IPC Midwest 2011

Use of the IPC Solder Spread Coupon to Evaluate Pb-Free Solder Pastes and PCB Surface Finishes

Brian Madsen, Ph.D.

Continental AG

Executive Summary:

Continental is using the IPC solder spread coupon (adopted from NPL) to evaluate Pb-free solder pastes and PCB surface finishes for Solderability. This presentation will compare and contrast solder spread results for multiple PCB finishes using multiple Pb-free solder pastes. The spread data is collected for as-received PCBs and after one or more reflow processes to observe the degradation in spread for the different surface finishes. Very different behavior is observed when comparing common Pb-free compatible PCB surface finishes such as ENIG, OSP and immersion tin. Efforts to define specific pass/fail criteria for the solder spread coupon, based on comparisons to other common criteria, will be included. Additional information on the impact of variations in solder paste print volume to the resulting spread performance may also be presented if time/space allows. An overview of the results of the IPC 4-14 ENEPIG solder spread results may also be included if agreed to by the 4-14 committee.

Use of the IPC Solder Spread Test to Evaluate PCB Solderability

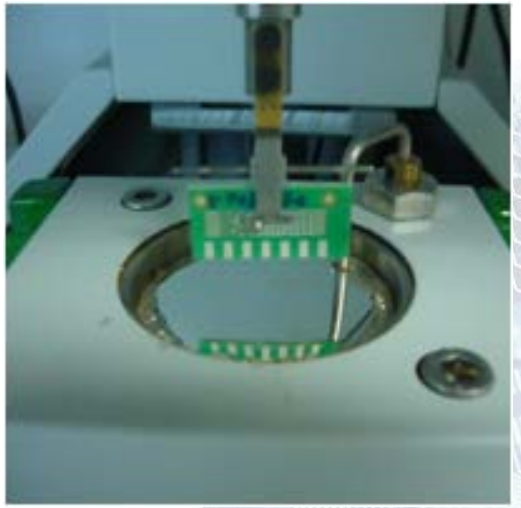
Brian Madsen

Continental AG

IPC Midwest 2011

Evaluating PCB Solderability

- IPC J-STD-003B
  - Test methods do not always reflect solderability performance observed during manufacturing
  - Ex: solderability issue in assembly; PCB supplier reports passing solderability by solder float test
    - Different flux, application method, solder material

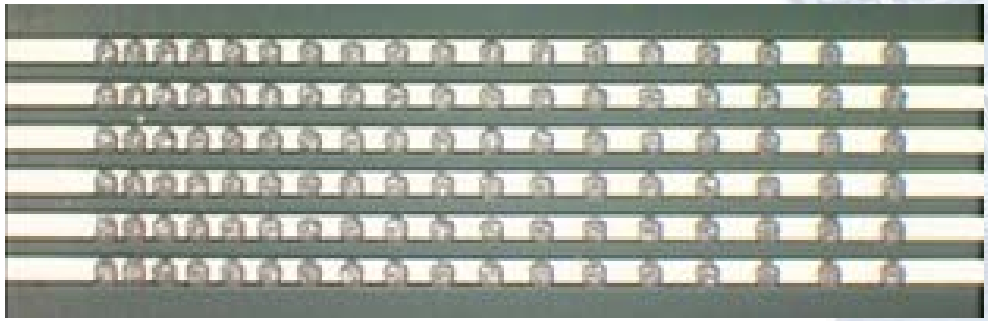


J-STD-003B Test Methods	Flux Application	Solder Method	Pads or Holes	Accept/Reject Criteria	Represent SMD Process
A/A1 – Edge Dip Test	Immersion	Immersion in solder pot	Pads	95% pad coverage	No
B/B1 – Rotary Dip Test	Immersion	Contact with solder pot	Both	95% pad coverage; fully wetted PTH walls (class 3)	No
C/C1 – Solder Float Test	Immersion	Contact with solder pot	Both		No
D/D1 – Wave Solder Test	Production process	Contact with solder wave	Both		No
E/E1 – Surface Mount Simulation Test	N/A	Solder paste print and reflow	Pads	95% pad coverage	Yes
F/F1 – Wetting Balance Test	Immersion	Contact with solder pot	Both	Wetting time and force requirement	No

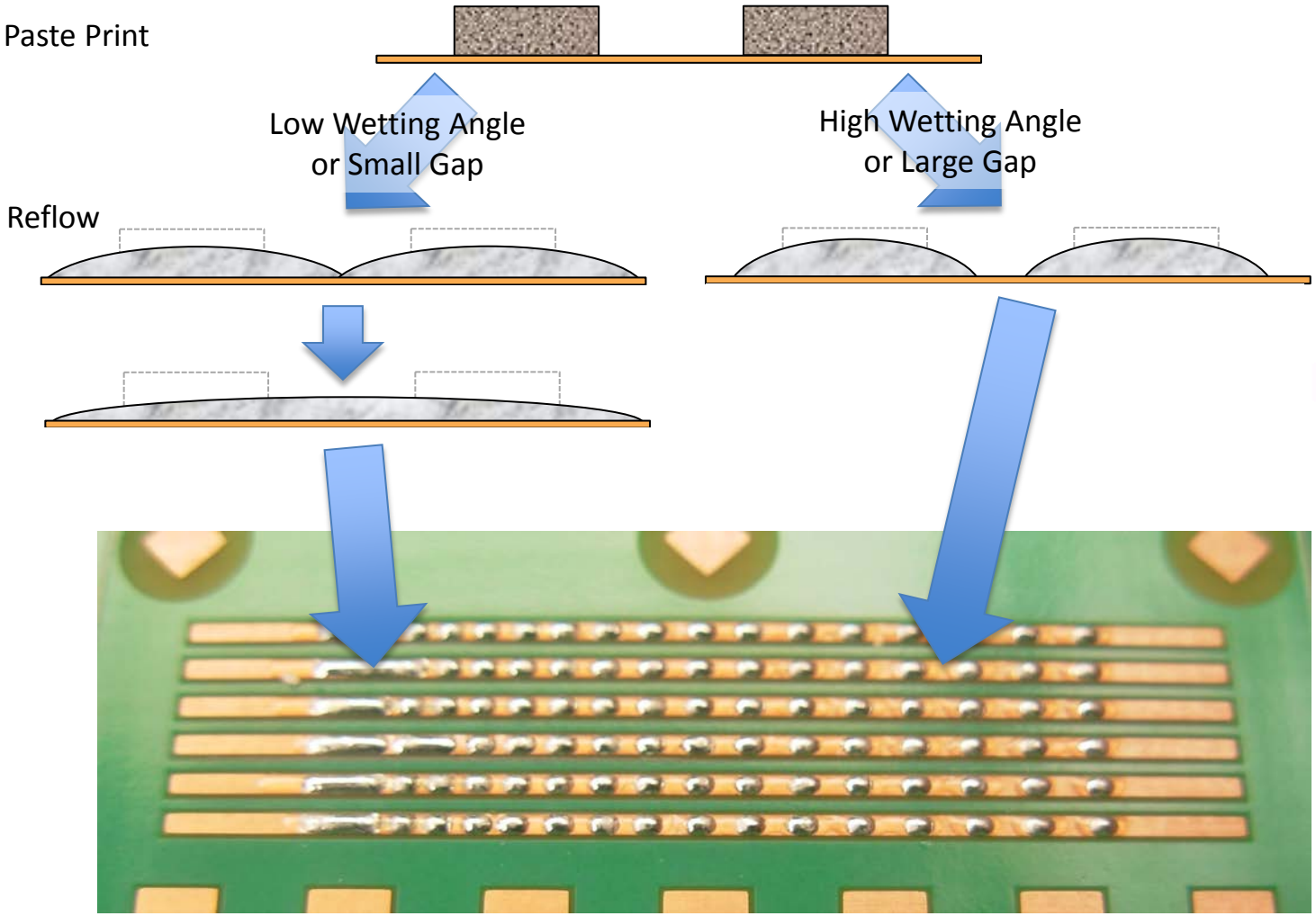
Evaluating PCB Solderability: IPC Solder Spread Test Coupon Details

