Thin Foil Printing in Today's Miniaturized World: Do Printing Rules Change?

Chrys Shea
Shea Engineering Services
East Greenwich, RI, USA
chrys@sheaengineering.com

Greg Smith
BlueRing Stencils
Lumberton, NJ, USA
gsmith@blueringstencils.com

Ray Welch
Koh Young America
Duluth, Georgia, USA
ray.welch@kohyoung.com

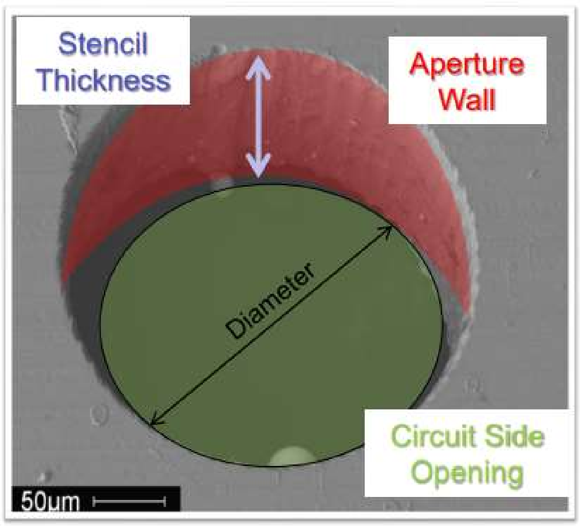
Presented at SMTA International 2021

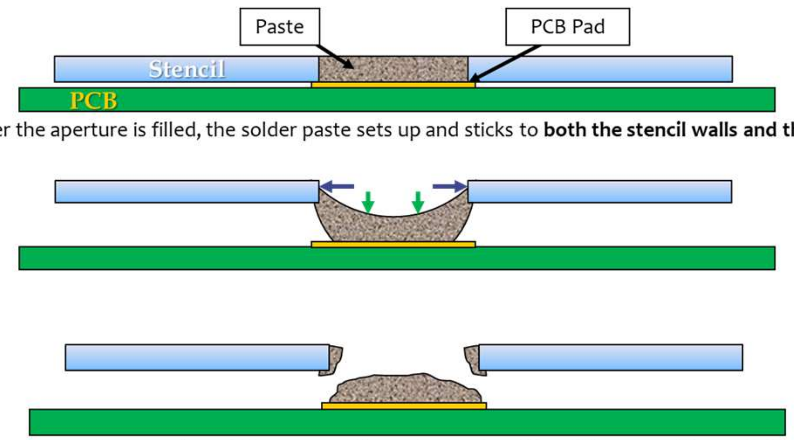
Agenda

- Introduction
- Experimental Design
- Results of Experiments
- Discussion
- Conclusions
- Q & A

Introduction: Area Ratio and Paste Transfer

$$AR = \frac{\textit{Area of Circuit Side Opening}}{\textit{Area of Aperture Walls}}$$





After the aperture is filled, the solder paste sets up and sticks to **both the stencil walls and the pads.**

Depending on area ratio, **a portion of the paste will transfer to the PCB, while some will remain in the aperture.** Some paste may also stick to the bottom of the stencil due to stringing, bad gasketing or "pump out."

