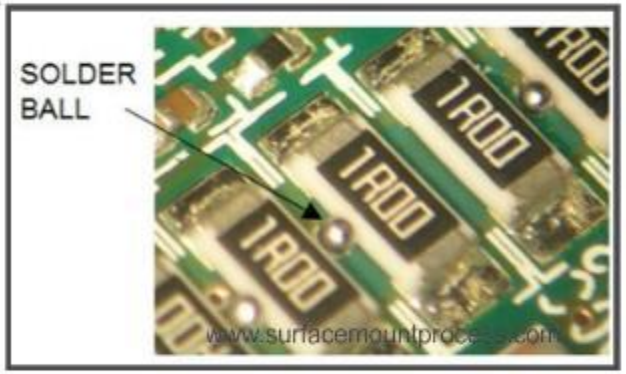
# Outline

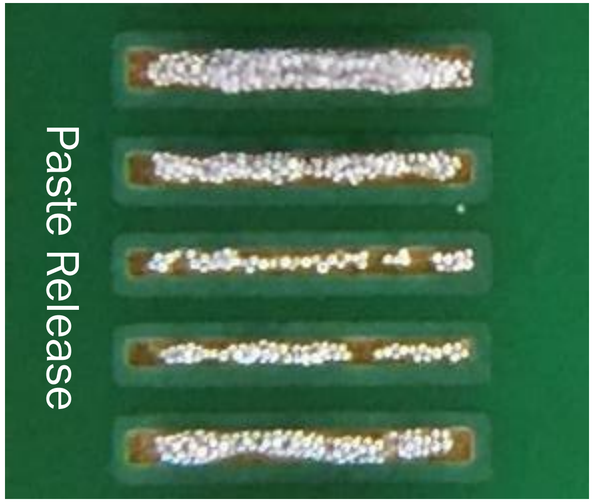
- Introduction
- Root cause analysis to improve solder paste release
- Solder-balls (mid chip solder beads): stencil design to minimize solder balls
- Tombstoning: improving tombstoning with stencil design
- Bridging at print: simple guidelines to eliminate bridging
- Bridging at SMT reflow: what causes bridging after reflow when it is not present after print
- Insufficient solder volume at SMT reflow: the correlation of stencil design to solder volume after reflow for leadless devices.
- Voiding: design ideas to reduce voiding through stencil design
- Conclusions

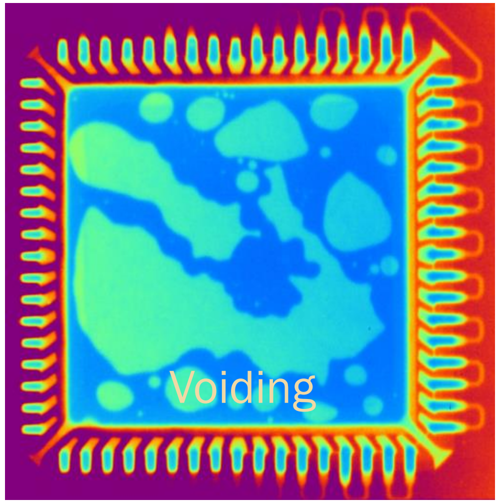
# Introduction

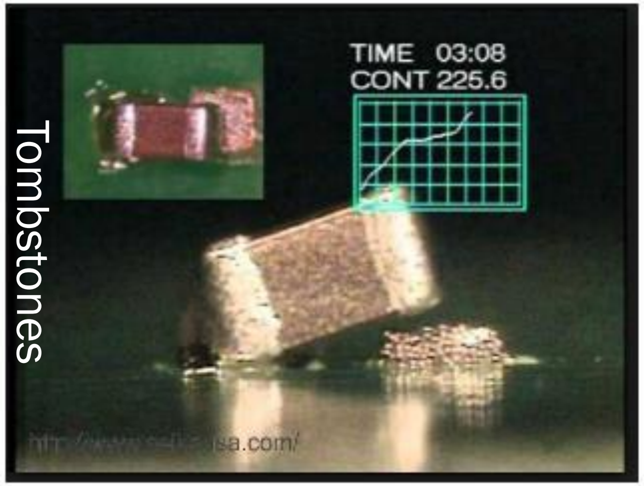


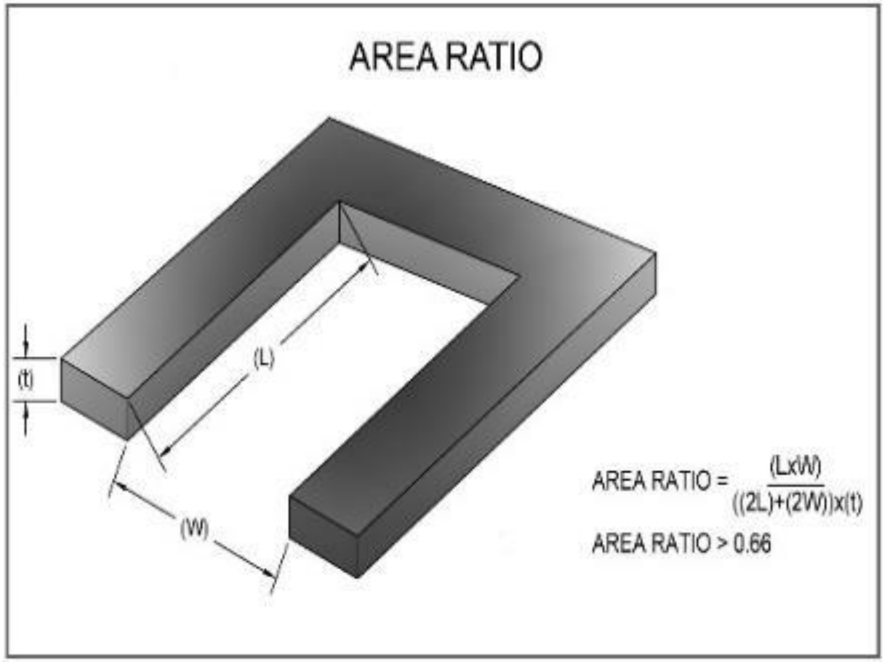



# Root Cause Analysis to Improve Solder Paste Release



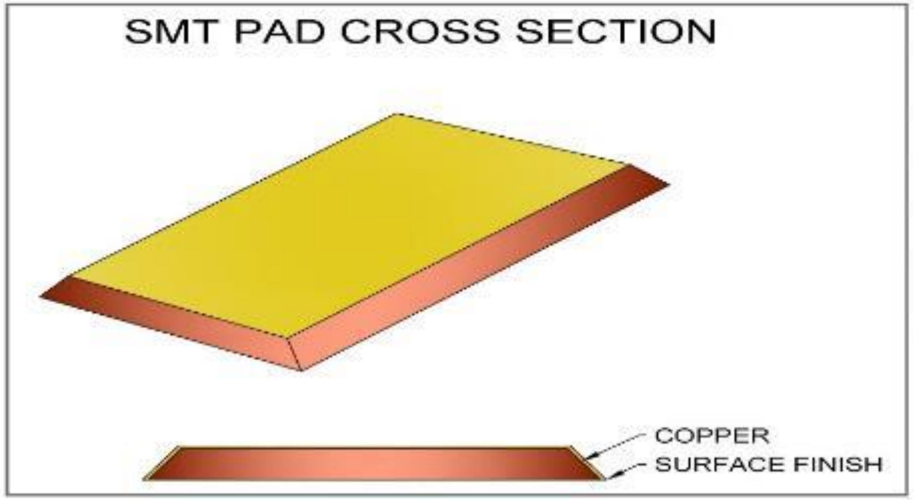

- IPC 7525B recommends 0.66 area ratio for acceptable solder paste release
- Area ratio considers stencil thickness and aperture area
- As components and stencil apertures get smaller, is there a better way to predict solder transfer efficiency?

***Question: Does the size of the PWB SMT land pad affect solder paste release?***

# Root Cause Analysis to Improve Solder Paste Release

- *Does the adhesion force of the solder paste to the actual PWB land pad affect transfer efficiency?*
- *Does the surface area of the actual PWB land pad affect transfer efficiency?*
- *Do the copper weight of the PWB outer layers and surface finish affect PWB land pad sizes? If so, could the actual PWB land size affect transfer efficiency?*

# Root Cause Analysis to Improve Solder Paste Release





- PWB designers modify the lines, traces and pads to allow for etch back
- As copper weights increase PWB land pads become smaller than nominal
- Non-uniform surface coatings such as HASL also affect the contact surface of the PWB land pad to the stencil aperture.

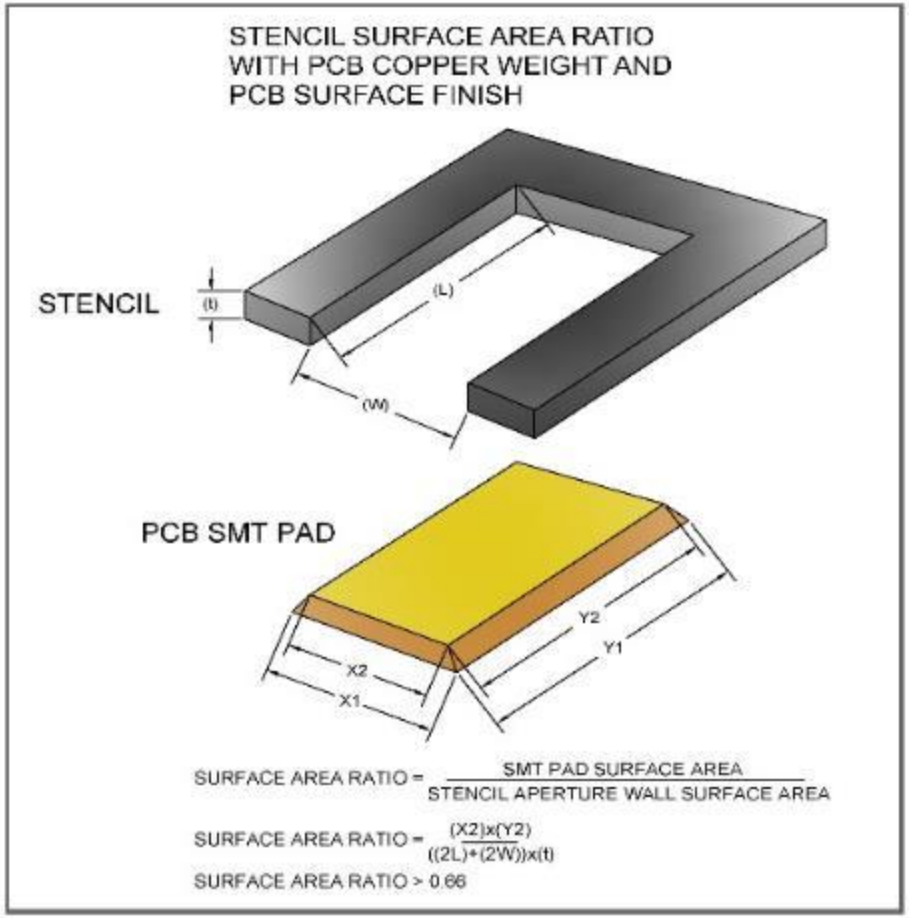
# Root Cause Analysis to Improve Solder Paste Release

## Actual PWB Cross Section Measurement

| Cu Weight | Surface Finish | Surface Measurement (in) | Nominal Trace Width (in) | Difference-Surface vs Nominal (in) | Foot Measurement (in) |
|---|---|---|---|---|---|
| 2 oz | HASL | 0.004173 | 0.0050 | -0.0008 | 0.005906 |
| 2 oz | HASL | 0.004016 | 0.0050 | -0.0010 | 0.005492 |
| 2 oz | ENIG | 0.004874 | 0.0050 | -0.0012 | 0.005911 |
| 1 oz | HASL | 0.004252 | 0.0050 | -0.0007 | 0.005236 |
| 1 oz | HASL | 0.003937 | 0.0050 | -0.0011 | 0.005807 |
| 1 oz | ENIG | 0.004823 | 0.0050 | -0.0002 | 0.005906 |
| 1 oz | ENIG | 0.004991 | 0.0050 | -0.0000 | 0.006063 |

Should a new modified “surface area ratio” be used to predict sufficient transfer efficiency of the solder paste for small component printing?