- Method E/E1 - Surface Mount Simulation Test
  - 6 parallel copper traces, 0.5 mm wide (20 mils)
  - 18 paste deposits per line, with increasing gaps
  - 0.5 x 0.5 mm paste deposits
  - Continental pattern: metal-defined with mask between traces

Deposit	Gap mm [mils]	Deposit	Gap mm [mils]	Deposit	Gap mm [mils]
1-2	0.158 [6]	7-8	0.466 [18]	13-14	0.770 [30]
2-3	0.211 [8]	8-9	0.516 [20]	14-15	0.820 [32]
3-4	0.265 [10]	9-10	0.566 [22]	15-16	0.872 [34]
4-5	0.312 [12]	10-11	0.618 [24]	16-17	0.922 [36]
5-6	0.364 [14]	11-12	0.668 [26]	17-18	0.974 [38]
6-7	0.414 [16]	12-13	0.720 [28]		

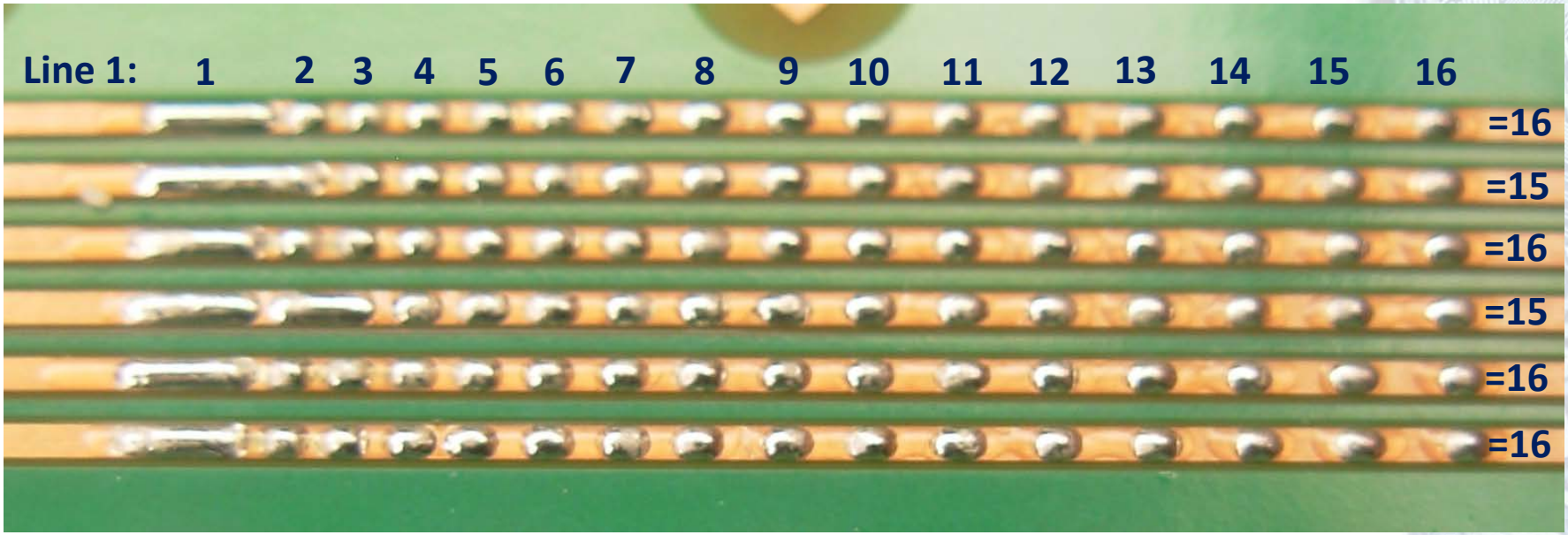


Solder Spread Test Concept



Analysis of IPC Solder Spread Test

- Count number of remaining solder deposits



- Input total (16+15+16+15+16+16) into equation to calculate solder spread as a %
- Sample = 14%

$$\text{solderability}, \phi = \frac{108 - \sum_{i=1}^{6} x_i}{102} \cdot 100\%$$