The smaller the AR, the lower the TE

Introduction: Area Ratio

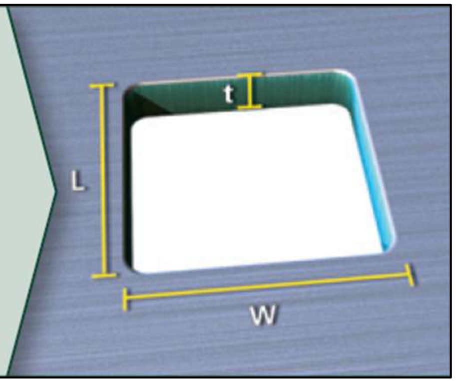
Rectangles

$$\text{Area Ratio} = \frac{\text{Area Opening}}{\text{Area Walls}}$$

$$\text{Area Opening} = L \times W$$

$$\text{Area Walls} = 2t(L+W)$$

$$\text{Area Ratio} = \frac{L \times W}{2t(L+W)}$$



Squares, L=W

$$\text{Area Ratio} = \frac{W^{\cancel{2}}}{2t(\cancel{2}W)}$$

$$\text{Area Ratio} = \frac{W}{4t}$$

Squares and circles have same ARs

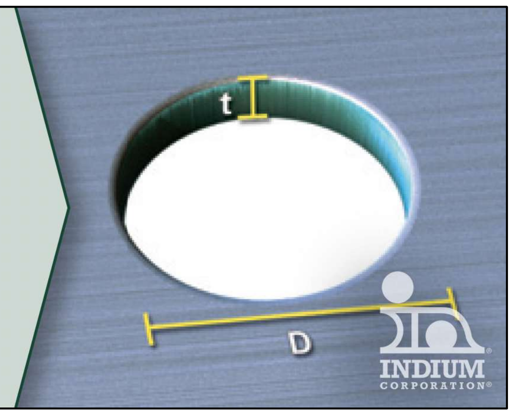
Circles

$$\text{Area Ratio} = \frac{\text{Area Opening}}{\text{Area Walls}}$$

$$\text{Area Opening} = \frac{\pi D^2}{4}$$

$$\text{Area Walls} = \pi D t$$

$$\text{Area Ratio} = \frac{D}{4t}$$

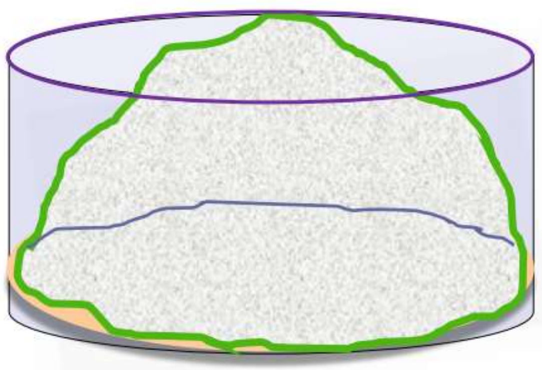


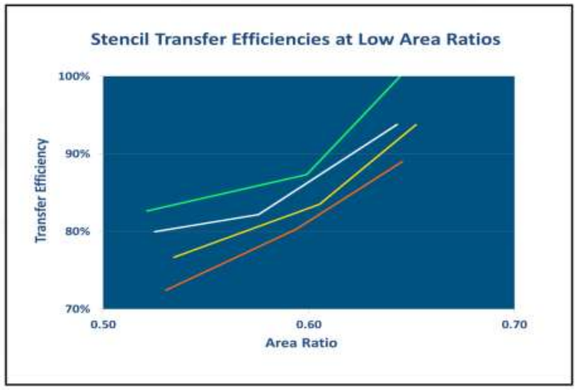
Images courtesy of Indium Corporation

Introduction: Transfer Efficiency

Relates how much paste we want to print versus how much we actually printed.

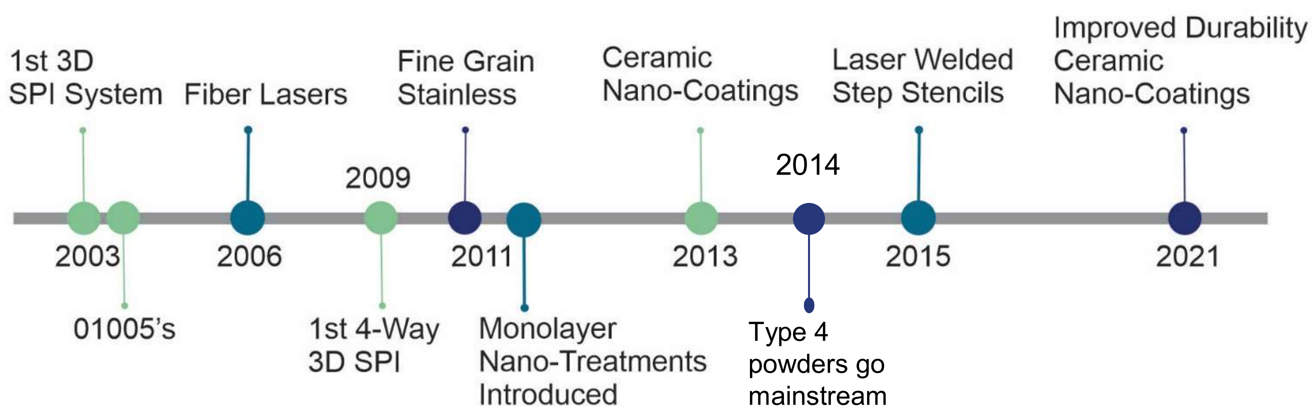
$$\%\,TE = \frac{\textit{Volume of Paste Deposited}}{\textit{Volume of Stencil Aperture}} \text{ X } 100$$





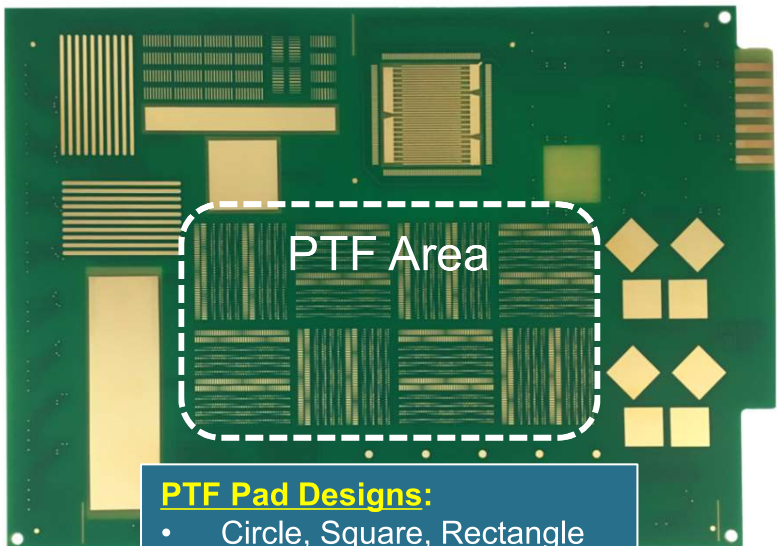
- Minimum of 80% TE when possible
- *Minimizing Variation* is more important than maximizing transfer!

