Mass Reflow Assembly of 01005 Components

Jeff Schake
DEK USA Inc.

Abstract
As the electronics assembly industry adjusts to building boards with 0201 components, the next level of shrinkage for chip components is just around the corner with the emergence of 01005s. A key objective of this work is to demonstrate assembly success using common 0201 build parameters of Type 3 solder paste and 4-mil stencil thickness. Stencil artwork strategies are implemented across 27 different pad designs to determine how assembly yield is influenced by aperture area ratio, component termination overlap level in paste, and tolerance to stencil defined misregistration levels. A pad design of 9 mils wide by 11 mils long and separated from the opposite pad by 6 mils is recommended when printed with 9 mil wide by 10 mil long apertures to achieve acceptable print quality and high assembly yield results. A direct comparison of tombstone ratios for 0201 and 01005 resistor components assembled together shows that 01005s are more stable across a variety of profile atmospheres compared to 0201s.

Introduction
With rising consumer demand for smaller high function portable electronic devices, it becomes ever more critical to creatively arrange circuits and components to support operation while satisfying cost and manufacturing assembly yield targets. While 0201 size resistor and capacitor components continue to grow in popularity to meet stricter form factor design constraints, the emergence of the 01005 passive is yet an even smaller size option now being offered. The assembly process concerns between 0201 and 01005 are quite similar, however, common expectation is that the smaller 01005 should behave more sensitively to process and design variables. There have already been several articles published on 01005 assembly and the common conclusion between these is to recommend fine powder solder paste (Type 4 or 5) used in conjunction with a thin stencil (3-mils).[1,2,3] It is the motivation of this work to challenge the previous two recommendations and prove or disprove the 01005 assembly process can be performed successfully with Type 3 powder size standard solder paste material using a 4-mil thickness stencil. This data should be useful to the typical SMT manufacturer who is already using Type 3 solder paste and may also be facing stencil thickness reduction to 4-mil thickness to accommodate small CSP and 0201s.

Process Design
Several experiment variables have been tested to determine sensitivity to 01005 resistor assembly yield, described in the following categories.

Circuit Board
The double sided 5.5″ x 8.0″ x 0.062″ FR-4 PCB was designed to accommodate 01005 components on Side A and both 01005 and 0201 components on Side B. Only Cu-OSP pads were considered. All pads were non-solder mask defined and without any mask inside the component outline. The majority of testing was conducted on Side A, which comprised of 27 different combinations of pad design, four levels side-to-side spacing, and two orientations. **Table 1** identifies the pad designs on Side A.

Table 1 - PCB Side A Pad Design Elements

Pad Factor	Level 1	Level 2	Level 3	Level 4
Width (W)	A = 7 mils	B = 9 mils	C = 11 mils	
Length (L)	D = 7 mils	E = 9 mils	F = 11 mils	
Separation (S)	G = 6 mils	H = 7 mils	I = 8 mils	
Pad to Pad (P)	4 mils	6 mils	8 mils	10 mils
Orientation (O)	0 deg	90 deg		

Side B consisted of fixed pad dimensions for both 0201 and 01005 components. **Table 2** clarifies the land details.

Table 2 - PCB Side B Pad Design Elements

Pad Factor	0201	01005
Width (W)	15 mils	8 mils
Length (L)	12 mils	9 mils
Separation (S)	6, 9 mils	6, 7.5 mils
Pad to Pad (P)	6 mils	6 mils
Orientation (O)	0, 90 deg	0, 90 deg

Stencil

There were three stainless steel laser cut stencils used in the assembly tests. All foils accommodated mounting on a 23″x 23″ frameless stencil system. **Table 3** lists the differences between them.

Table 3 - Stencil Design Features

Stencil	Thickness	Board Side Printed	Strategy
1	0.004"	A	Standard & Overprint Apertures
			Grab Level Test
			Print Offset Tolerance
2	0.004"	A	Decrementing Aperture Sizes
3	0.004"	B	0201 vs. 01005
			One Termination vs. Both

01005 aperture sizes designed for Stencil 1 were determined by applying pad to aperture size scaling from previous successful experience with 0201 assembly.[4] **Table 4** shows the equations used to generate the aperture sizes on Stencil 1. Each of the nine pad length and width combinations has both standard size apertures and overprint apertures. The solder paste is intended to print inside the pad boundaries using the “standard” aperture design while the “oversize” aperture means the solder prints outside the pad borders.