# Root Cause Analysis to Improve Solder Paste Release




- The modified comparison (***surface area ratio***) uses actual SMT pad surface
- Heavier copper weights will produce larger reductions at the surface
- Flat surface finishes, ENIG and OSP, do not change the size of the surface of the SMT pad.
- Non-flat surface finishes, like HASL, have a more domed/irregular surface. When HASL surface finishes are used, an additional reduction is made to calculate the surface area ratio.

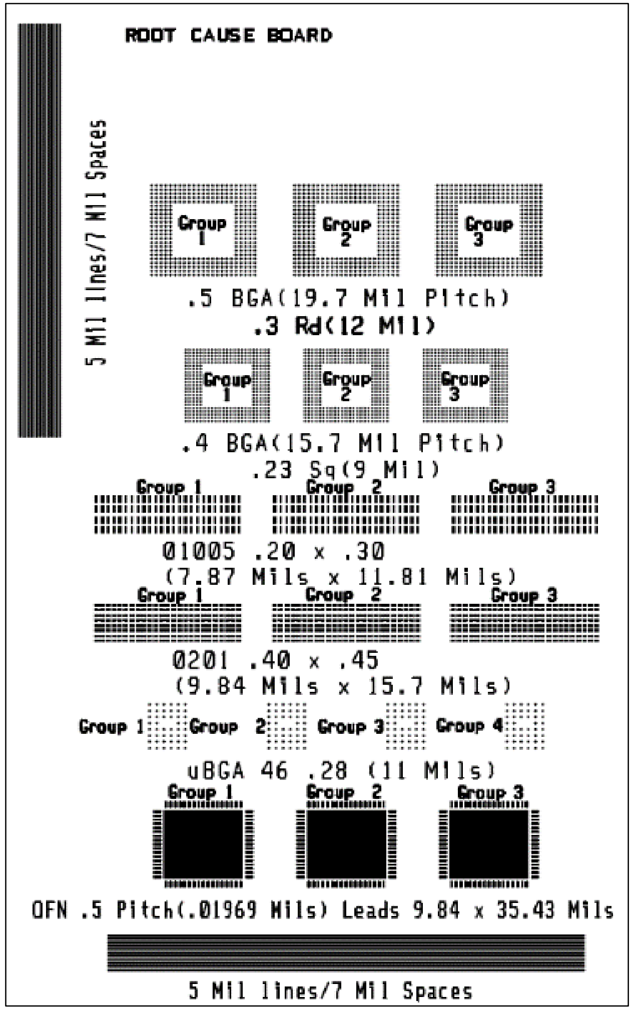
# Root Cause Analysis to Improve Solder Paste Release

| Copper Weight (oz) | Copper Thickness (µm) | Size Reduction mm (inches) | Size Reduction mm (inches) with ENIG, OSP, Ag, Sn | Size Reduction mm (inches) with HASL & HAL |
|---|---|---|---|---|
| **0.5** | 17.5 | 0 | 0 | -0.0008 |
| **1** | 35 | -0.0001 | 0 | -0.0008 |
| **2** | 70 | -0.0002 | 0 | -0.0008 |

Utilizing these SMT pad size reductions may provide a more realistic representation of what SMT pad sizes to expect on physical PCBs.  The modified surface area ratio will be used to test this theory.

| Component | Copper Weight | Surface Finish | PWB Pad Size | Stencil Aperture Size | Area Ratio | Surface Area Ratio |
|---|---|---|---|---|---|---|
| **.4 BGA** | 2 oz | HASL | 9 mil Round | 9 mil Round | 0.56 | 0.44 |
| **.4 BGA** | 2 oz | HASL | 9 mil Round | 11 mil Sq, 2 mil Radius | 0.69 | 0.57 |
| **uBGA** | 2 oz | HASL | 11 mil Round | 9 mil Sq, 2 mil Radius | 0.56 | 0.69 |
| **uBGA** | 2 oz | HASL | 11 mil Round | 11 mil Round | 0.69 | 0.57 |
| **uBGA** | 2 oz | HASL | 11 mil Round | 13 mil Sq, 2 mil Radius | 0.81 | 0.48 |

# Root Cause Analysis to Improve Solder Paste Release




- Stencil aperture design for each of the component shapes are shown below. Two 4 mil thick stencils were used.
- One stencil was uncoated and a second stencil was coated with a Flouro-Polymer Nano Coating (FPN).

| Component | Group | PWB Pad Size | Stencil Aperture Size |
|---|---|---|---|
| .5 BGA | 1 | 12 mil Round | 10 mil Sq, 2 mil Radius |
| .5 BGA | 2 | 12 mil Round | 12 mil Round |
| .5 BGA | 3 | 13 mil Round | 14 mil Sq, 2 mil Radius |
| .4 BGA | 1 | 9 mil Round | 7 mil Sq, 2 mil Radius |
| .4 BGA | 2 | 9 mil Round | 9 mil Round |
| .4 BGA | 3 | 9 mil Round | 11 mil Sq, 2 mil Radius |
| 01005 | 1 | 7.9 x 11.8 mils | 5.9 x 9.8 , 2 mil Radius |
| 01005 | 2 | 7.9 x 11.8 mils | 7.9 x 11.8, 2 mil Radius |
| 01005 | 3 | 7.9 x 11.8 mils | 7.9 x 9, 2 mil Radius |
| 0201 | 1 | 15.7 x 9.8 mils | 13.7 x 7.8, 2 mil Radius |
| 0201 | 2 | 15.7 x 9.8 mils | 15.7 x 9.8, 2 mil Radius |
| 0201 | 3 | 15.7 x 9.8 mils | 14.7 x 8.8, 2 mil Radius |
| uBGA | 1 | 11 mil Round | 9 mil Sq, 2 mil Radius |
| uBGA | 2 | 11 mil Round | 11 mil Round |
| uBGA | 3 | 11 mil Round | 13 mil Sq, 2 mil Radius |
| uBGA | 4 | 11 mil Round | 15 mil Sq, 2 mil Radius |
| QFN | 1 | 35.4 x 9.8 Oblong | 55.4 x 9.8, Oblong |
| QFN | 2 | 35.4 x 9.8 Oblong | 35.4 x 9.8, Oblong |
| QFN | 3 | 35.4 x 9.8 Oblong | 65.4 x 7.8, Oblong |

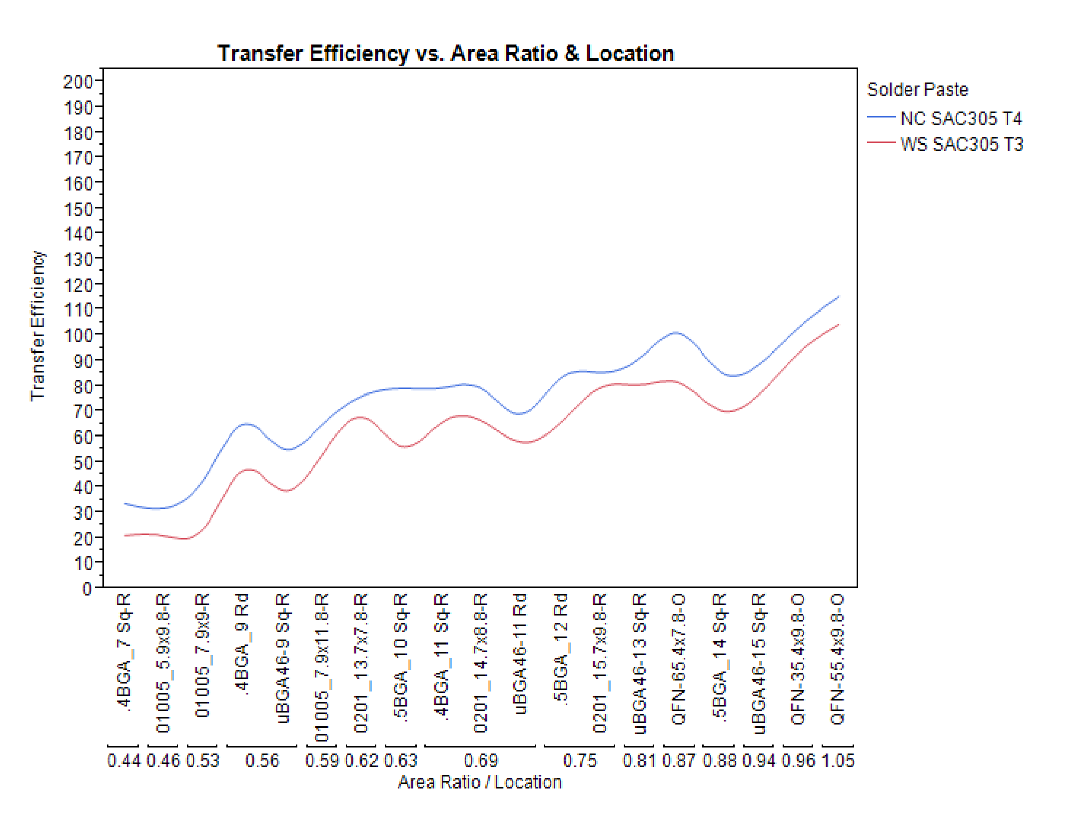
# Root Cause Analysis to Improve Solder Paste Release

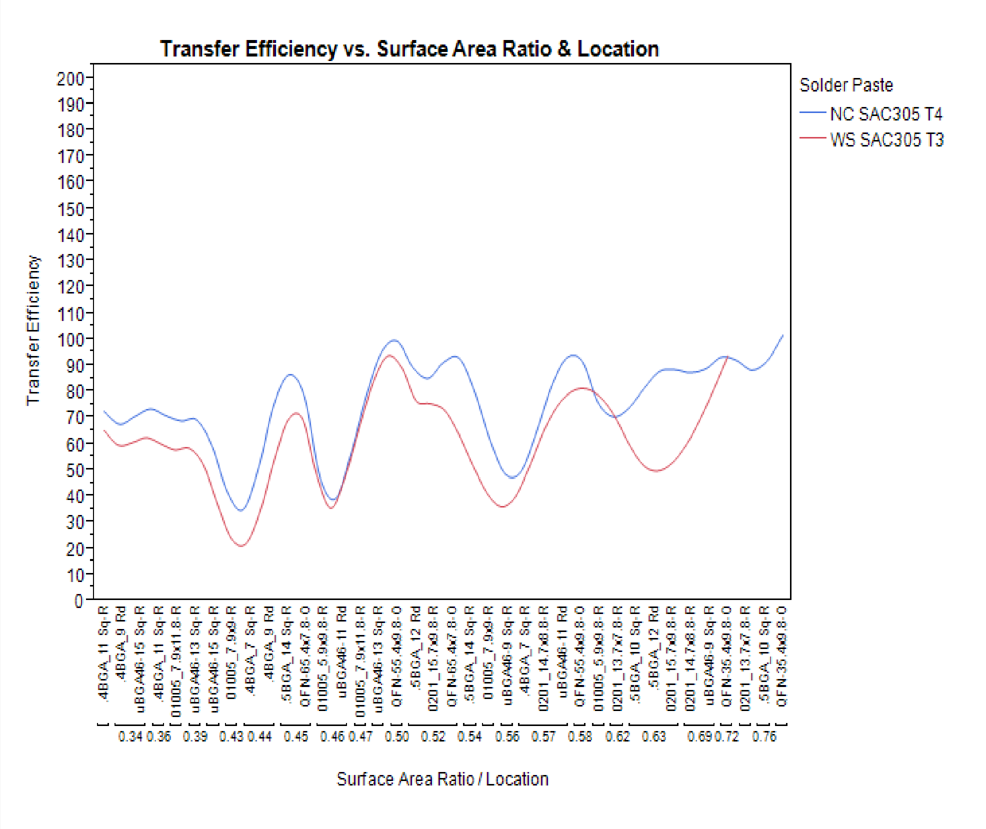
Experiment:

- 5 Boards, 2 oz Cu, HASL printed with No Clean SAC305, Type 4 Paste
- 5 Boards, 2 oz Cu, HASL printed with Water Soluble SAC305, Type 3 Paste
- 5 Boards, 1 oz Cu, ENIG printed with No Clean SAC305, Type 4 Paste
- 5 Boards, 1 oz Cu, HASL printed with No Clean SAC305, Type 4 Paste

All solder paste bricks measured using 3D, SPI to obtain volume and calculate transfer efficiency

# Root Cause Analysis to Improve Solder Paste Release

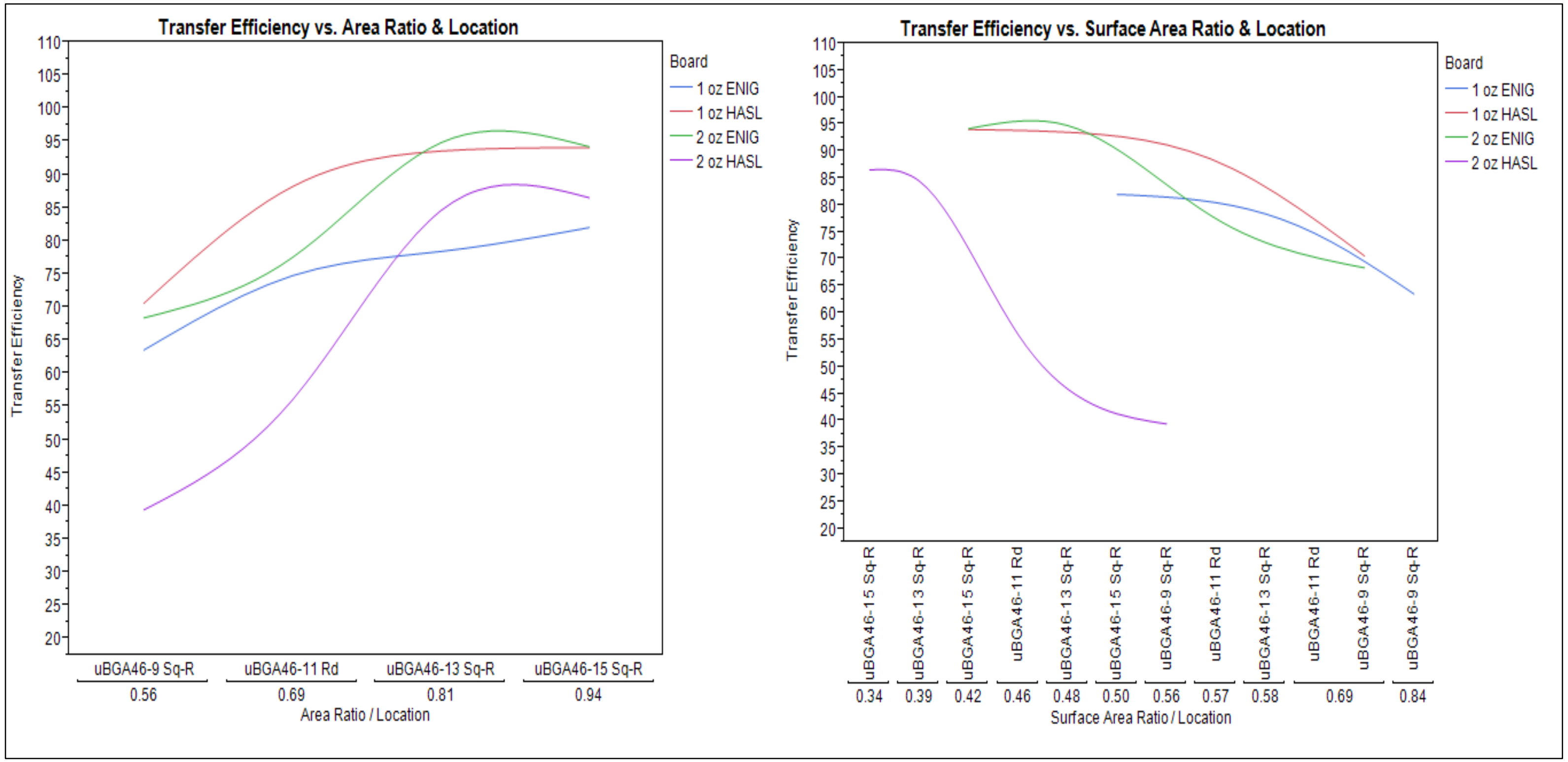



- As area ratio increases, transfer efficiency increases in a linear fashion
- As **surface area ratio** increases, transfer efficiency does not increase in a linear fashion

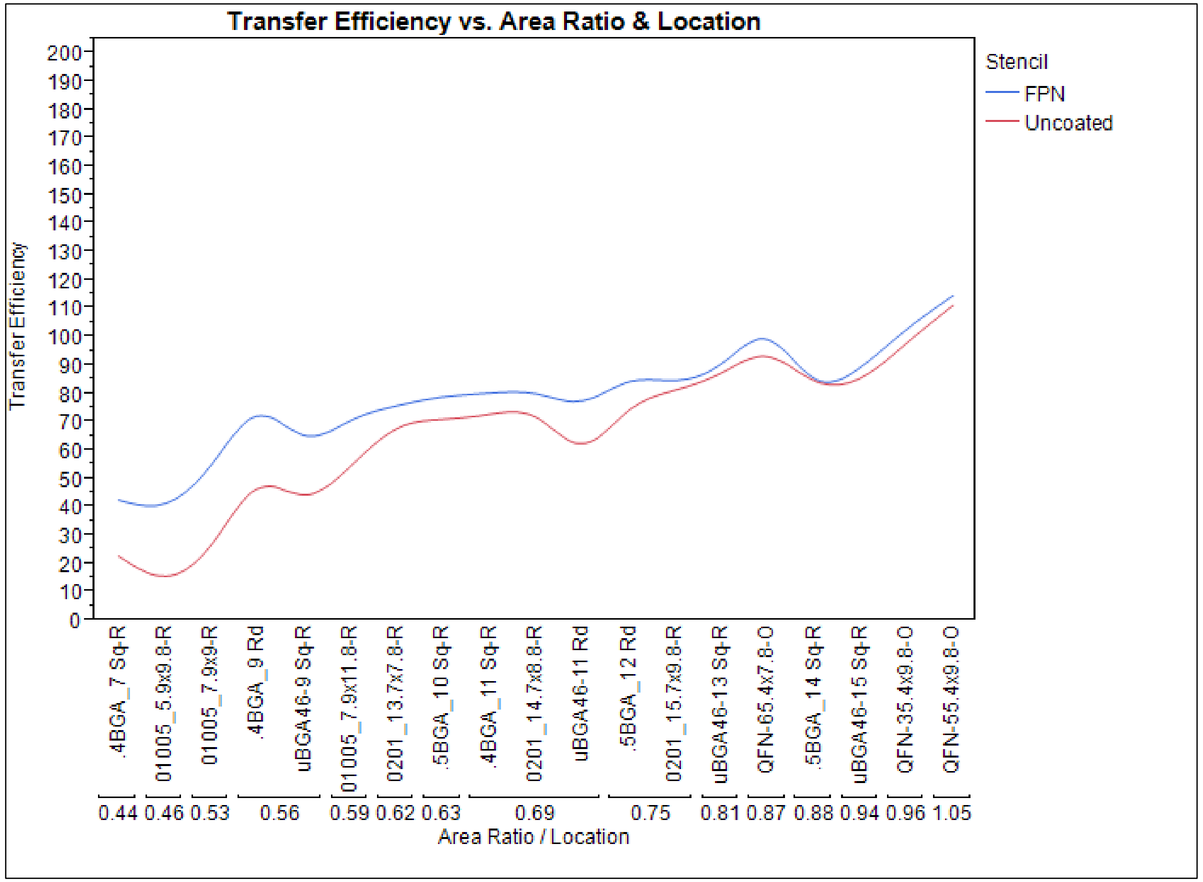
# Root Cause Analysis to Improve Solder Paste Release

## 46 Pin uBGA Data




Calculating surface area ratio based on copper weight of the outer layer and the surface finish on the board is not a good method to predict transfer efficiency.

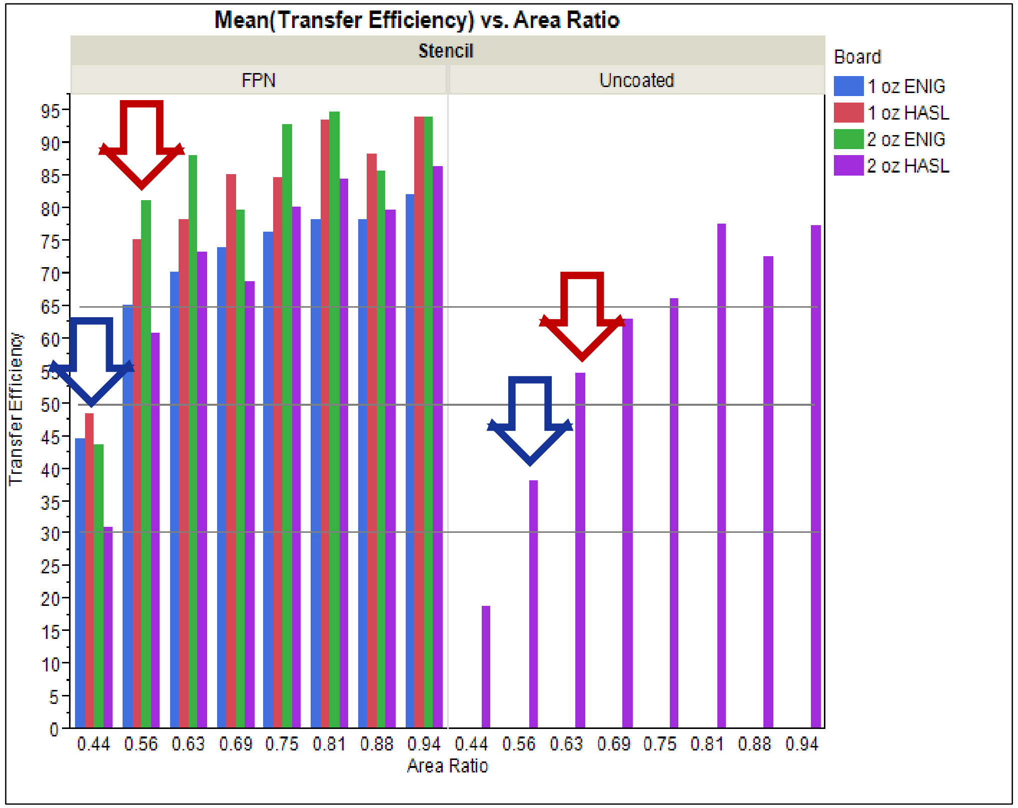
# Root Cause Analysis to Improve Solder Paste Release