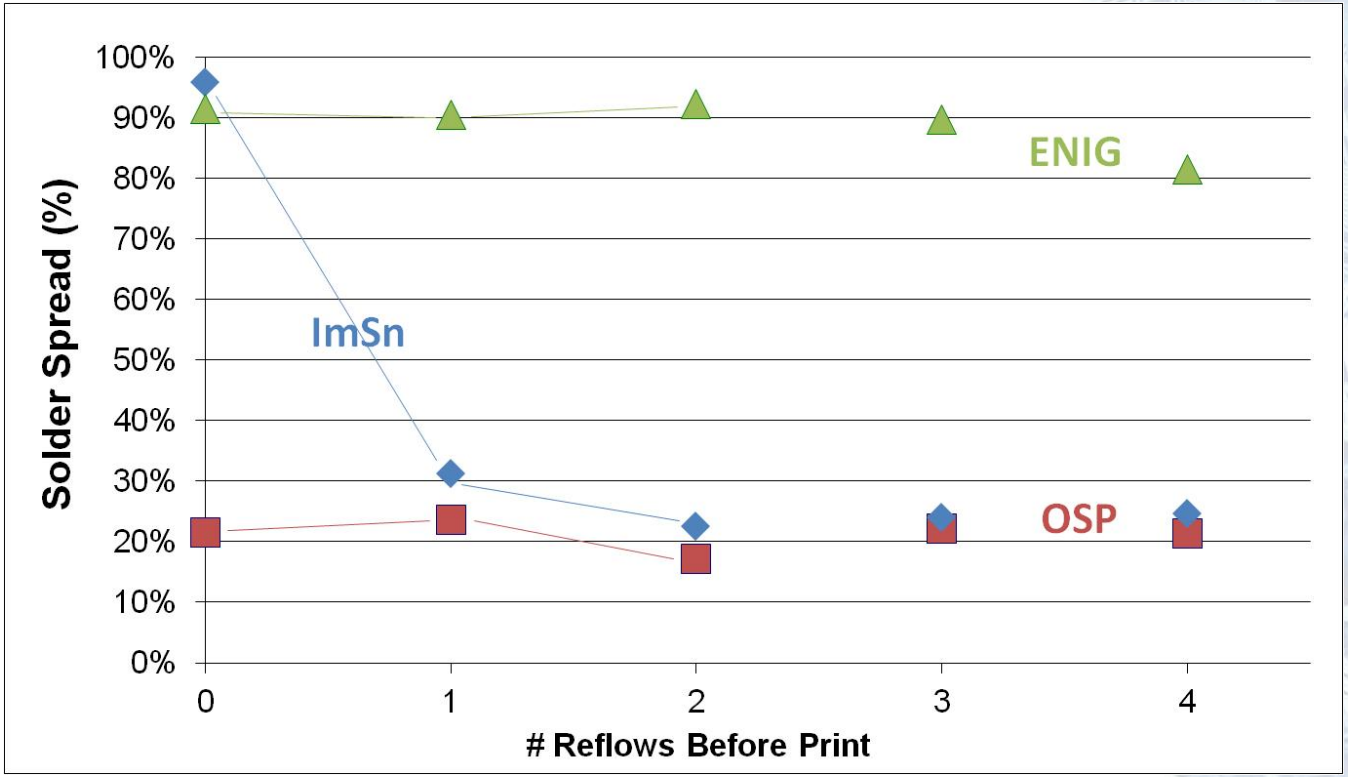
How are we using the spread pattern?

Comparison of Different Surface Finishes

- ROL0 Pb-free solder paste
- ImSn and ENIG spread is very good
- Spread on OSP is low
- Results match general mfg experience

Impact of Pb-free Reflow on Solder Spread

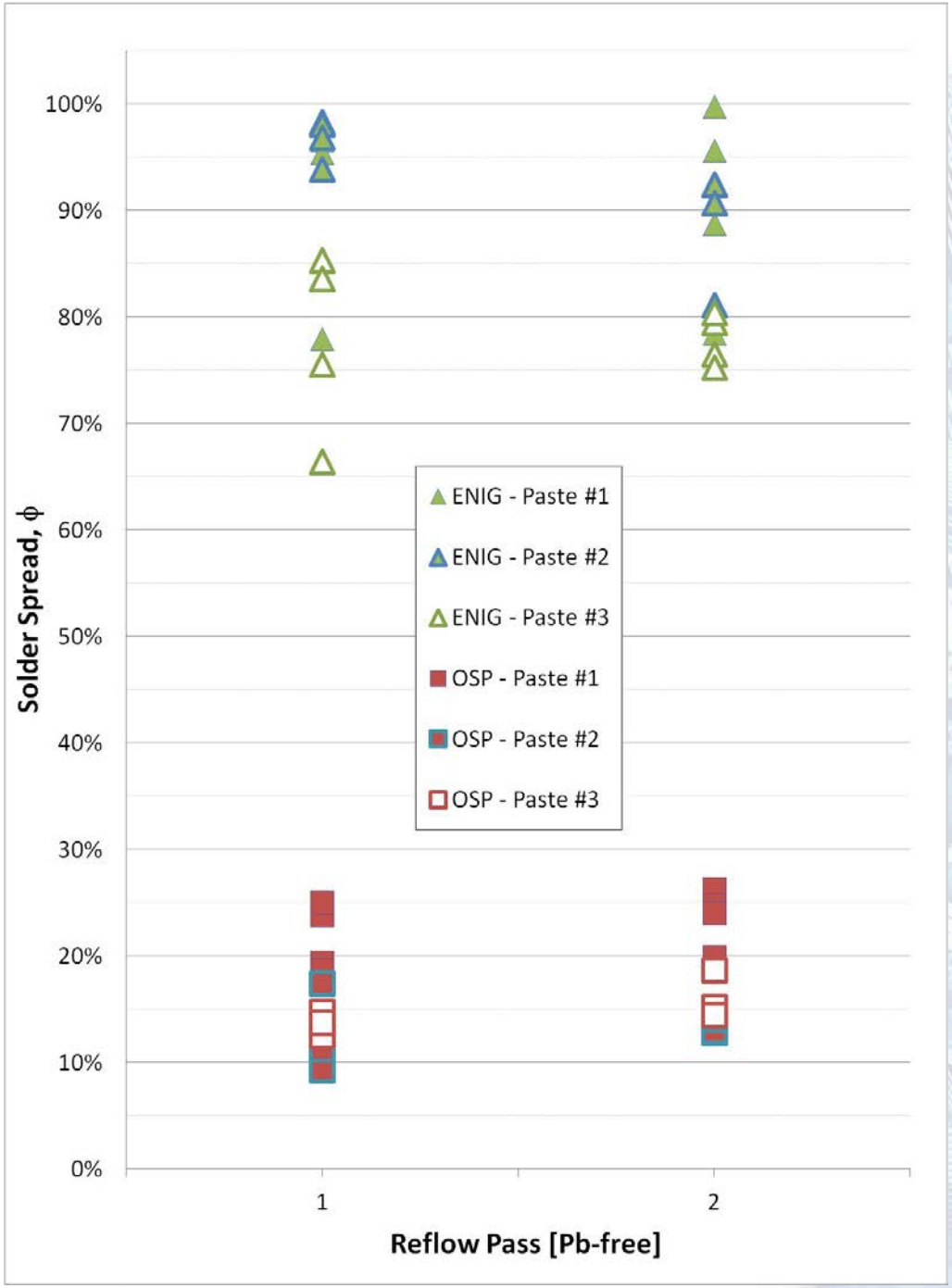
- Both ENIG and OSP show stability of solder spread for multiple reflows
- Spread on ImSn is very sensitive to reflow
- Typical results



- What is more important – high spread or predictability?