Table 4 - Stencil 1 Aperture Size Formulas

	0201 Pad		0201 Aperture		
	Length	Width	Length	Width	Cutback
Standard	12 mil	15 mil	11 mil	15 mil	N/A
Overprint*	12 mil	15 mil	14.6 mil	20 mil	2 mil

	01005 Scaled Aperture Equation		
	Length	Width	Cutback
Standard	Pad Length - 1 mil	Pad Width	N/A
Overprint*	Pad Length / 0.8219	Pad Width / 0.75	Aperture Length x 0.137

*Overprint aperture is bowtie shape

Two additional designed features of Stencil 1 include “grab” testing and print offsets. Grab is defined as the overlap between passive component termination and printed solder paste. The stencil has been designed to produce three grab levels per pad design as shown in **Figure 1**. Apertures positioned closest together will have the most solder residing under the component and will be considered the highest grab level indicated by “H”. Note that also the pad separation dimension (S) on the board also has a significant influence on the grab factor.

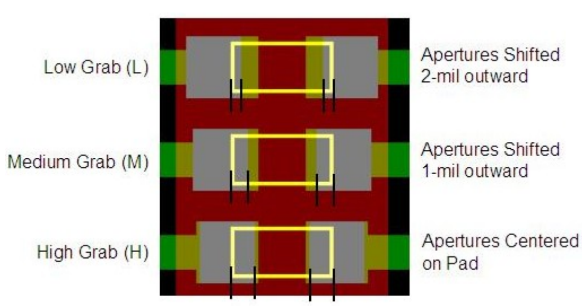


Figure 1 - Stencil Designed “Grab” Levels

While the majority of apertures on Stencil 1 are designed to be positioned symmetrically about the pad sets, there are also some pads designated to receive paste that has intentional print misregistration. Two levels of diagonally offset apertures at 0.5 mil and 1.0 mil increments are used to test assembly yield tolerance of print deposit position across pads.

Stencil 2 was designed to investigate the 01005 resistor assembly yield tolerance to gradually reduced aperture sizes across all pad designs. All apertures were centered on the pads for these tests. Largest apertures for any pad were set at a 1:1 design. From here, aperture width was reduced in 0.5 mil increments if the corresponding pad width exceeded length. If pad length exceeded width then aperture length was reduced in 0.5 mil increments instead. The pad length to width ratio was used as the scale factor to determine the remaining aperture dimension so that all decrementing aperture sizes would have the same aspect ratio with respect to the pad design. Matching length and width would both reduce by 0.5 mil increments.

Stencil 3 is the only stencil that prints on PCB Side B and contains aperture designs for both 0201 and 01005 components. There are three 0201 aperture designs, the smallest one being 1:1 with the pad and incrementing 1.0 mil larger per side on the two other designs. There are eight different 01005 apertures sizes, three of which are square shape from 8.0 mils to 10.0 mils. The remaining five 01005 apertures are rectangles ranging between 7.0 and 11.0 mils per side.

Solder Paste

Three standard commercially available solder pastes were used in the assembly experiments, as identified in **Table 5**. These were all Type 3 powder size and no-clean flux paste formulations. Two SAC305 materials were tested. The eutectic tin-lead paste is the same formulation we used to investigate the 0201 assembly process.[5]

Table 5 - Solder Paste List

	Vendor	Alloy
Paste I	A	96.5Sn/3.0Ag/0.5Cu
Paste II	B	96.5Sn/3.0Ag/0.5Cu
Paste III	A	63Sn/37Pb

Printing

A high-speed automatic stencil printing machine with programmable fiducial location was used for depositing the solder paste onto the circuit boards. The printing parameters listed in **Table 6** were fixed for all builds. Selection of these settings is based on the recommendations described in an earlier report focusing on 01005 printing.[6]

Table 6 - Print Parameters On Assembled Boards

Parameter	Setting
Print Direction	Rear to Front
Squeegee	Metal, 60 deg, 250mm long
Print Speed	50 mm/sec
Print Pressure	5.0 kg
Separation Speed	20 mm/sec
Print Gap	0 mm
Prior Dummy Prints	3
Stencil Clean	Once before dummy prints

Placement

A standard SMT 4-spindle automatic placement machine outfitted with prototype nozzles was used to assemble the 01005 resistors. A schematic of the nozzle design used is shown in **Figure 2**. A 0.4 mil per pixel magnification upward looking camera was installed on the tool to inspect for component presence and location on the nozzles. No modifications were made to the electronic feeder to index the components. Default machine settings were used to operate a normal pick and place sequence.