Introduction: Printing Technology Timeline



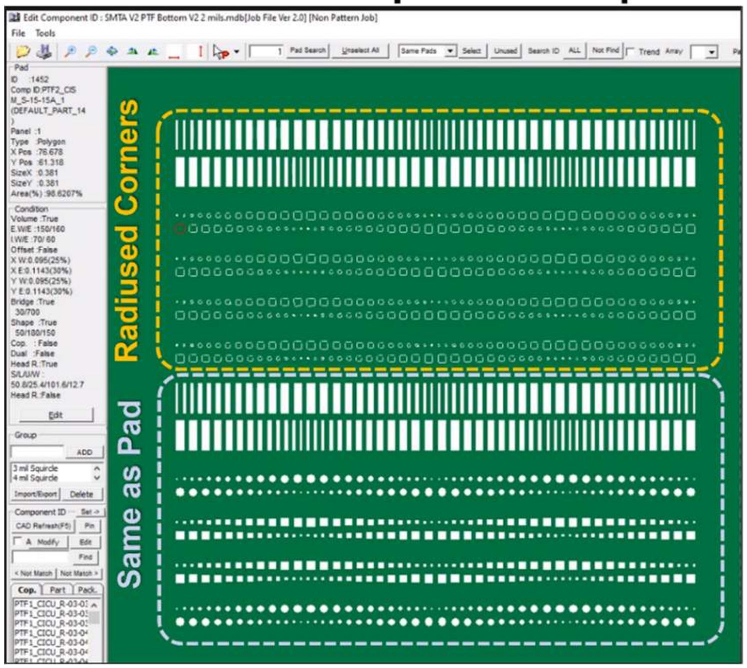
Experiment: Test Vehicle

SMTA Board, unpopulated side



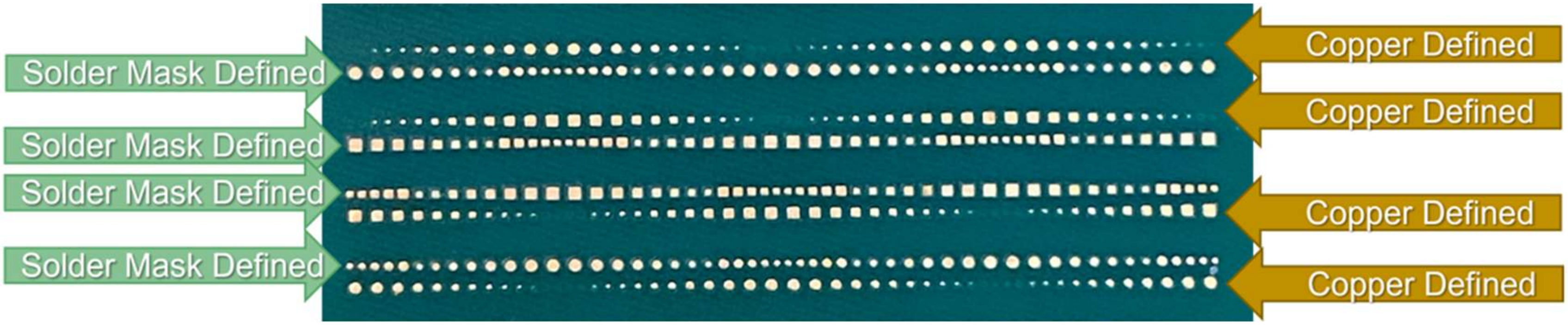
PTF Pad Designs:
- Circle, Square, Rectangle
- SMD and NSMD
- 3 - 15 mils
- 32 of each combo per print

Print-To-Fail Aperture Shapes



Experiment: PTF Pattern Layout

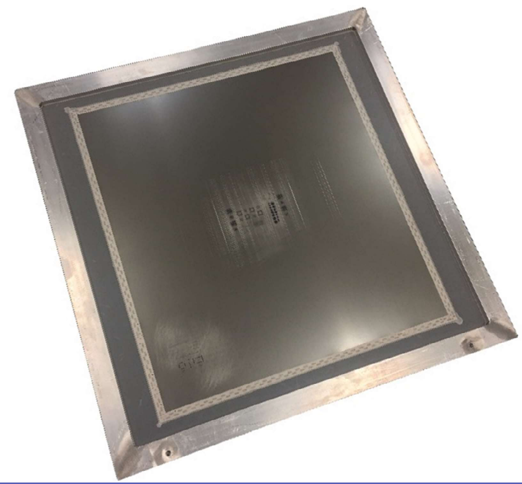
Print To Fail (PTF) Design



Alternates pad definition, shape, and print order

Experiment: Materials

Solder Paste:
Indium 8.9 HF Type 5-MC
15-25µm particle diameter
SAC305 alloy



Stencils		
Thickness	Coating	Uncoated
2 mil (50 µm)	X	X
3 mil (76 µm)	X	X

SMTA International

Experiment: Equipment

<u>MPM Momentum BTB</u>
Support Tooling: Quik-tool (3)
Speed: 30 mm/sec
Pressure: 7.0 kg
Separation Speed: 5.0 mm/sec
Separation Distance: 3.0 mm

<u>Set Up</u>
Knead: 4-6 strokes
Setup prints: 2-7 for paste alignment

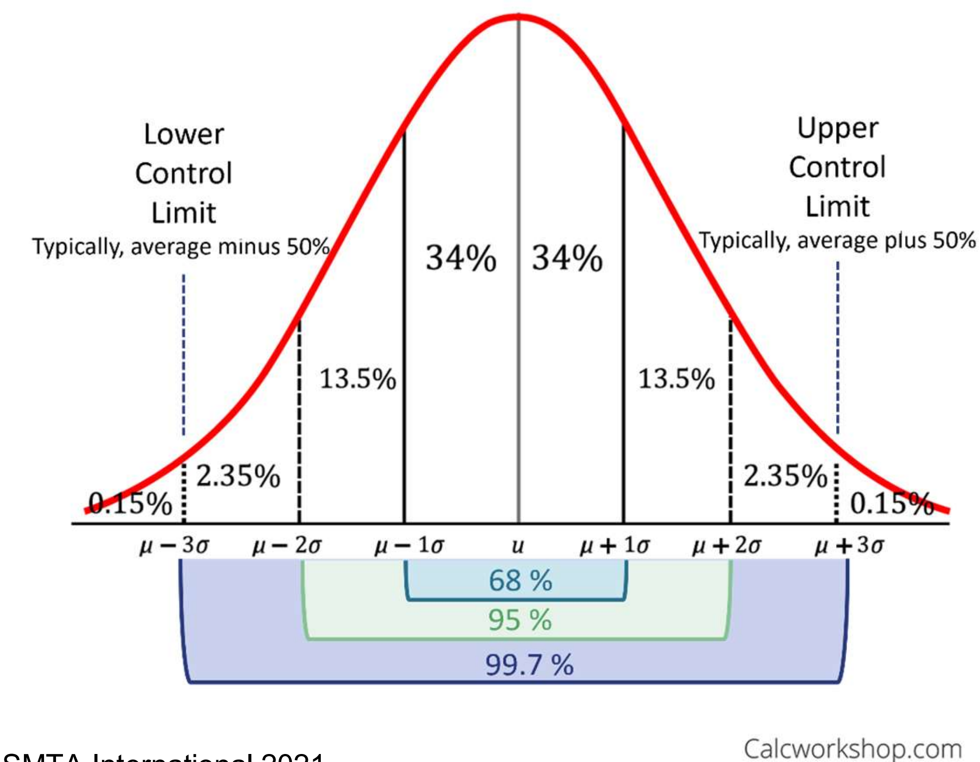
<u>Under-Stencil Wipe</u>
Speed: 30 mm/sec
Sequence: wet/vac/vac or wet/vac/vac/dry
Frequency: 1 (each print)