**Results and Recommendations for Improving Paste Release**

- Area Ratio is best method to predict Transfer Efficiency
- FPN coatings allow lower Area Ratios to achieve same Transfer Efficiency

# Root Cause Analysis to Improve Solder Paste Release

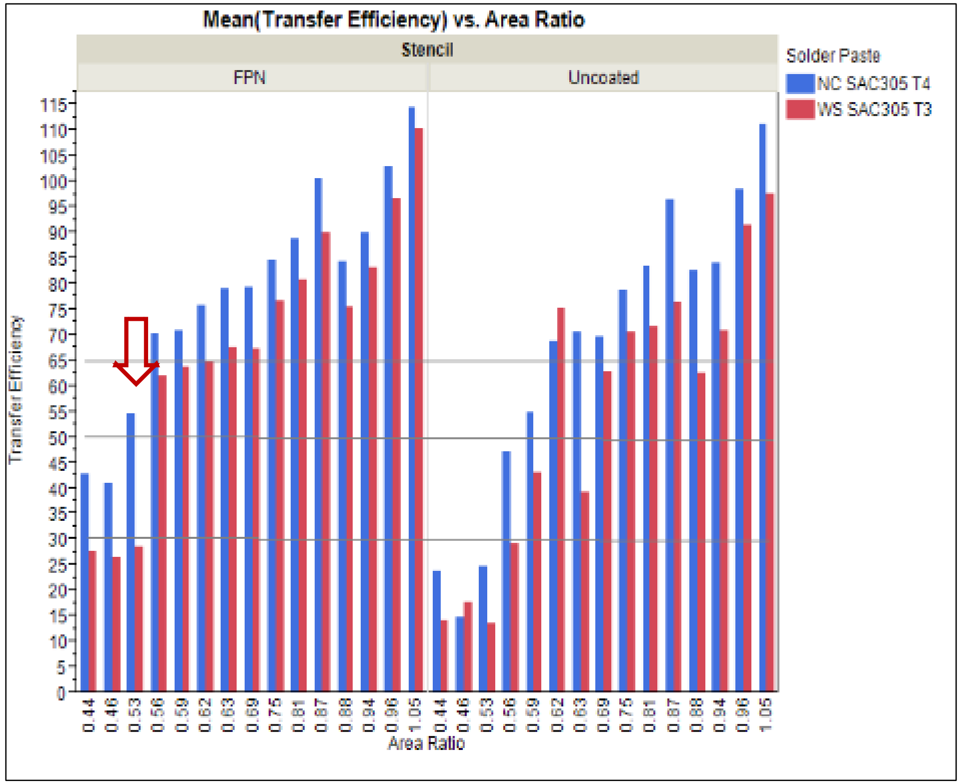



## Results and Recommendations for Improving Paste Release-Small Apertures

According to the results shown here:
(4 Mil Stencil; No Clean SAC 305 T4 Paste, All board types)

- **50% Transfer Efficiency**
  - Uncoated: Approx. 0.60 area ratio
  - FPN Coated: Approx. 0.50 area ratio
- **30% Transfer Efficiency**
  - Uncoated: Approx. 0.55 area ratio
  - FPN Coated: Approx. 0.45 area ratio

# Root Cause Analysis to Improve Solder Paste Release

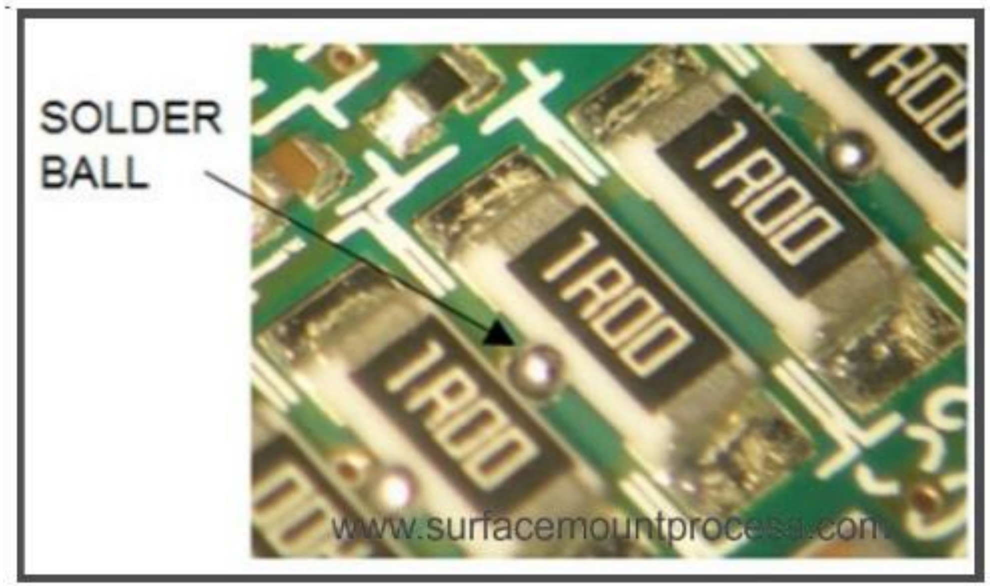



## Results and Recommendations for Improving Paste Release-Small Apertures

Another indicator of transfer efficiency is solder paste:

- Paste type and powder size play a role in transfer efficiency
- Looking at FPN data, 0.53 area ratio component has 55% TE with NC Type 4, WS Type 3 paste releases at 28% TE
- This data is dependent on the actual brand and product of solder paste being used. In order to perform precise root cause analysis to optimize paste release one must investigate their specific solder paste

# Solder-balls (Mid Chip Solder Beads): Stencil design to minimize solder balls

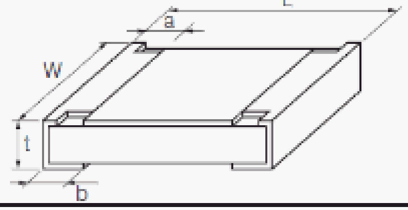



- Solder balls on two pin chips-appear after reflow
- Can be prevented or greatly reduced by stencil design.
- Root cause analysis of solder ball defects
  - Compare design of stencil apertures to actual package
  - Stencil apertures must fall in the correct location vs component leads

# Solder-balls (Mid Chip Solder Beads): Stencil design to minimize solder balls

The actual dimensions of part package are identified

■ Dimensions in mm (not to scale)

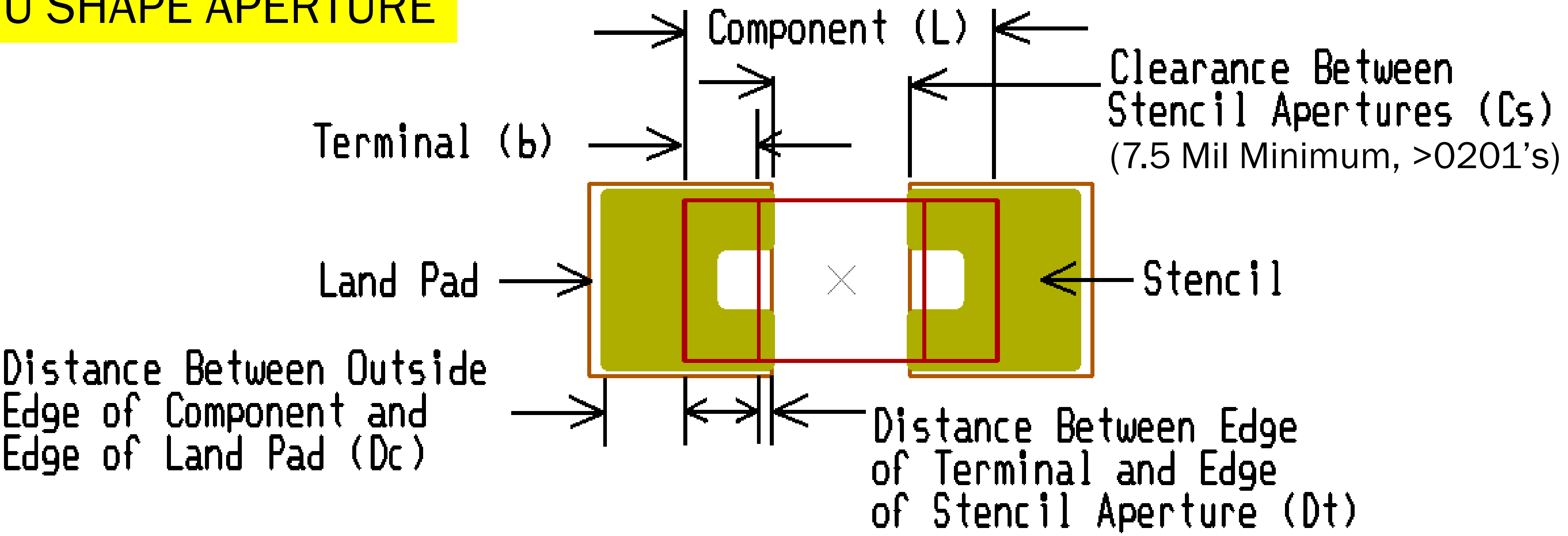



| Type (inches) | Dimensions (mm) | | | | | Mass (Weight) (g/1000 pcs.) |
|---|---|---|---|---|---|---|
| | L | W | a | b | t | |
| ERJXG (01005)[*] | $0.40^{\pm0.02}$ | $0.20^{\pm0.02}$ | $0.10^{\pm0.03}$ | $0.10^{\pm0.03}$ | $0.13^{\pm0.02}$ | 0.04 |
| ERJ1G (0201) | $0.60^{\pm0.03}$ | $0.30^{\pm0.03}$ | $0.10^{\pm0.05}$ | $0.15^{\pm0.05}$ | $0.23^{\pm0.03}$ | 0.15 |
| ERJ2G (0402) | $1.00^{\pm0.05}$ | $0.50^{\pm0.05}$ | $0.20^{\pm0.10}$ | $0.25^{\pm0.05}$ | $0.35^{\pm0.05}$ | 0.8 |
| ERJ3G (0603) | $1.60^{\pm0.15}$ | $0.80^{+0.15}_{-0.05}$ | $0.30^{\pm0.20}$ | $0.30^{\pm0.15}$ | $0.45^{\pm0.10}$ | 2 |
| ERJ6G (0805) | $2.00^{\pm0.20}$ | $1.25^{\pm0.10}$ | $0.40^{\pm0.20}$ | $0.40^{\pm0.20}$ | $0.60^{\pm0.10}$ | 4 |
| ERJ8G (1206) | $3.20^{+0.05}_{-0.20}$ | $1.60^{+0.05}_{-0.15}$ | $0.50^{\pm0.20}$ | $0.50^{\pm0.20}$ | $0.60^{\pm0.10}$ | 10 |
| ERJ14 (1210) | $3.20^{\pm0.20}$ | $2.50^{\pm0.20}$ | $0.50^{\pm0.20}$ | $0.50^{\pm0.20}$ | $0.60^{\pm0.10}$ | 16 |
| ERJ12 (1812) | $4.50^{\pm0.20}$ | $3.20^{\pm0.20}$ | $0.50^{\pm0.20}$ | $0.50^{\pm0.20}$ | $0.60^{\pm0.10}$ | 27 |
| ERJ12Z (2010) | $5.00^{\pm0.20}$ | $2.50^{\pm0.20}$ | $0.60^{\pm0.20}$ | $0.60^{\pm0.20}$ | $0.60^{\pm0.10}$ | 27 |
| ERJ1T (2512) | $6.40^{\pm0.20}$ | $3.20^{\pm0.20}$ | $0.65^{\pm0.20}$ | $0.60^{\pm0.20}$ | $0.60^{\pm0.10}$ | 45 |