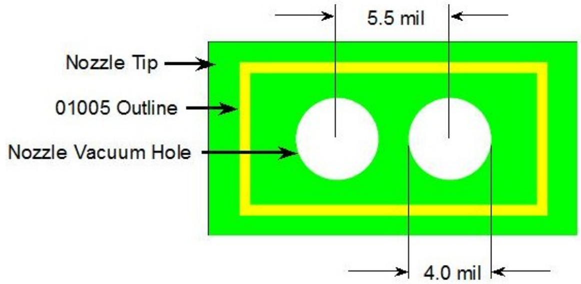


Figure 2 - Nozzle Tip Design (Shown To Scale)

Reflow

An eight heating zone forced hot air convection furnace with board edge supporting adjustable conveyer rails was used to reflow assembled boards. The system is compatible with achieving elevated temperatures required for Pb-free assembly and also plumbed for nitrogen capability. For all assemblies reflowed in nitrogen the atmosphere is consistently recording O_2 levels below 50ppm in the final heat zone. Convection was set to the maximum 3500rpm fan speed setting and top/bottom zone temperatures settings were balanced. The temperature profiles used were programmed to match the paste vendors' recipes listed in the material technical data sheets, and displayed in **Figure 3**.

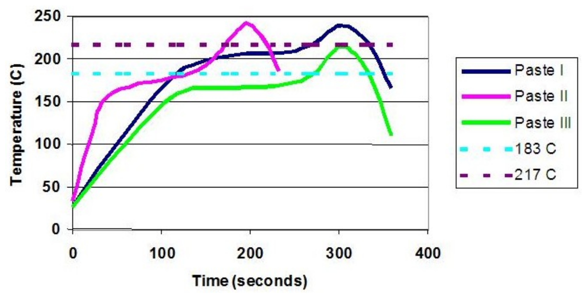


Figure 3 - Reflow Profiles

Assembly Tests

The 01005 components used for assembly were resistor types only. The termination metal on these passives is 100Sn. The averaged dimensions from a sample of measured components are shown in **Table 7**. Note this component more accurately resembles a metric 0402 rather than a true "01005" size.

Table 7 - 01005 Resistor Dimensions

	Imperial	Metric
Length	0.015"	0.381 mm
Width	0.0075"	0.191 mm
Height	0.0043"	0.109 mm
Termination Length (Contact Side)	0.005"	0.127 mm

A total of 12 circuit boards were assembled. Builds are grouped according to the stencil used and listed with the specific experimental condition details in **Tables 8-10**.

Table 8 - Stencil 1 Builds

Build	Paste	Reflow Atm.	Board Side	Passive Orientation	Pad to Pad	Stencil Grab Test	Diagonal Print Offsets	Lowest Area Ratio	Pad Designs Assembled	Replicates	Total Passives
1	I	N2	A	90 deg	6 mil	All 3	No	0.51	15 of 27	8	360
1	I	N2	A	90 deg	10 mil	All 3	No	0.51	All 27	4	324
2	III	N2	A	90 deg	6 mil	All 3	No	0.51	15 of 27	8	360
2	III	N2	A	90 deg	10 mil	All 3	No	0.51	All 27	4	324
3	III	N2	A	0 deg	6 mil	High level	No	0.51	5 of 27	4	20
3	III	N2	A	90 deg	4 mil	All 3	No	0.51	15 of 27	4	180
3	III	N2	A	90 deg	6 mil	All 3	0.5, 1.0 mil	0.51	15 of 27	4	360
3	III	N2	A	90 deg	10 mil	All 3	1.0 mil	0.51	12 of 27	4	144
4	I	N2	A	0 deg	6 mil	High level	No	0.51	5 of 27	4	20
4	I	N2	A	90 deg	4 mil	All 3	No	0.51	15 of 27	4	180
4	I	N2	A	90 deg	6 mil	All 3	0.5, 1.0 mil	0.51	15 of 27	4	360
4	I	N2	A	90 deg	10 mil	All 3	1.0 mil	0.51	12 of 27	4	144
5	I	Air	A	90 deg	4 mil	All 3	No	0.51	15 of 27	4	180
5	I	Air	A	90 deg	6 mil	All 3	No	0.51	15 of 27	8	360
5	I	Air	A	90 deg	10 mil	All 3	No	0.51	12 of 27	4	144