<u>Koh Young aSPIer3</u>
Resolution: 10 um
Height Threshold: 20 µm

Data Review: Metrics

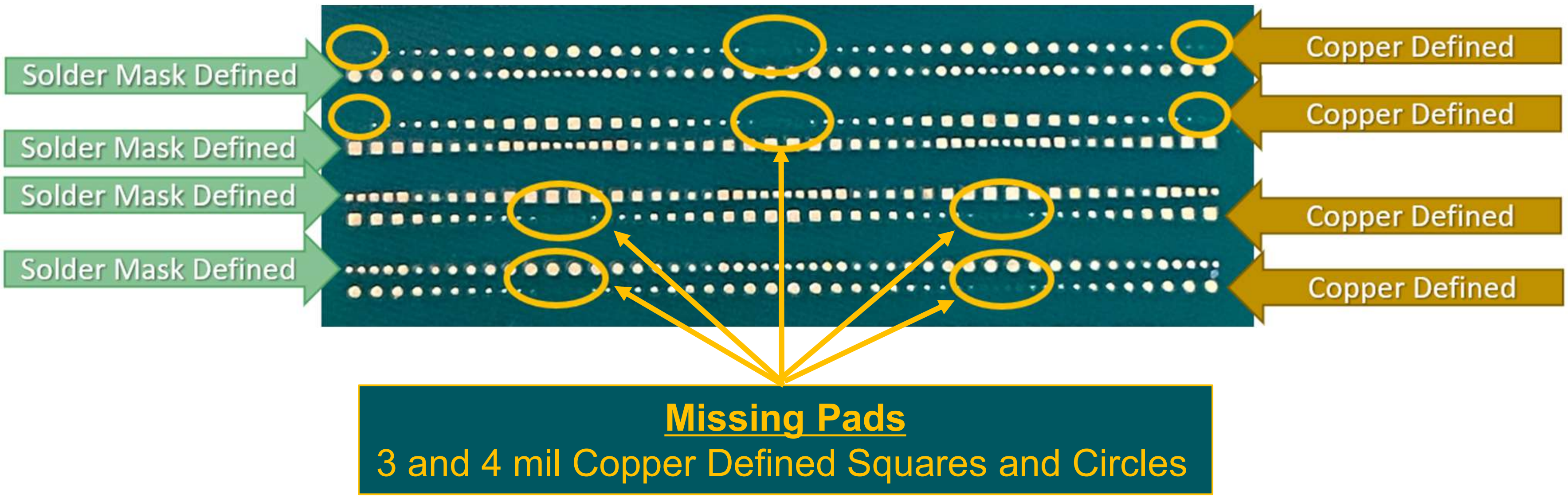
Coefficient of Variation - CV

- Calculate average and standard deviation
- Divide the standard deviation by the average
  - Look at std dev as a % of the average:
    - **<10%: desired**
    - **10 - 15%: acceptable**
    - **Over 15%: unacceptable**