Comparison of Pb-free Solder Pastes

- Evaluated first and second pass reflow performance with 3 ROL0 Pb-free solder pastes
  - Some difference between solder pastes
  - In general, the surface finish is key with the solder paste playing a minor impact (few %)
- Expected for similar activity levels

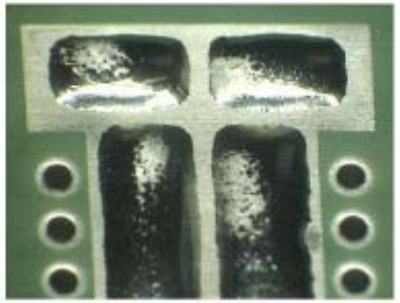


Can we set a pass/fail criteria?

Who cares about spread?

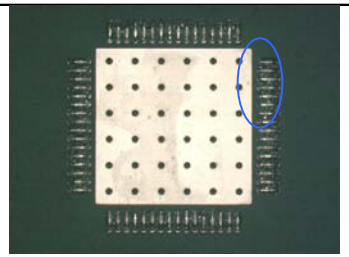
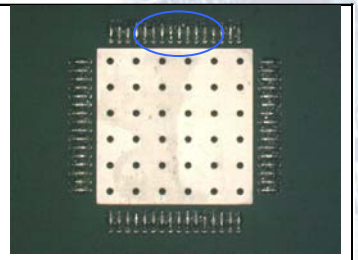
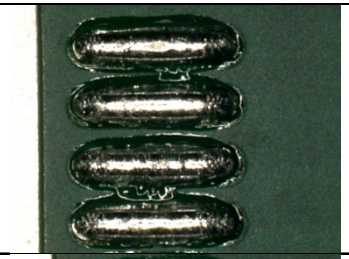
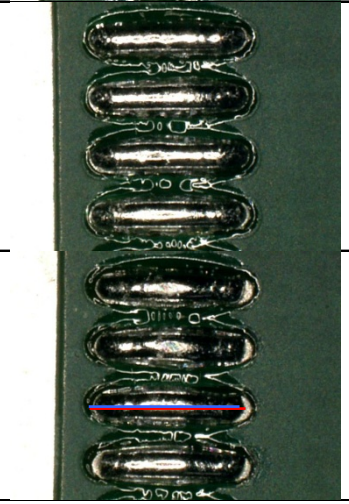
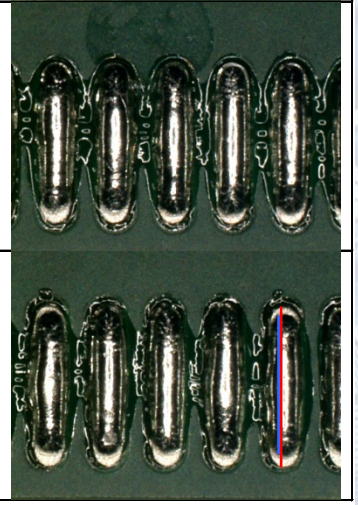
- Wetting (minimal spread) is main requirement
- Spread may be important for
  - Thermal transfer
  - Coverage of exposed metal
  - Solder joint inspection
- Can we use a spread test as a solderability requirement?


first reflow soldering


second reflow soldering

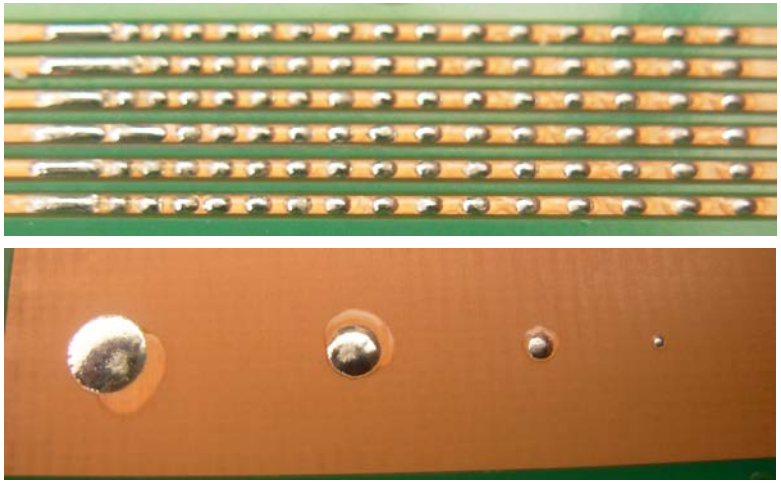
Comparison: Solder Spread to Pad Coverage

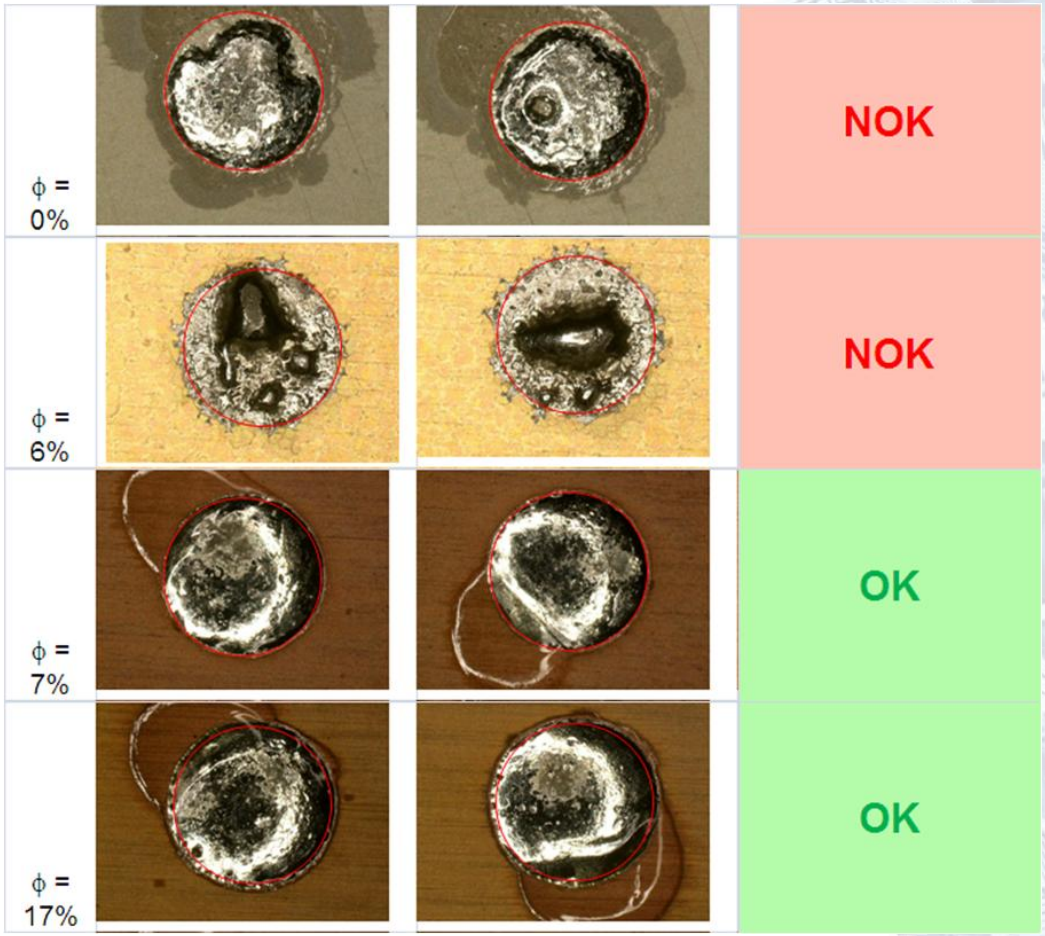
- Solderability criteria for 95% pad coverage
- Compare QFP pad coverage to the spread result from the same PCB
- ImSn: > 25-35% spread required (125 um [5 mil] stencil)