# Solder-balls (Mid Chip Solder Beads): Stencil design to minimize solder balls

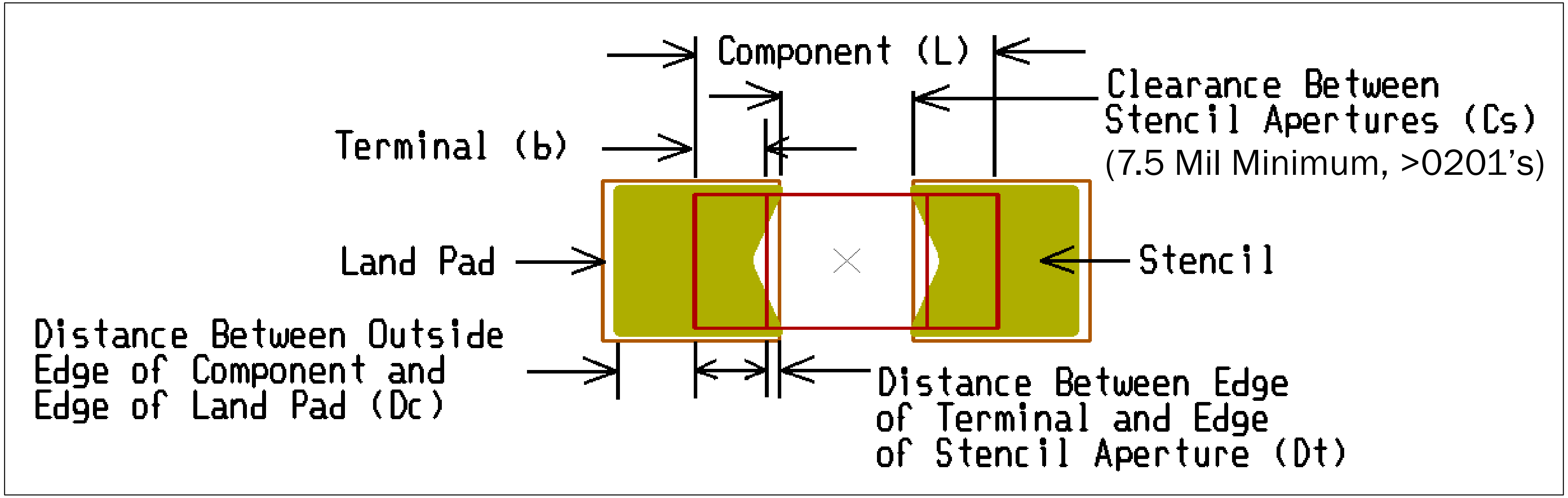
The chip design must be overlaid onto the board land pads using a Gerber editor.

U SHAPE APERTURE



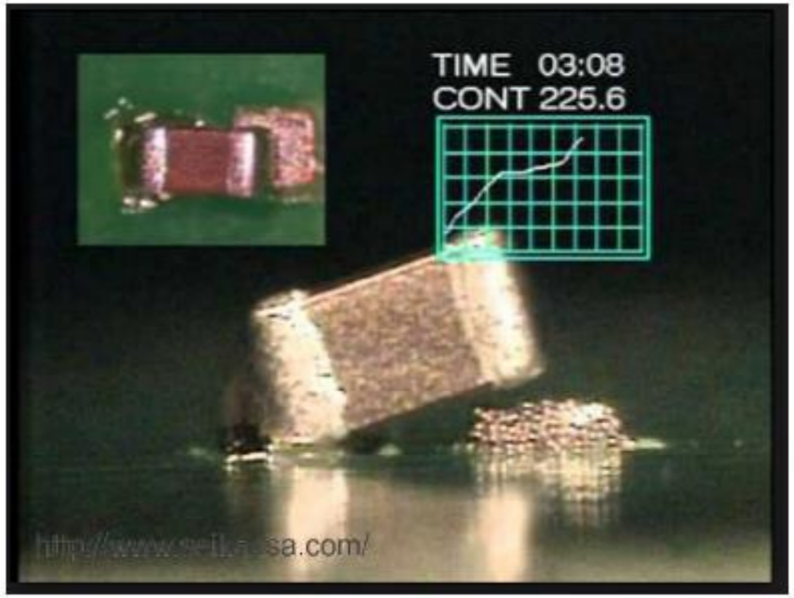
# Solder-balls (Mid Chip Solder Beads): Stencil design to minimize solder balls

The chip design must be overlaid onto the board land pads using a Gerber editor.




*ALWAYS CALCULATE AREA RATIO WHEN CHANGING TO U SHAPE OR INVERTED HOMEPLATE APERTURES*

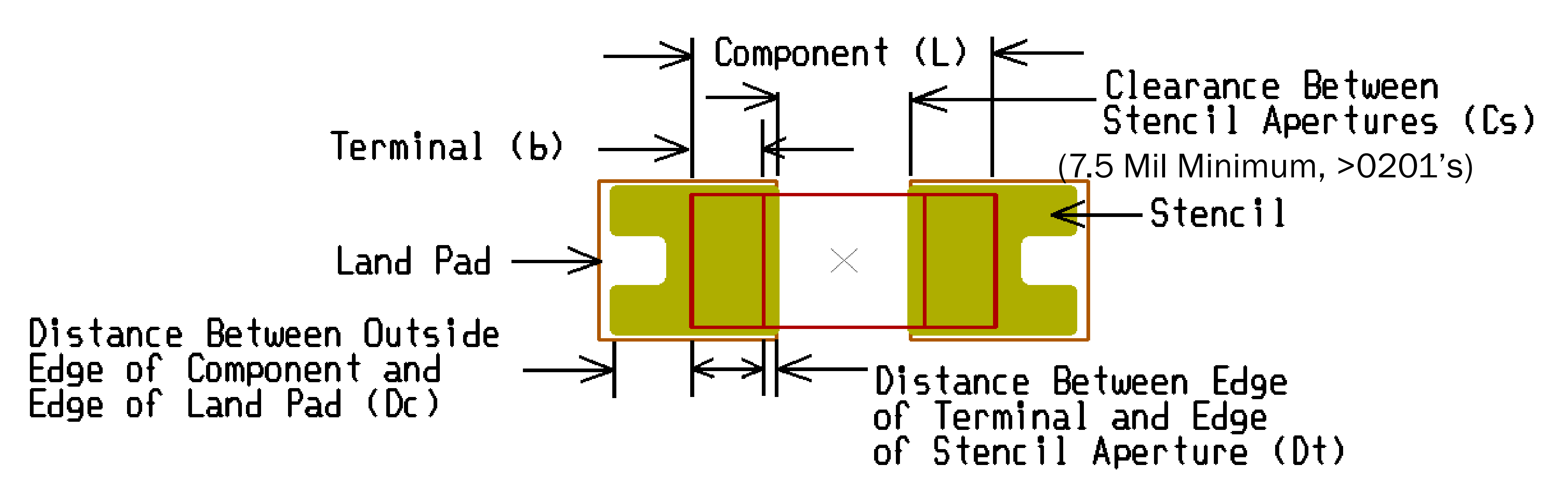
# Tombstoning: Improving tombstoning with stencil design




- Appears after reflow.
- Can be corrected through stencil design.
- Using root cause analysis to prevent tombstones is similar to eliminating solder balls thru stencil design.
- Outside of chip component should fall at midpoint of stencil apertures/PWB land pad.
- Large stencil apertures/PWB land pads that extend further than half the length of component can allow solder to go liquidous on one side before the other allowing tombstoning.

# Tombstoning: Improving tombstoning with stencil design

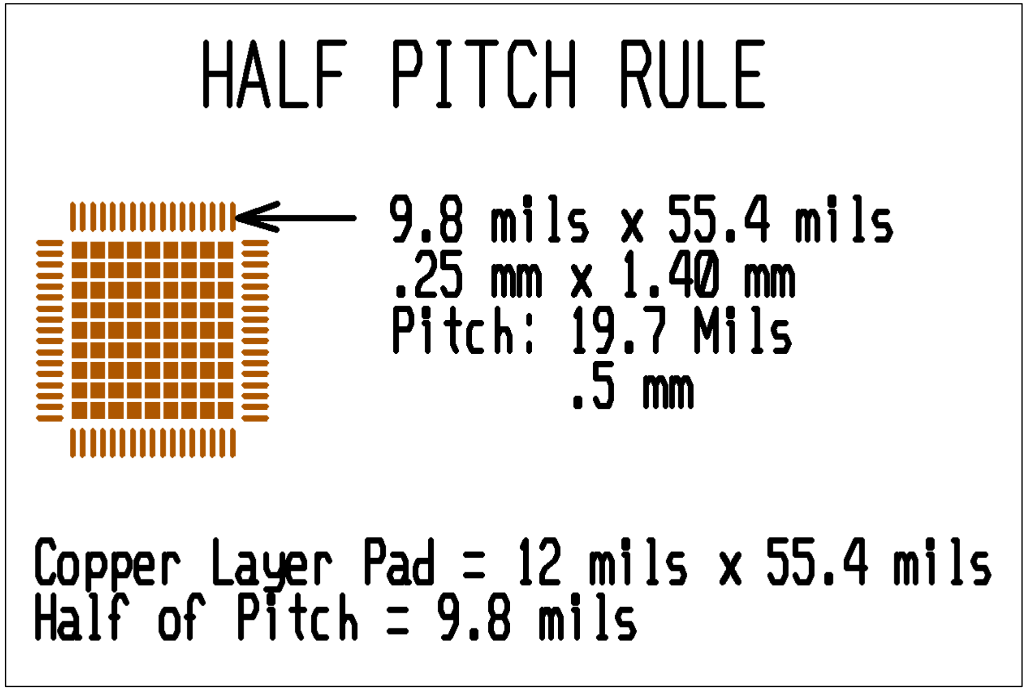
The chip design must be overlaid onto the board land pads using a Gerber editor.