Data Review: Excess Variation and Missing Pads

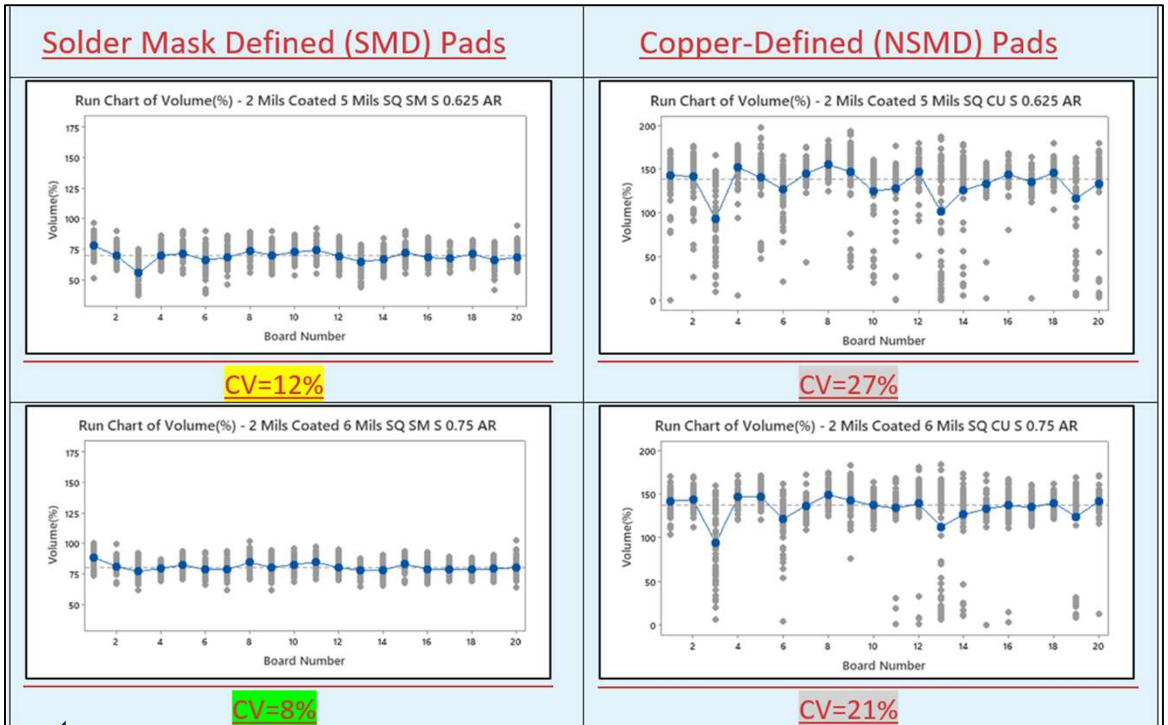
Print To Fail (PTF) Design



** Fabricator given waiver for features 5 mil or less to mitigate cost*

Results: SPC Run Charts

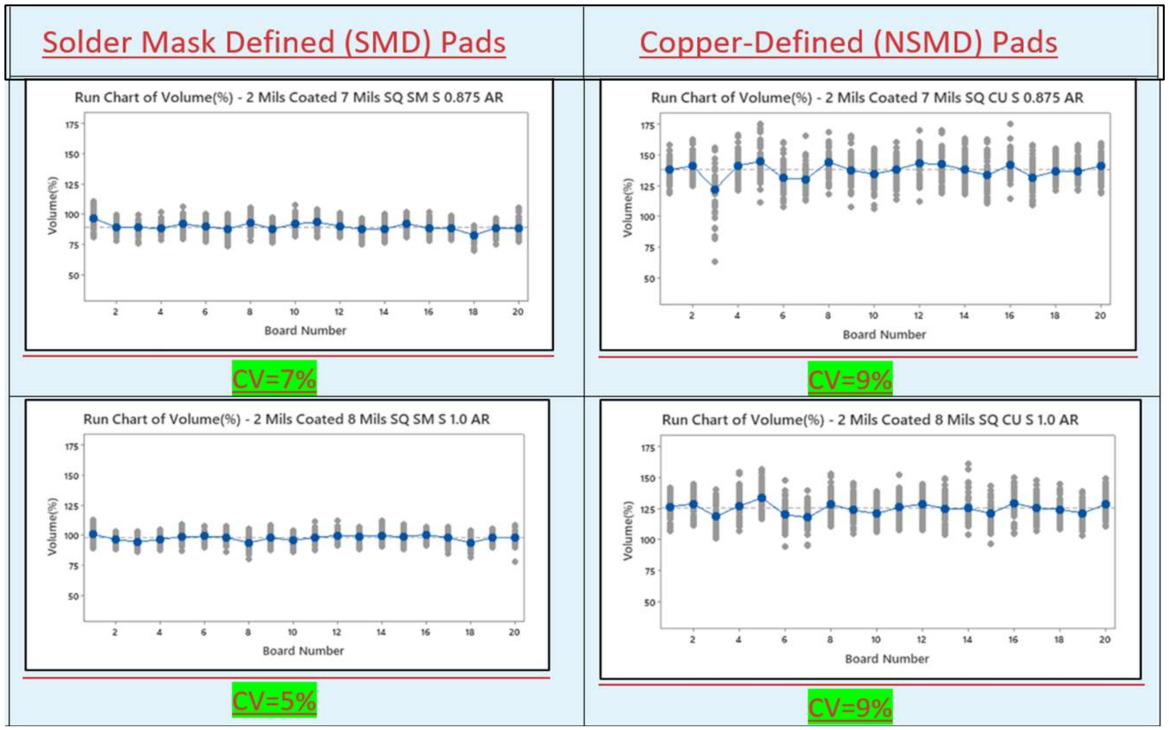
INCREASED FEATURE SIZE=*better process control*