Table 9 - Stencil 2 Builds

Build	Paste	Reflow Atm.	Board Side	Passive Orientation	Pad to Pad	Lowest Area Ratio	Pad Designs Assembled	Replicates	Total Passives
6	I	N2	A	90	8 mil	0.46	18 of 27	4	360
7	I	Air	A	90	8 mil	0.46	18 of 27	4	360
9	II	N2	A	90	8 mil	0.46	18 of 27	4	360
10	II	Air	A	90	8 mil	0.46	18 of 27	4	360

Table 10 - Stencil 3 Builds

Build	Paste	Reflow Atm.	Board Side	Passive Orientation	Pad to Pad	Lowest Area Ratio	Pad Designs Assembled	Replicates	Total 0201 Passives	Total 01005 Passives
11	II	N2	B	0, 90	6 mil	0.5	All 4	4	144	384
12	II	Air	B	0, 90	6 mil	0.5	All 4	4	144	374
13	I	N2	B	0, 90	6 mil	0.5	All 4	4	144	383

Stencil 1 Builds

Build 1 & 2: The strategy for these builds was to determine if the 01005 resistor assembly process would show any sensitivity to pad design, stencil induced grab, and standard vs. oversize print deposit. Nitrogen was present in reflow to promote wetting and enhance occurrence of tombstoning. Not all pad designs were assembled on the 6-mil pad to pad spacing as it was considered impractical to assemble components onto sites that were printed with aperture area ratio below 0.5.[7] Component orientation was selected at 90° only so that one termination would follow the other into reflow, which was found to increase tombstoning in earlier 0201 assembly testing.[4,5] The difference between Build 1 and Build 2 is the solder alloy and reflow profile. Inspection of the reflowed boards from both builds showed no bridging, very low occurrence of solder balls, and no open solder joints. This introduces evidence that a wider than expected pad and stencil design process window may exist to achieve high yield 01005 resistor assembly using both SAC305 and eutectic Sn/Pb solder. Several pad designs printed with the "oversize" apertures that were assembled show instances of floating components. An example of this is shown in **Figure 4**. The pad designs that show excess solder delivery by oversized apertures are shown in **Figure 5**.

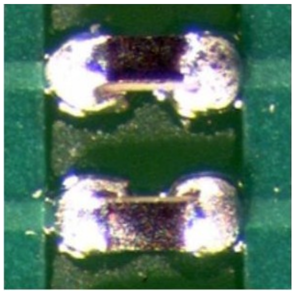


Figure 4 - Excess Solder Joints On Pad CEG

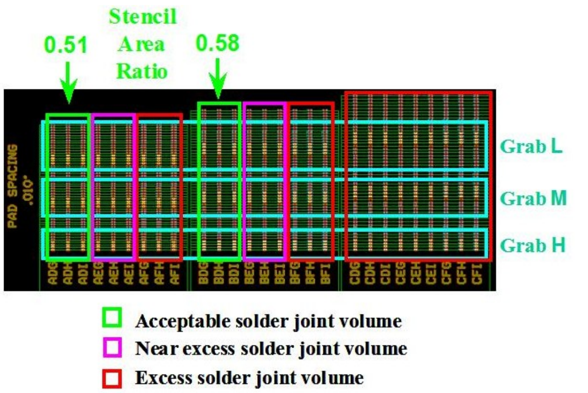


Figure 5 - Oversize Aperture Solder Joint Volume Inspection

Build 3 & 4: This time more challenging component placement locations were programmed in this run to encourage assembly yield loss. Nitrogen was again active in reflow to promote wetting. One board was printed with Sn63/Pb37 solder and the other with SAC305 solder. Only "standard" aperture designs were assembled. The finest side-to-side pad spacing of 4 mils were populated at 90° for all three stencil grab levels wherever the print area ratio exceeded 0.5 shown in **Figure 6**. There were two tombstoned components located on pad CFI at the lowest stencil designed grab level for Build 3, corresponding to the eutectic Sn/Pb assembly. The Pb-free assembled Build 4 board did not produce any defects. There were no bridging defects observed on both builds at this component placement density.