*ALWAYS CALCULATE AREA RATIO WHEN CHANGING TO U SHAPE OR INVERTED HOMEPLATE APERTURES*

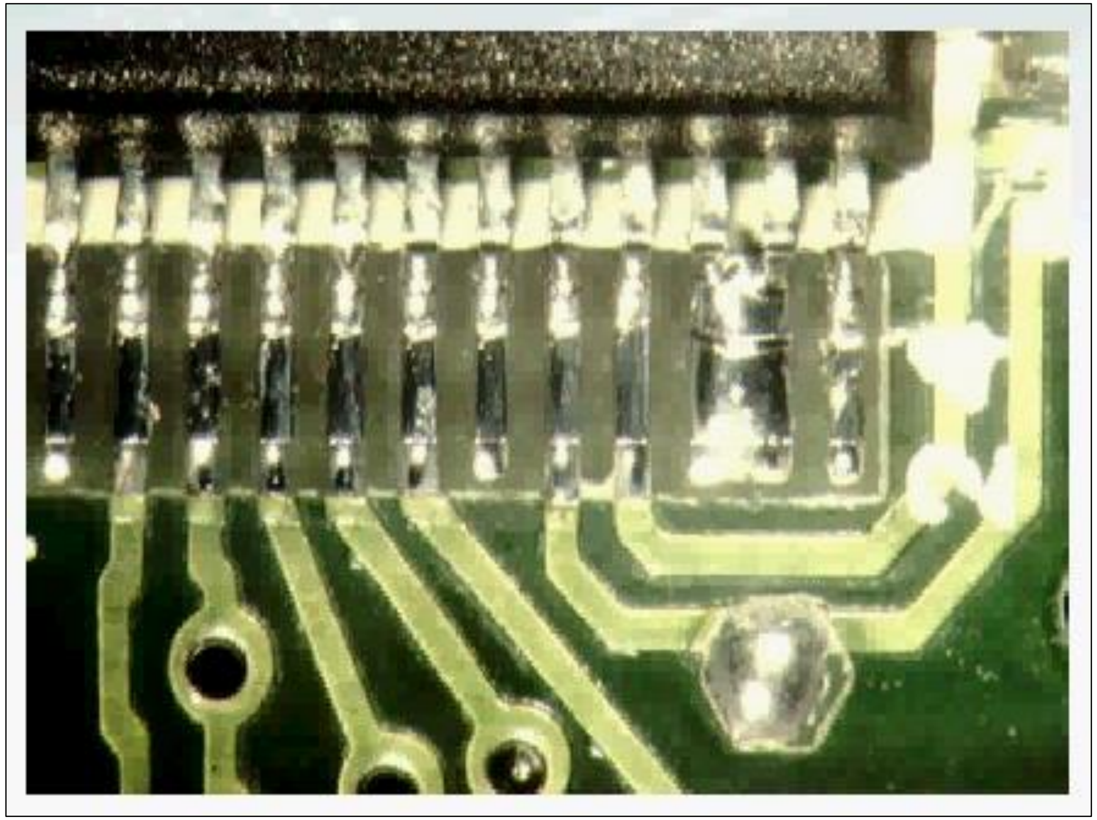
# Bridging at Print: Simple guidelines to eliminate bridging

*There are no guarantees for good board design.*



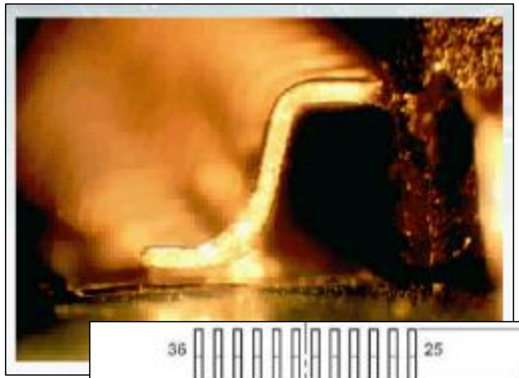

- Use "Half Pitch Rule" to determine aperture width and leave length 1:1 with land pad.
- If land pad width is less than half pitch, leave aperture 1:1 with land pad.
- On apertures larger than 33 mil (.84mm) pitch, reduce apertures to half pitch up to 20% maximum reduction.
- Use area ratio rule to determine acceptable thickness for stencil foil.

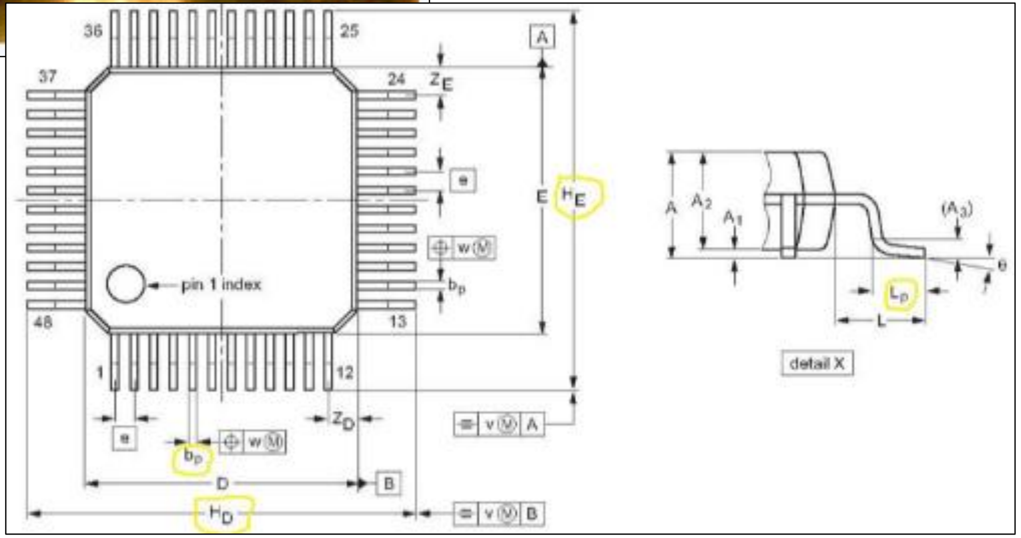
# Bridging at SMT Reflow: What causes bridging after reflow when it is not present after print



Most often seen on Gull Wing, QFP devices

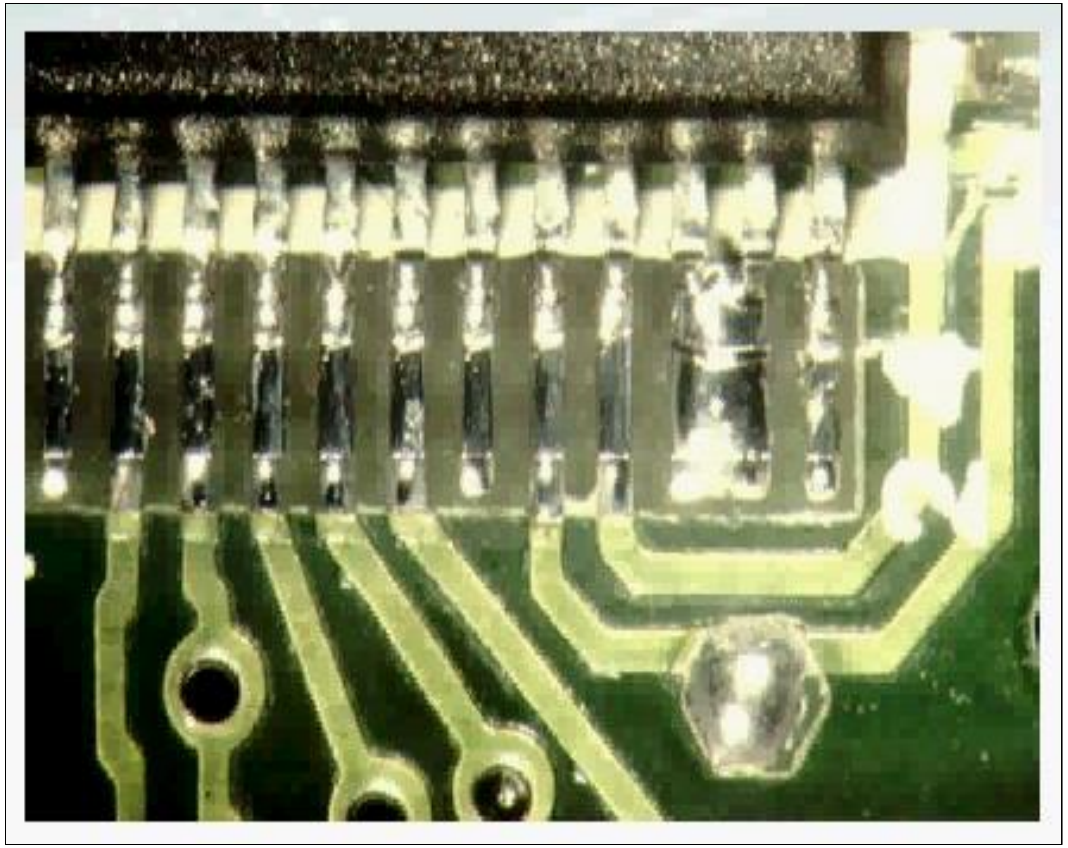
# Bridging at SMT Reflow: What causes bridging after reflow when it is not present after print





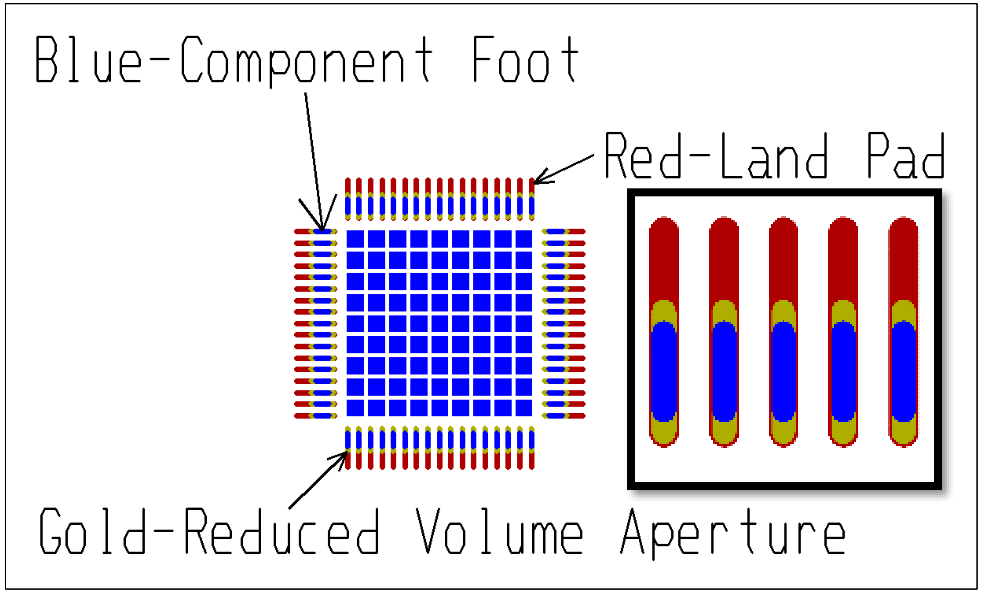

- Component data sheet is necessary.
- Identify foot dimension (Lp). When foot is much shorter than land pad, bridging can occur.
- Gull wing lead is greatly exposed to convection heat in reflow allowing it to heat quickly.
- If foot is 25% shorter or more than land pad, lead can heat faster than land pad.
- Paste can wet up lead to shoulder then has no where to wet.
- Pooling can occur at foot before paste wets out to end of lead causing bridging across leads.

# Bridging at SMT Reflow: What causes bridging after reflow when it is not present after print



When the PWB land pad is designed for a specific component and the component is replaced by a different component with a shorter foot, bridging at reflow can occur.