Pre-reflows before paste (Paste #3)	Solder spread, ϕ	Pad Coverage (Per J-STD-003B)		
0	100%	98 - 100%		
1	33%	93 - 100%		
2	25%	86 - 98%		

Comparison: Solder Spread Coupon to Spread

- Does the solder spread out or pull back?
- Visual comparison based on original solder deposit size

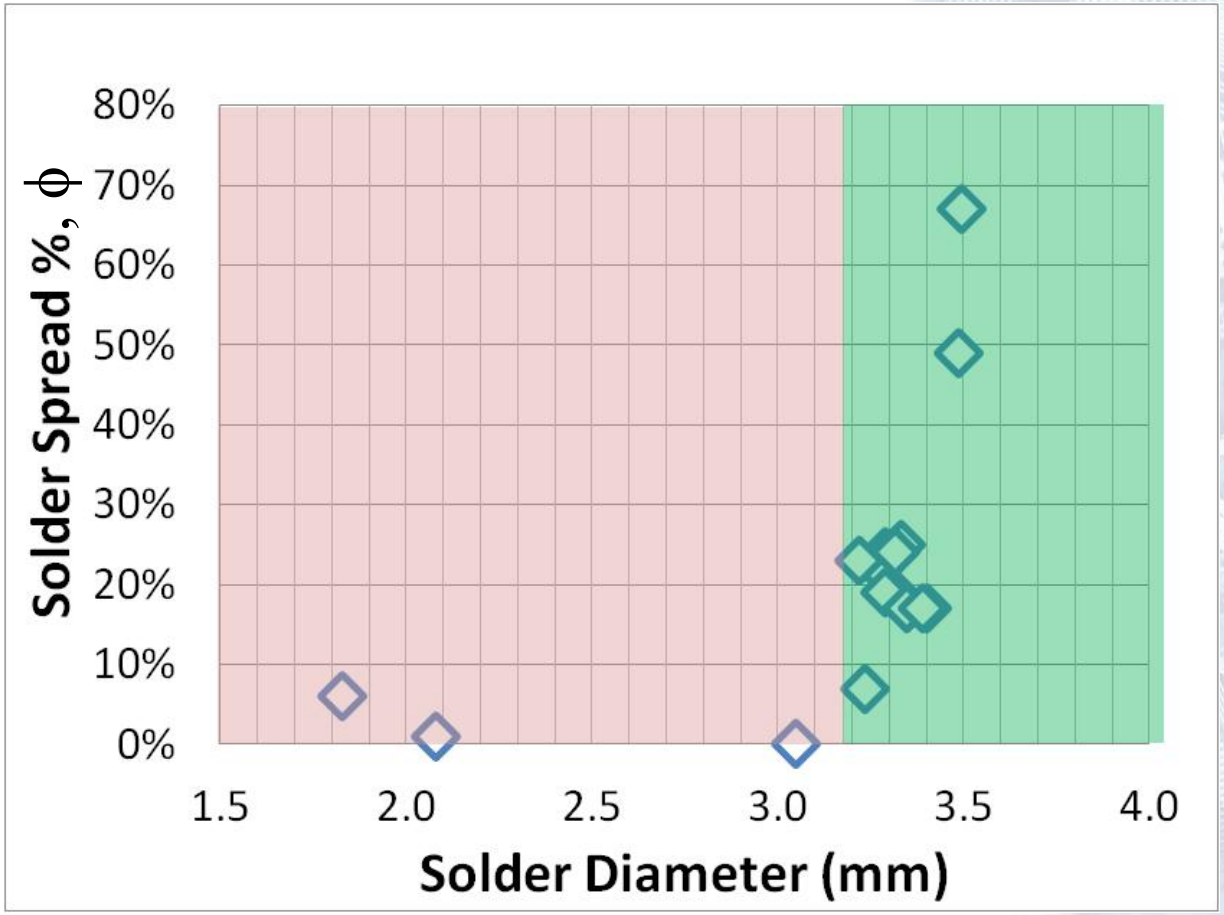




Comparison: Solder Spread Coupon to Spread

- Light microscope measurements of solder deposit diameter (3.175 mm is the original deposit size

- Spread > 15% is wetting
- Spread < 10% is generally dewetting
- Spread > 15% required