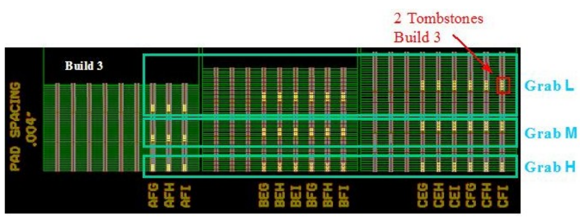


Figure 6 - Component Locations For 4-mil Pad Spacing

Figure 7 shows a comparison between leaded and non-leaded solder paste assembly on pad design BEG using the largest paste grab "standard" aperture design at 4-mil pad spacing. Note the significant difference in wetting between the two alloys. Even with a high concentration of nitrogen present in reflow, the lead free solder paste tends to form bulging solder joints at the terminations due to reduced wetting.

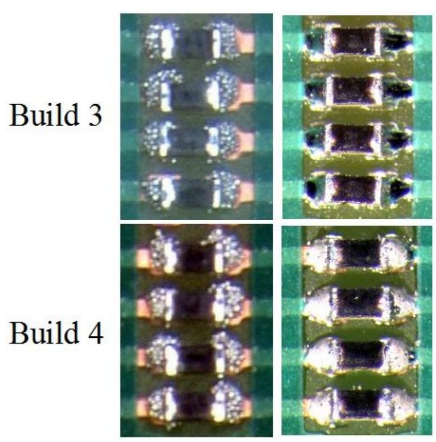


Figure 7 - Successful Assembly On 4-mil Pad Spacing

Pad sites were also added to the assembly list that printed with intentional diagonal print misregistration of 0.5 and 1.0 mil at 6-mil pad side-to-side spacing. Pad designs were not assembled for print area ratios below 0.5. All three stencil grab levels were assembled on print area ratio qualified pads. The eutectic Sn/Pb Build 3 showed ten tombstone defects while the Pb-free Build 4 did not have any defects. One tombstone defect was observed for pad width “B” and the remaining nine occurred in pad width “C”. These defect locations are shown in the board map on **Figure 8**.

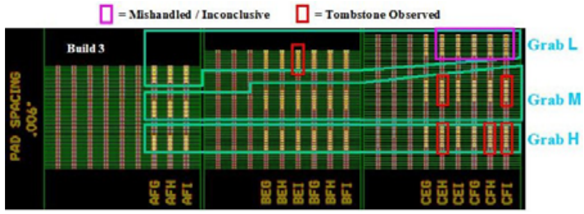


Figure 8 - Assembly Defect Locations For Print Offset Test

Figure 9 shows a pad design BEI with simulated 1.0 mil diagonal offset print and accurate 01005 placement. The west positioned paste deposit barely contacts the left termination, while the east positioned paste deposit still overlaps the right termination. The grab imbalance caused by the print misregistration at highest pad separation “I” and lowest designed stencil grab level creates a condition that favors paste climbing the right termination during reflow and pulling the component to the tombstoned state as shown. Pb-free paste has shown more resistance to tombstoning, likely explained be a weaker wetting force being exerted on the terminations, allowing greater tolerance for paste misregistration and grab imbalance.

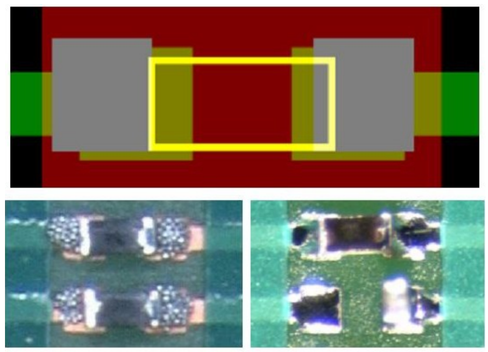


Figure 9 - Print Offset & Grab Influence On Tombstone, Sn63 Solder

A sample of 01005 resistors was also assembled at 0° rotation to determine orientation sensitivity. No influence on assembly yield was observed.

Build 5: This build used the same component placement locations and Pb-free solder paste as Build 4. The only difference with this assembly is that the profile atmosphere was changed from nitrogen to air. This combination of solder paste and profile produced a significantly defective result. Nearly all deposits showed incomplete solder coalescence. Larger deposits, particularly on the largest width pads, show better results (**Figure 10**). Assembly yield conclusions for this build were unfeasible due to challenges with reflow.