# Bridging at SMT Reflow: What causes bridging after reflow when it is not present after print

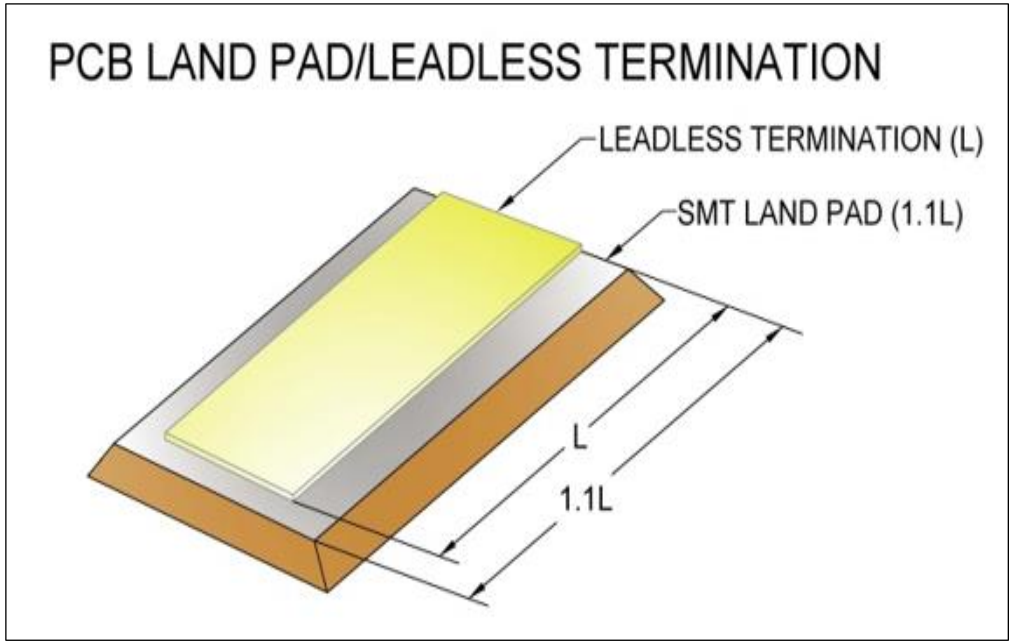



| Component Foot vs Land Pad | Volume Reduction of PWB Land |
|---|---|
| 25% Shorter | 10% |
| 50% Shorter | 25% |
| 70% Shorter | 40% |

New Aperture to be centered on Comp. Foot

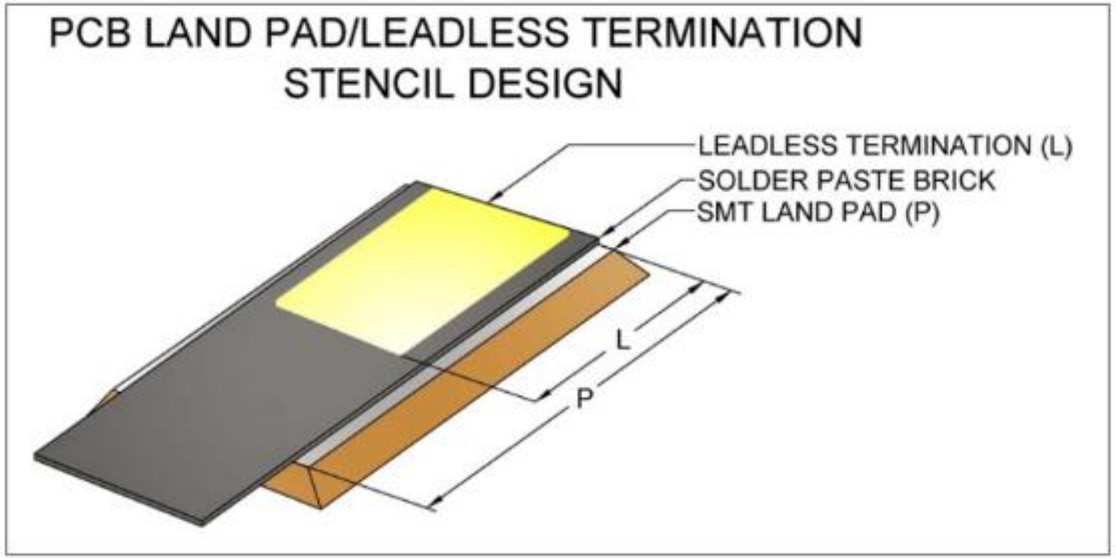
*When the new apertures are used, the reduced paste volume will flow up to the lead shoulder as it wets to the land pad eliminating bridging.*

# Insufficient Solder Volume at SMT Reflow:
# The correlation of stencil design to solder volume for leadless devices.




- To prevent this defect, compare the leadless package termination size to the PCB land pad.
- Package design allows for uniform wetting of solder across surfaces.
- To obtain sufficient solder paste volume after reflow and produce acceptable solder joints, the PCB land pad length is ~110% of the leadless termination length
- If PWB land pad length is more than 110% of termination length solder has more area to cover and solder volume must be increased.

# Insufficient Solder Volume at SMT Reflow:
# The correlation of stencil design to solder volume for leadless devices.

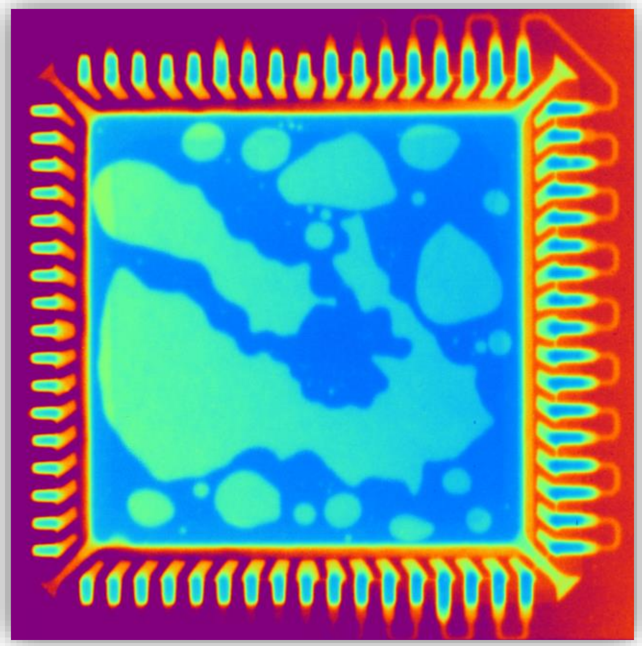



$$\text{Volume increase } (\%) = 50 * \left(1 - \left(\frac{L}{P}\right)\right)$$

- Aperture widths should follow "Half Pitch Rule".
- Additional solder paste volume should always be printed to the "toe" side for leadless components.
- Stencil apertures can be extended up to 40 mils beyond SMT land pad toe and still pull back during reflow. Typical is 5 – 10 mils beyond toe of land pad.
- Formula is based on 5 mil thickness. If foil thickness is reduced stencil aperture volume for the leadless components should be increased accordingly.

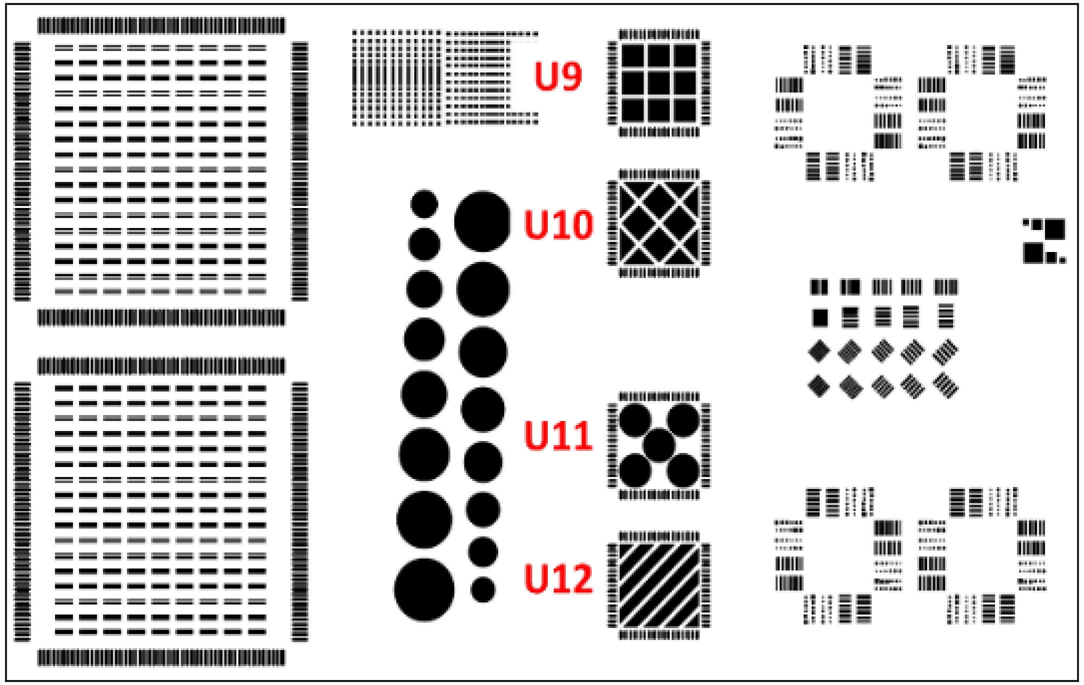
# Voiding: Design ideas to reduce voiding on ground pads

IPC 7525(B) Stencil Design Guidelines suggest 20% - 50% reduction in area, use of window pane design when possible and to avoid applying paste directly over vias.



- Window pane design most utilized historically.
- Five dot pattern along with other designs have been successfully used over past few years.
- Goal - to reduce voiding along with removing enough volume to eliminate skew or twist during reflow.

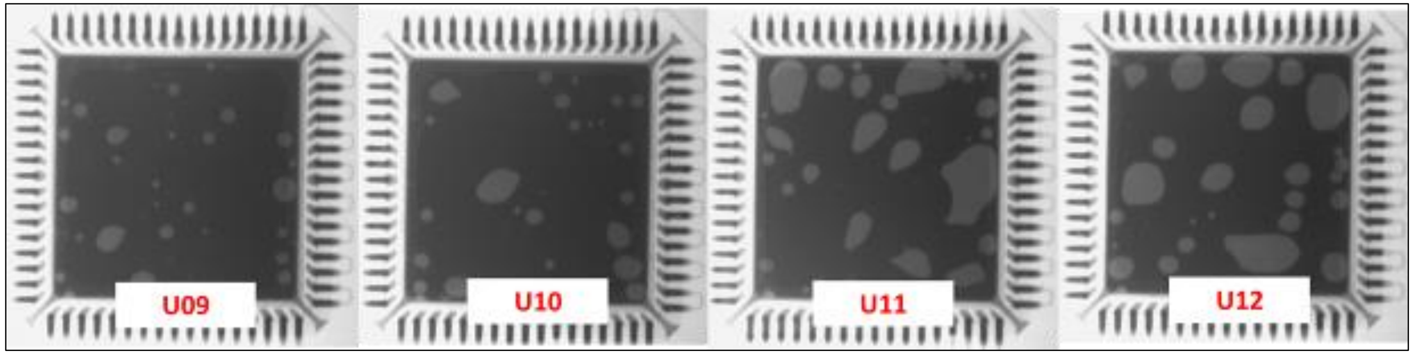
# Voiding: Design ideas to reduce voiding on ground pads




Stencil Design used to determine void percentage based on ground pad stencil design.