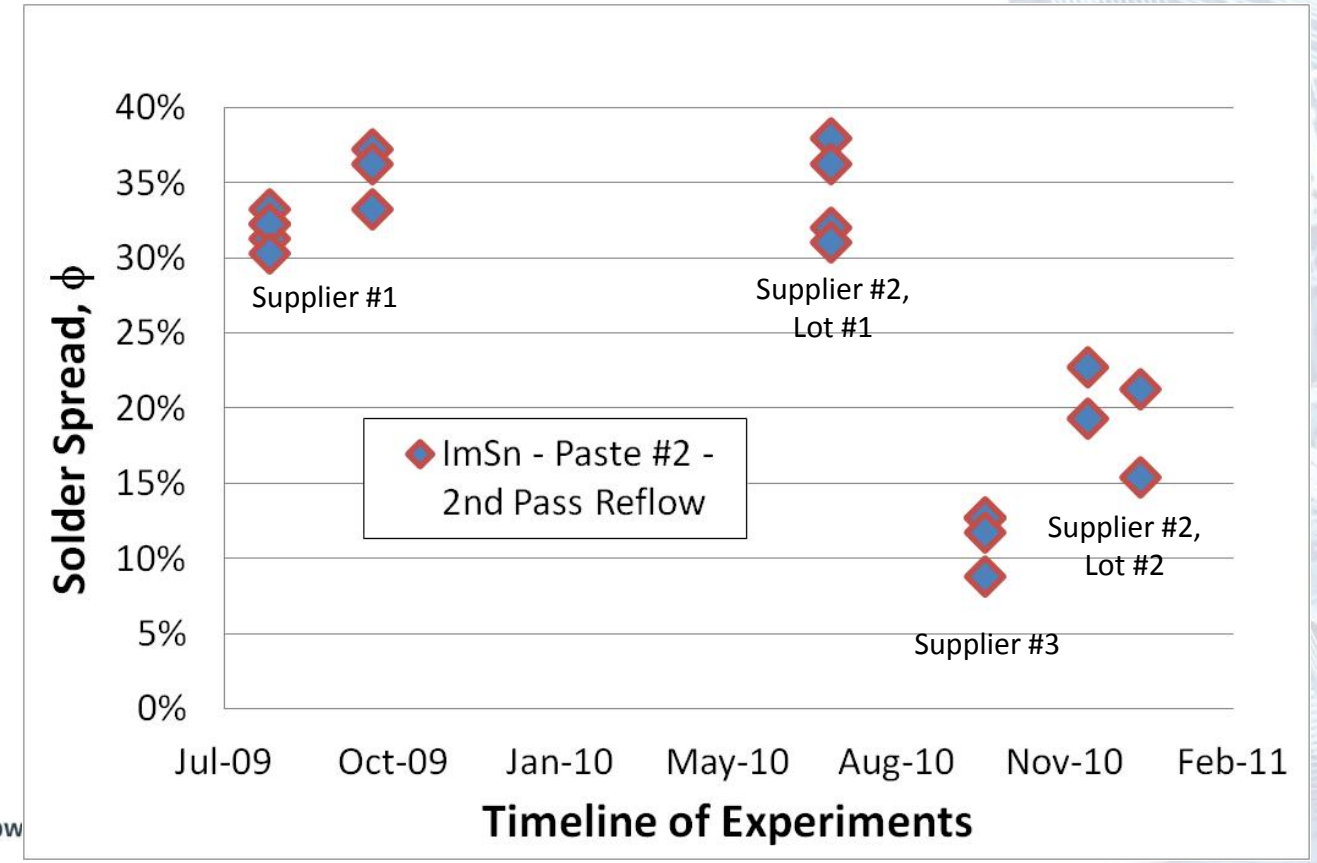
Correlation of Solder Spread to Measured Wetting Angle

- In progress
  - In cooperation with Theron Lewis, IBM Corporation

Are there limitations?
Lessons learned?

Repeatability

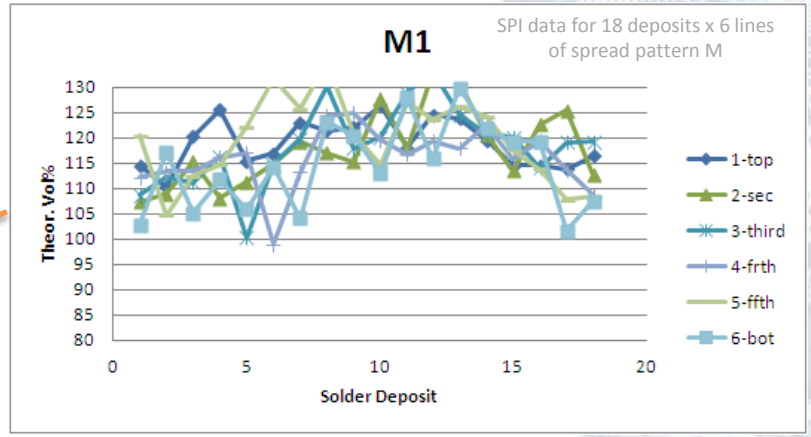
- Do we get the same results again and again?
  - Yes and no
- Hard to differentiate test repeatability from other factors
  - Different PCB suppliers
  - Different lots/batches
  - Age of boards
- Repeatable in general, but best to have self-contained experiment

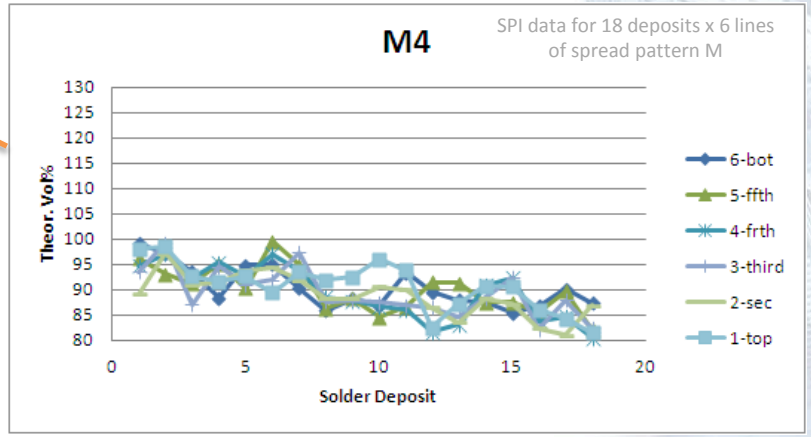


Influence of Solder Paste Printing

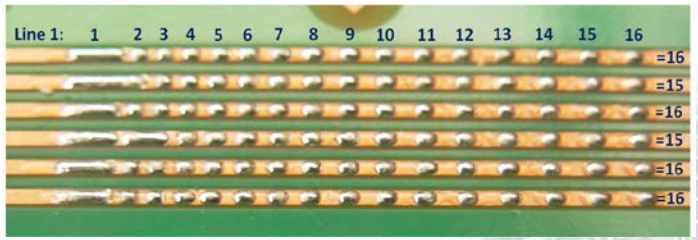
- Sometimes high variability in spread occurs on a single circuit board – why?
- Selected 4 boards with high variation in the 4 solder spread patterns
- Koh Young (SPI) data was available from board processing

	Spread %, by Coupon			
Board	**M1**	**M2**	**M3**	**M4**
G37	**68%**	52%	64%	**43%**
G36	52%	59%	44%	48%
G12	50%	39%	10%	5%
T15	29%	19%	18%	12%