SOLDER MASK DEFINED PADS = *better process control*

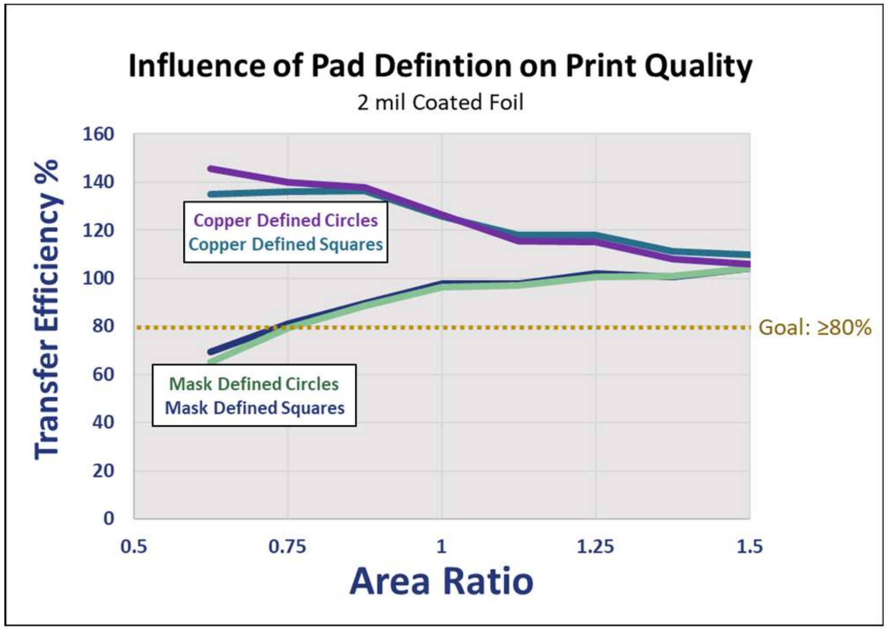
Results: SPC Run Charts

INCREASED FEATURE SIZE=better process control



SOLDER MASK DEFINED PADS = *better process control*

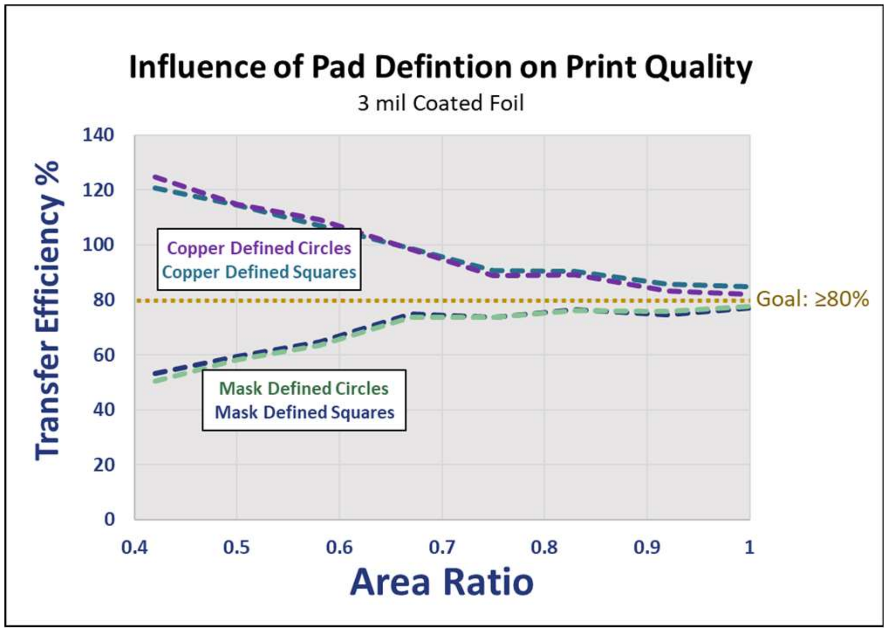
Results: TE vs AR on 2 mil foil



Notice the divergence between copper defined and mask defined pads

- Mask defined show acceptable TEs
- Copper defined show excessive solder, presumably from less robust gasketing