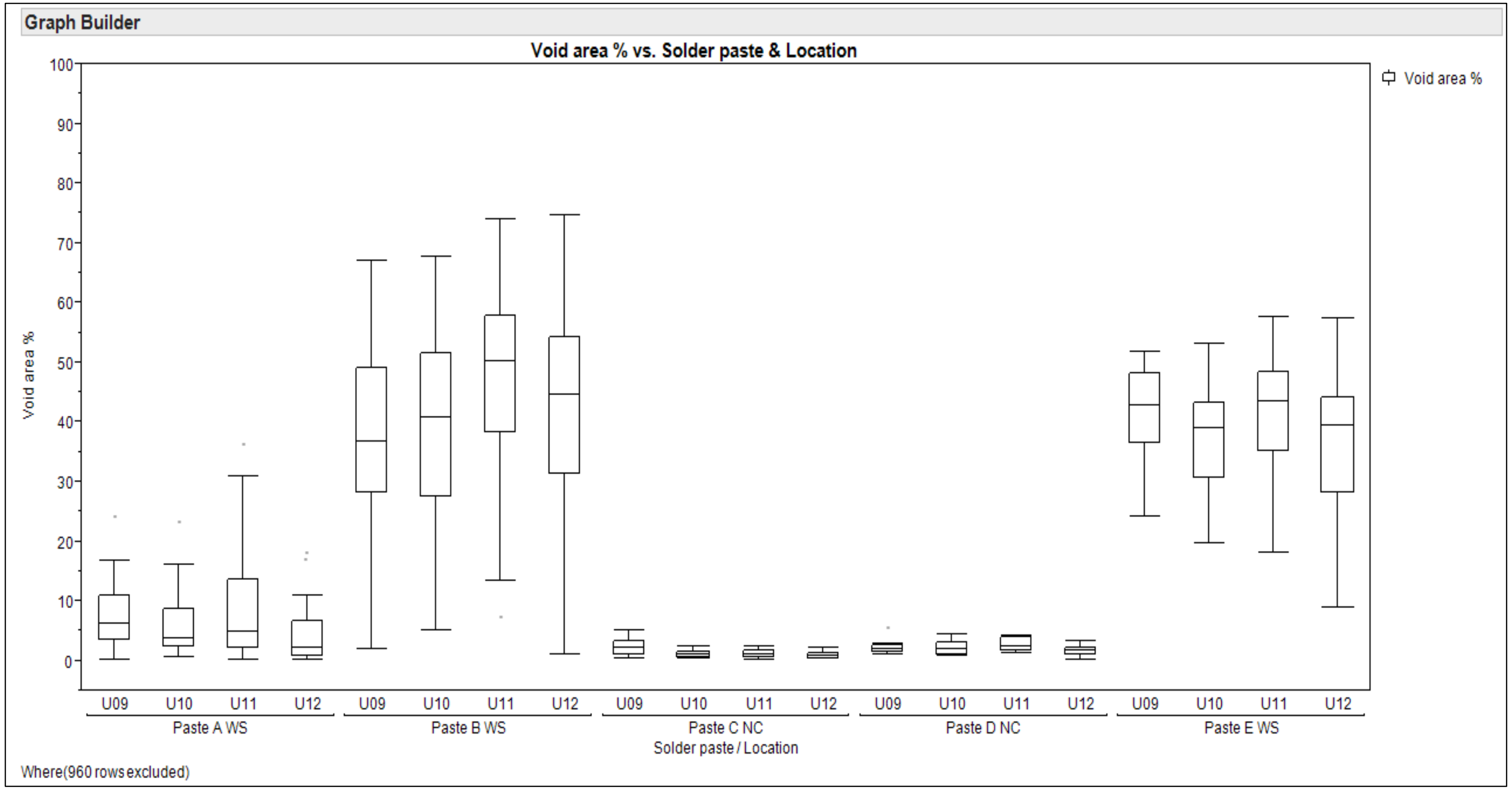
| Location | Aperture Shape | Aperture Size in mils (mm) | Spacing in mils (mm) | Paste Coverage Area (%) |
|---|---|---|---|---|
| U9 | Square | 88 (2.24) | 20 (0.51) | 65.3 |
| U10 | Diamond | 99 (2.51) | 20 (0.51) | 65.3 |
| U11 | Circle | 132 (3.35) | 8 (0.20) | 63.9 |
| U12 | Stripe | 40 (1.02) | 20 (0.51) | 65.0 |

# Voiding: Design ideas to reduce voiding on ground pads



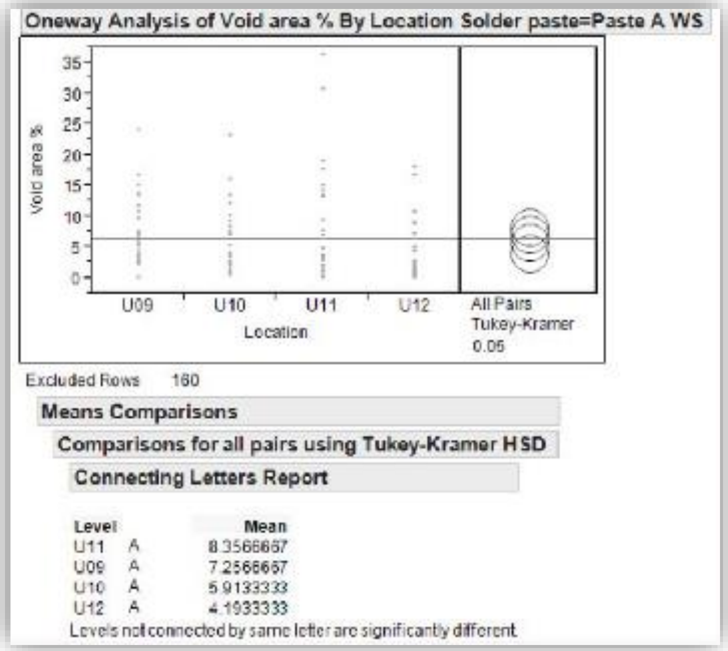

- 3 Different Water Soluble, SAC305 pastes printed
- 2 Different No Clean, SAC305 pastes printed
- Components placed and reflowed
- X-Ray Images taken and void percentage measured

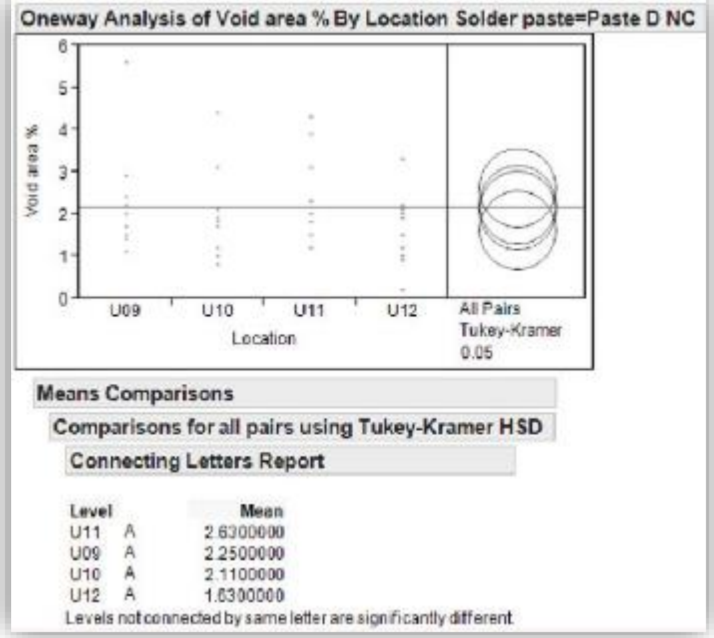
# Voiding: Design ideas to reduce voiding on ground pads

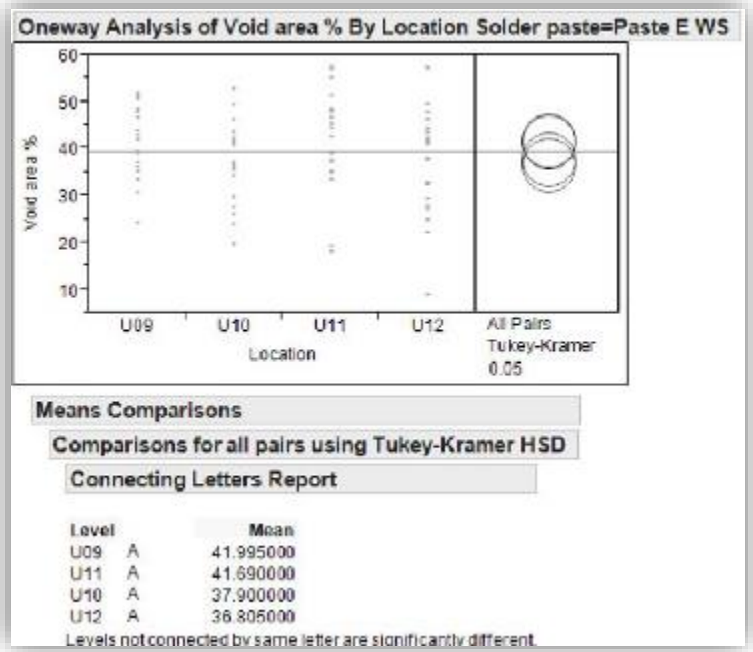



- Water soluble pastes (A, B, E) generate higher voiding than no cleans (C, D)
- WS paste A generated lowest voiding of the water solubles
- Stencil design had little impact on voiding levels

# Voiding: Design ideas to reduce voiding on ground pads
# Tukey-Kramer HSD used to validate results



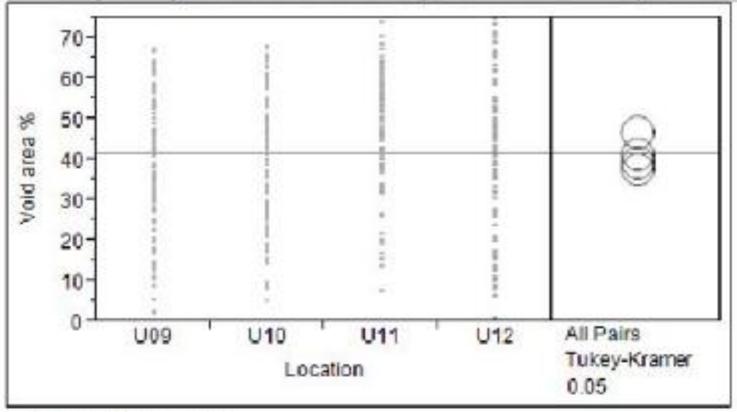




Voiding by stencil design (location) was statistically similar for each of these solder pastes.

**Stencil design made no difference in this test**

# Voiding: Design ideas to reduce voiding on ground pads
# Tukey-Kramer HSD used to validate results

Oneway Analysis of Void area % By Location Solder paste=Paste B WS




Excluded Rows 600

**Means Comparisons**

**Comparisons for all pairs using Tukey-Kramer HSD**

**Connecting Letters Report**

| Level | | | Mean |
|---|---|---|---|
| U11 | A | | 47.224545 |
| U12 | | B | 41.633636 |
| U10 | | B | 39.655455 |
| U09 | | B | 38.027273 |

Levels not connected by same letter are significantly different.

- When looking at Water Soluble Paste B, voiding based on stencil design was statistically higher for the 5 dot pattern (U11).
- The other designs were not statistically different.

# Conclusions

- Identification of "Universal" defects in print process is critical to improve first pass yields.
- To prevent insufficient paste at print, stencil foil thickness must be chosen based on identified area ratios. The type of paste can also affect the minimum area ratios to be printed.
- Solder ball issues can be addressed using a "U-Shape" or "Inverted Homeplate" with proper design of apertures over actual component land pads.
- Tombstoning can be reduced or eliminated with a "Reverse U-Shape" or "Reverse Inverted Homeplate" design. Proper design of apertures over actual component land pads is critical.

# Conclusions

- The "Half Pitch Rule" is a valid method to prevent bridging at print.
- When bridging after reflow is present, the size of the PWB land pad must be compared to the actual component lead and the stencil aperture size must be adjusted to remove volume.
- Reducing voiding on BTC ground pads is an ongoing challenge. Different stencil aperture designs were shown not to be statistically different in voiding results with the exception of one design on a specific paste used.

# Future Work

Further investigation into ground pad designs for BTC's will be conducted to identify stencil patterns that can minimize voiding after reflow.

# THANK YOU!

Greg Smith
FCT Assembly, Inc.
gsmith@fctassembly.com