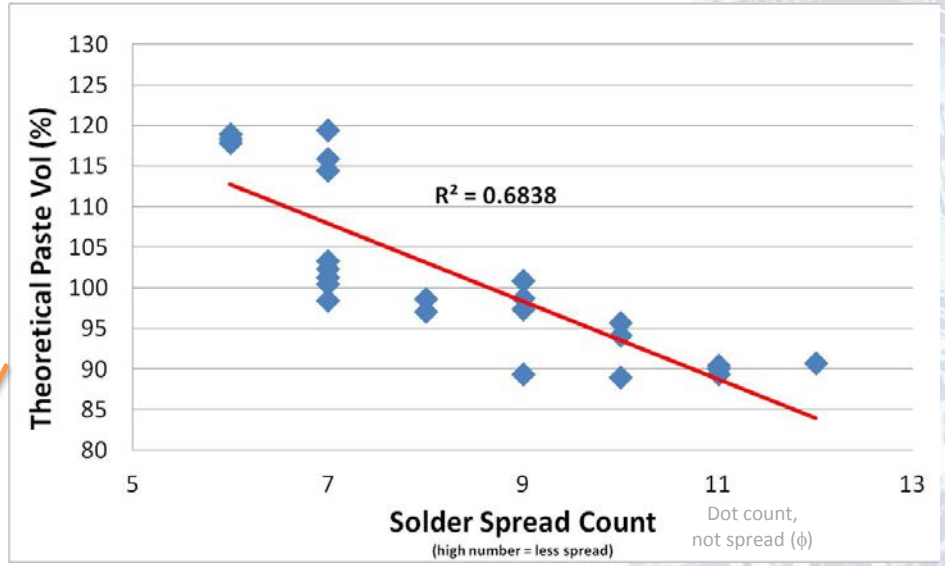


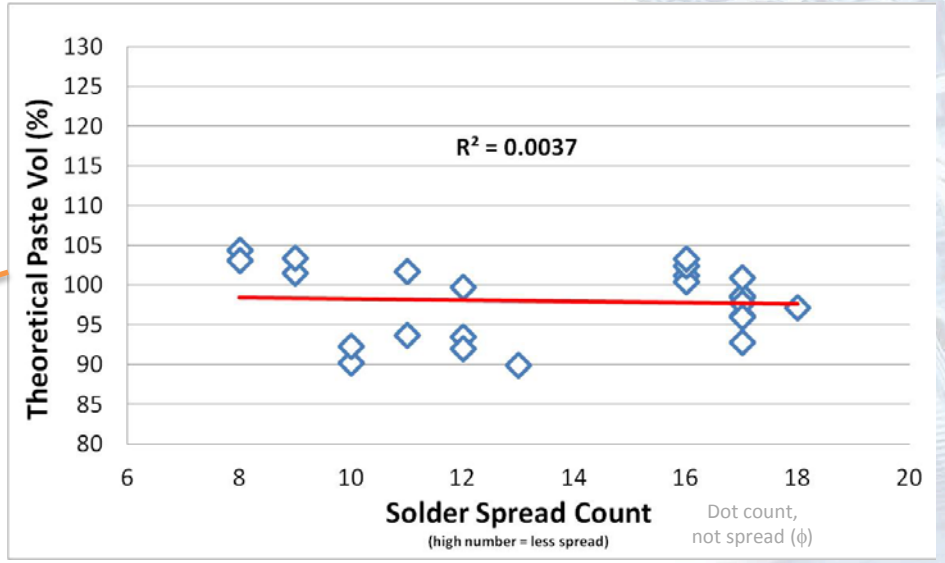
Influence of Solder Paste Printing



- Review of data from four boards (4 patterns each)
  - Average paste print volume vs. spread count for each individual line (24 lines per board)
- Keep print volume 90% < x < 110% to minimize impact on spread result
- Print variation explains some variation is solder spread, but not all spread variation is due to print differences

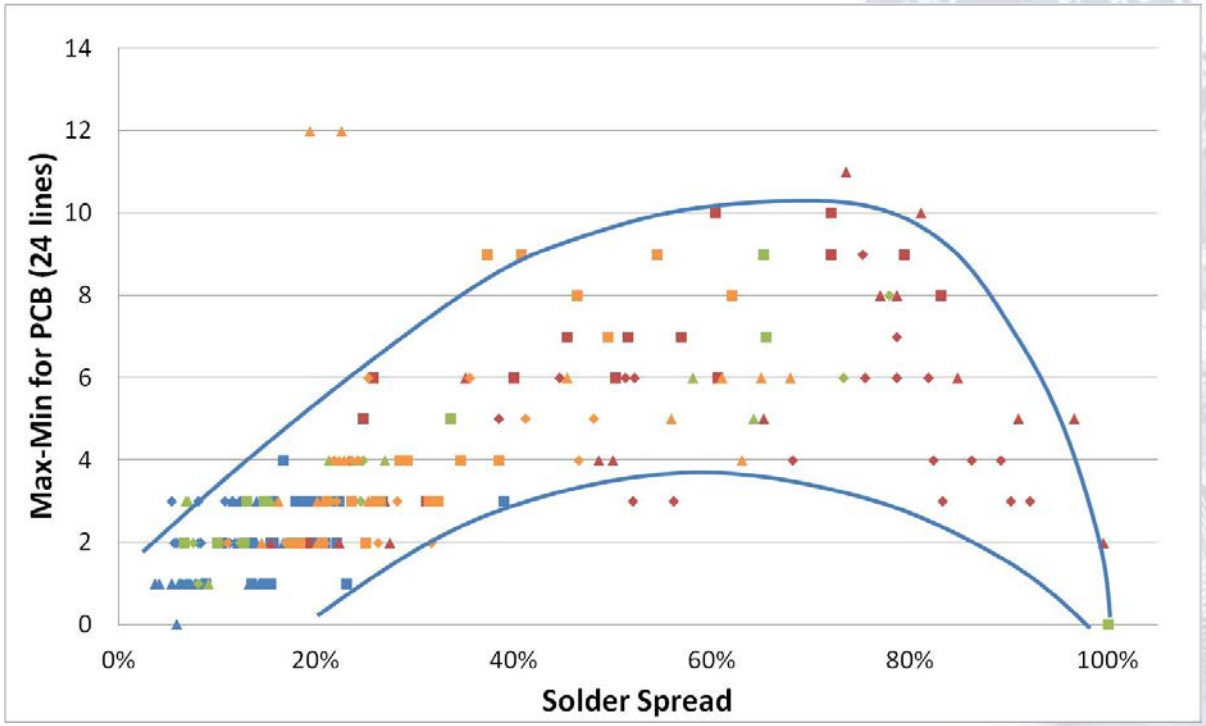
Board	Diff In Avg Paste Vol% (by line)	Correlation to Spread?
G37	**30.5%**	Yes
G36	12.3%	No
G12	**14.5%**	No
T15	29.6%	Yes