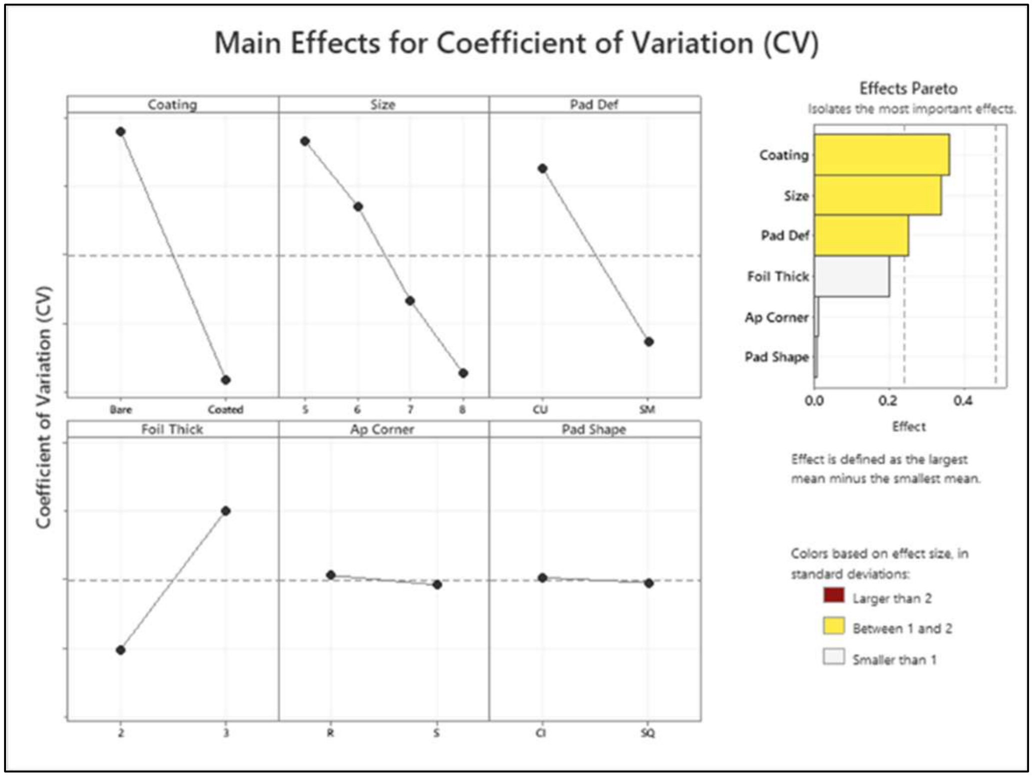
Results: TE vs AR on 3 mil foil



Similar divergence between mask and metal defined pads

Mask defined pads offer far better process control

Results: Main Effects Plots



Factors in Variability

#1: Stencil Coating
#2: Pad Size
#3: Pad Definition
#4: Foil Thickness

Non-Factors

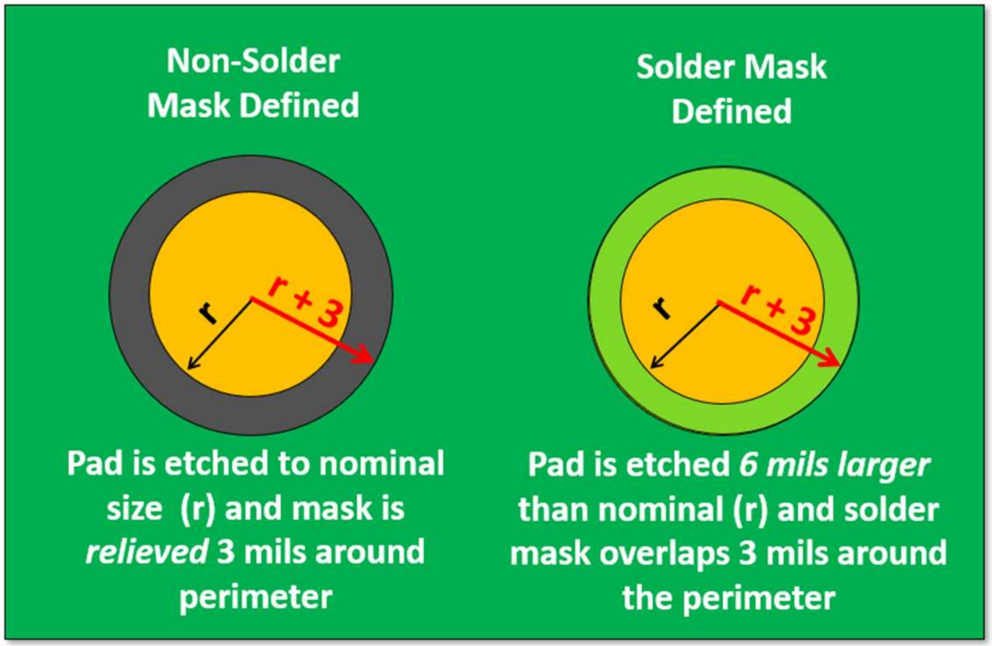
Aperture Shape
Pad Shape

Discussion: Stencil Coating and Pad Size

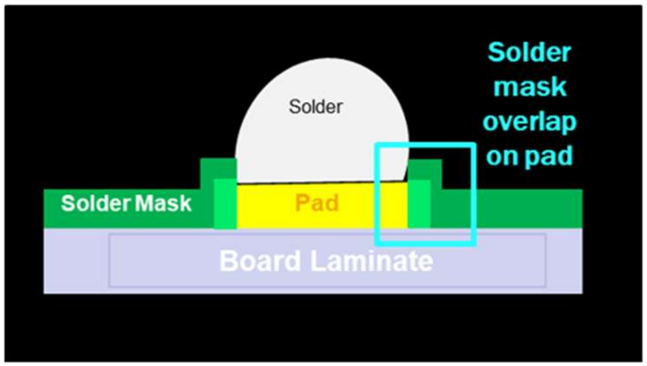
- No surprises here…
  - Benefits of stencil coating have been documented for over 10 years
  - Challenges of smaller pad sizes have been documented for over 30 years

Discussion: The Differences in Pad Definition

PCB Pad Definition

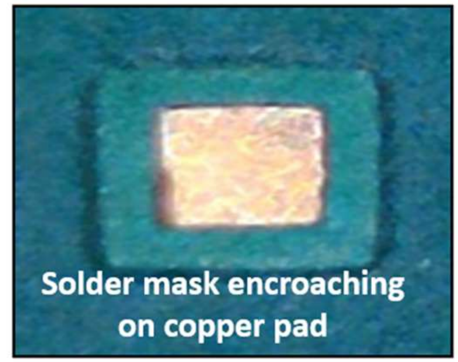


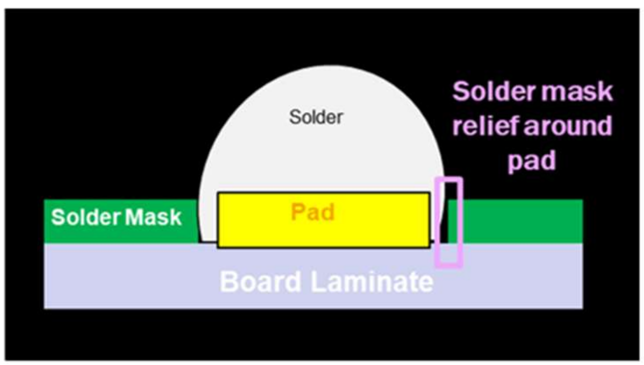
Discussion: Features of PCB Pad Definition



Solder Mask Defined (SMD)

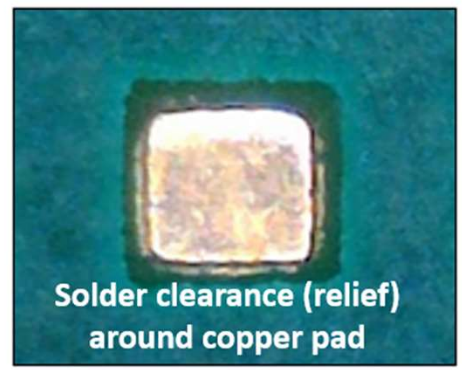
- Pad is larger than nominal size: *easier to etch (DFM)*
- Solder mask overlapping on all sides creates gasketing surface: *easier to print (DFA)*



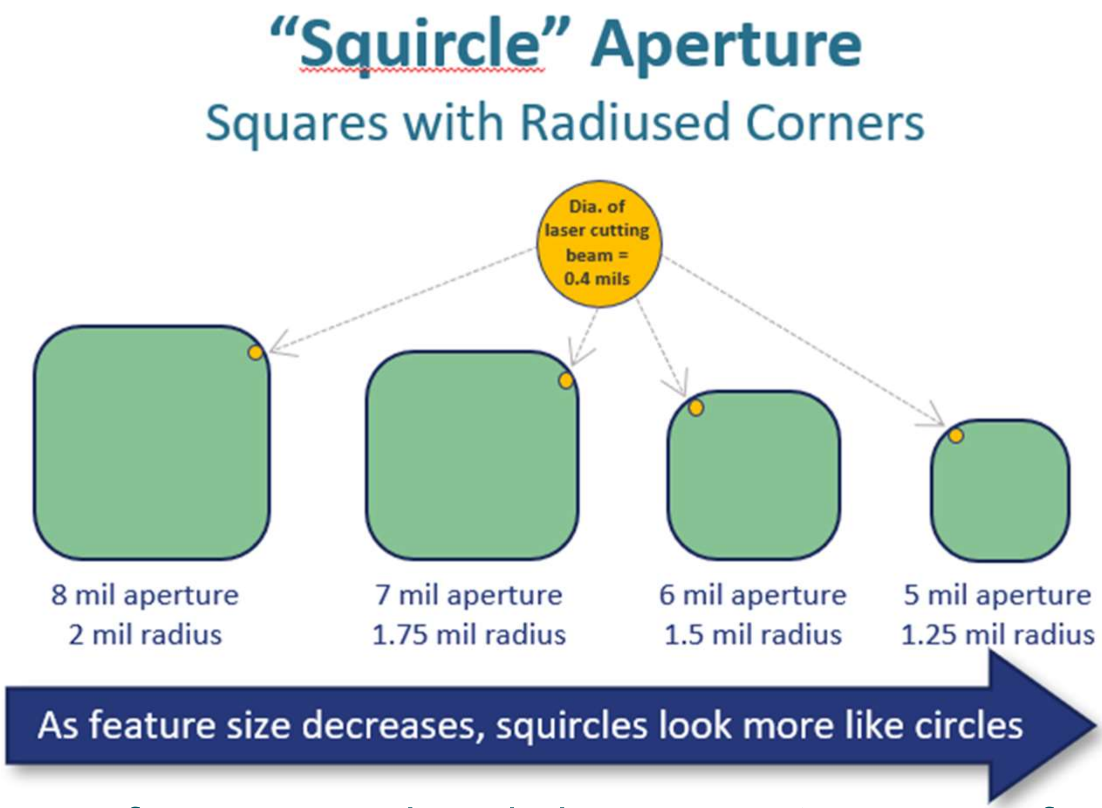


Non-Solder Mask Defined (NSMD or Copper Defined)