Variability of Spread Results on a Single Board

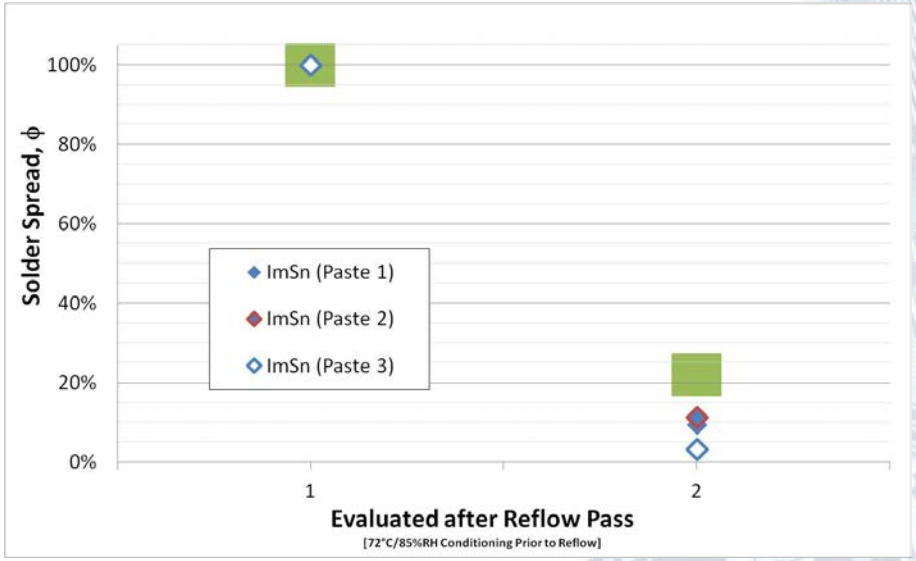
- PCB with 4 spread coupons = 24 lines with a spread count for each line
- For board: take max – min spread count (24 lines)
- More variation at 40-80%
- Minimal variability around proposed pass/fail at ~10-30%

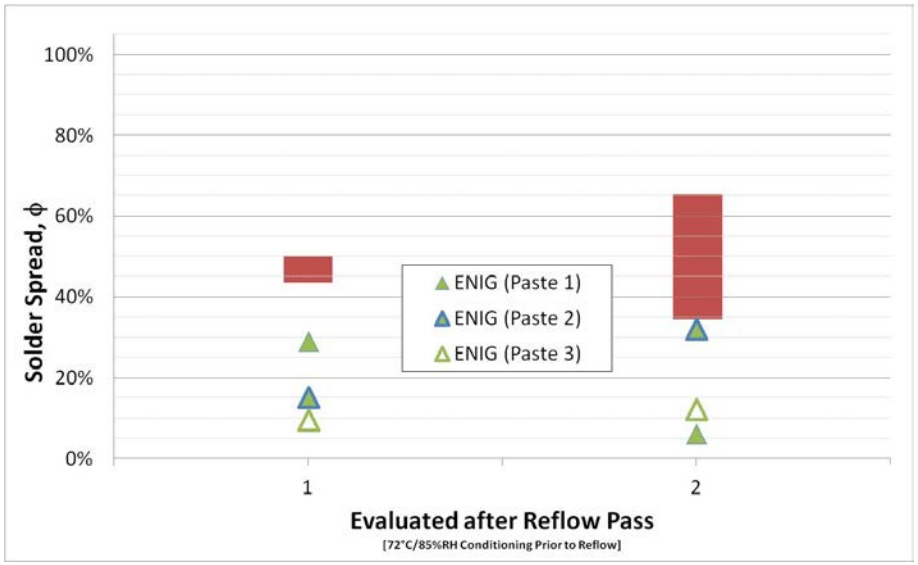


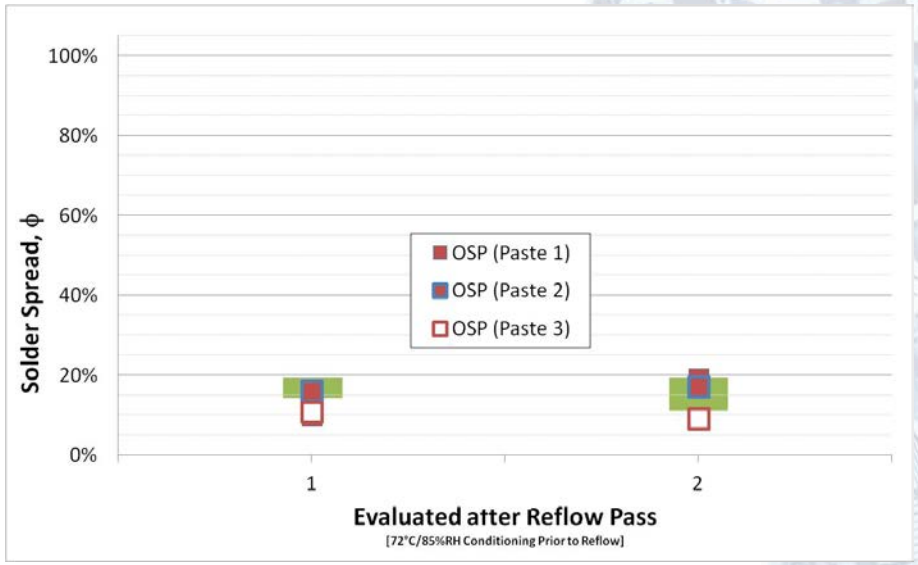
Additional Testing Possibilities

Effect of J-STD-003B Conditioning

- 8h 72°C/85%RH
  - Minimal impact on OSP
  - Noticeable decrease for ENIG
  - Impact only on 2nd pass reflow for ImSn (2x reflow not covered by J-STD-003B)







IPC 4-14

- Placeholder for IPC 4-14 ENEPIG round robin data

Summary

- The IPC solder spread pattern was used to
  - Compare the solderability of surface finishes
  - Evaluate the impact of multiple Pb-free reflows
  - Compare the spreading performance of solder pastes
- A pass/fail criteria is not yet defined
  - Current test results indicate minimum spread in the 10-33% range is required to meet established IPC standards
- For the test method
  - Repeatability was partially demonstrated
  - Limits to the solder paste print volumes were suggested to minimize the influence on the spread result
- Multiple reflow testing critical for some finishes

Acknowledgements

- Continental
  - Mark Fulcher – Deer Park, IL, USA
  - Mike Pomeroy, David VanBooven – Seguin, TX, USA
  - Jose Servin, Cynthia Gomez – Cuautla, Mexico
  - Emilio Alegre – Manila, Philippines