- Pad is nominal size: challenging to etch below 8mil
- Undersized pads present gasketing issues: harder to print
- "Wrap around" of solder on sides of pad improves joint strength



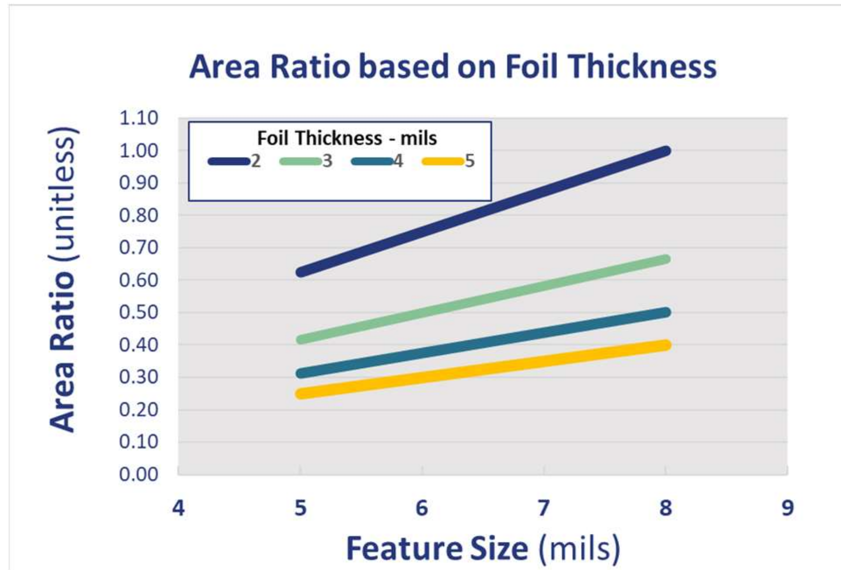
Discussion: Rounding Sharp Corners



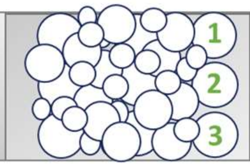
The advantage of squares and squircles appear to wane as features shrink

Discussion: The effect of foil thickness on AR

- ARs increase dramatically as foils become thinner
- AR Formulas:
  - W/4t (square)
  - D/4t (circle)
- Differences in foil thickness have 4X the effect of aperture sizes on AR



Discussion: Three Ball Rule



To ensure good aperture fill, the thinnest foil that can be successfully printed is 3X the diameter of:

The **largest** T5 solder particle
25 µm

25
x 3
75

75 µm =

3 mil

OR

The **average size** T5 solder particle
20 µm

20
x 3
60

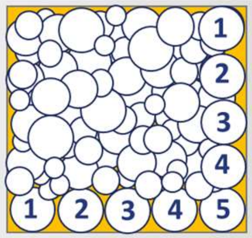
60 µm =

2.4 mil

Solder Mask Defined pads printed well with coated stencils all the way down to 2 mils!

The three-ball rule does not seem to apply

Discussion: Five Ball Rule



To ensure good aperture release, the narrowest aperture that can be successfully printed is 5X the diameter of:

The **largest** T5 solder particle

25
x 5
125

125 µm =

5 mil

OR

The **average size** T5 solder particle

20
x 5
100

100 µm =

4 mil

Mask Defined pads printed well with coated stencils all the way down to 5 mil features

If applied, the 5-ball rule would pertain only to mask defined pads and coated stencils

Conclusions: Reducing Variation

In leading edge electronic miniaturization, reducing variation is more important than high average paste transfer rates

Variation can be controlled by:

- **Product Design:** Mask Defined pads and their sizes had the largest influences on repeatability. PCB Features smaller than 8 mils should be mask defined.
- **Product Manufacturing:** Nanocoating the stencils was the most dominant main effect in reducing variation. Uncoated stencils could not produce CV values <10% for apertures 8 mils or smaller and were more difficult to auto-clean in the printer. They also were far more sensitive to changes in foil thickness.

Conclusions: The Rules

Area Ratio of 0.6 or higher:

- For the case of solder mask defined pads and coated stencils, an Area Ratio of 0.6 or higher should be applied. Both saw just under 80% Transfer Efficiency but met Coefficient of Variation targets of less than 10%.

Three Ball Rule:

- For the case of solder mask defined pads and coated stencils, the Three Ball Rule predicts a minimum stencil thickness of 2.4 mils. The actual minimum was 2 mils, exceeding the expectation, even based on the average particle diameter.

Five Ball Rule:

- For the case of solder mask defined pads and coated stencils, the finest feature to print repeatably was 5 mils, agreeing with the theory that the smallest aperture should be at least as wide as 5 of the *largest* particles in the solder paste.

Notice the rules all apply in the context of SMD pads and coated stencils.
The data shows poor process capability without these key features.

Thank You……..Questions?

Chrys Shea
Shea Engineering Services
East Greenwich, RI, USA
chrys@sheaengineering.com

Greg Smith
BlueRing Stencils
Lumberton, NJ, USA
gsmith@blueringstencils.com

Ray Welch
Koh Young America
Duluth, Georgia, USA
ray.welch@kohyoung.com