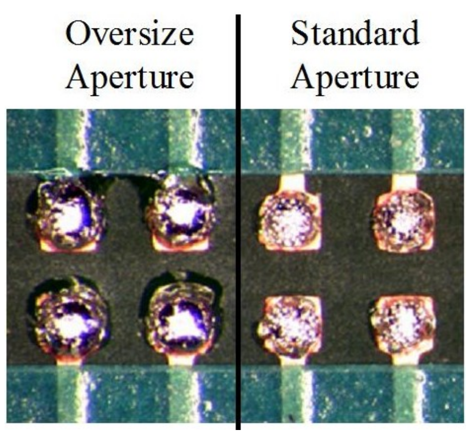


Figure 10 - Poor Reflow Of Lead Free Paste I, CFI Pad Design

Stencil 2 Builds

Build 6, 7, 9, & 10: This stencil was designed with largest aperture designs to be sized 1:1 with pads and then decreasing in size to 0.46 area ratio for determining the influence of solder volume on assembly yield. **Figure 11** illustrates the aperture design and assembly plan.

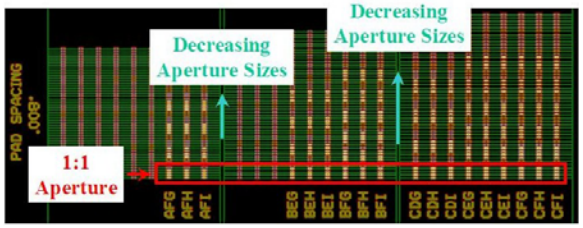


Figure 11 - Assembly Strategy For Stencil 2 Builds

All four boards were printed with Type 3 SAC305 Pb-free solder paste, two using vendor A and two from vendor B materials. For each paste vendor, one board was reflowed in nitrogen and the other in air. A new jar of the same paste printed in Build 5 was opened and used here to determine if fresh material would improve air reflow performance. Indeed there was some improvement in Pb-free reflow noted with new Paste I, as reflow coalescence did not appear to be as sensitive to changes in printed solder volume. However, the solder joints formed from Paste I in air reflow did exhibit some granularity on most deposits. SAC305 Paste II displayed better reflow performance in air compared to Paste I with incomplete coalescence only occurring on pads where paste transfer was starved due to aperture clogging. **Figure 12** shows difference in appearance between solder joints formed with the two solder pastes across nitrogen and air reflow conditions.

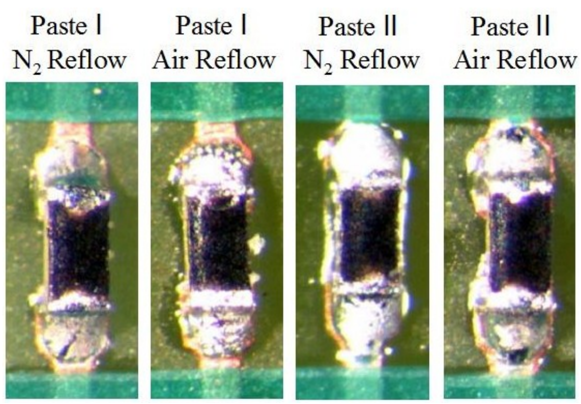


Figure 12 - Reflow Atmosphere Influence On SAC305 Solder Joints

There were no occurrences of shorting or solder balling defects witnessed. At locations where both component pads receive printed solder, no open joints resulted. As the gradually reduced aperture sizes starve solder away from the pads, there is also a tendency for unbalanced paste volume distribution to occur across terminations. In some extreme cases, apertures may be totally clogged at low area ratios so that only one component termination is placed into solder. This condition was observed to produce no instances of tombstones, but did result in a few occurrences of low angle drawbridging. **Figure 13** shows an example of such an observation. Further investigation into this behavior is the focus of testing with Stencil 3.

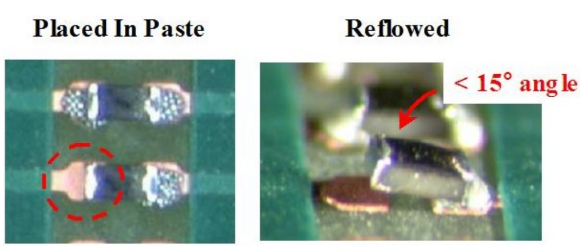


Figure 13 - Evidence Of Tombstone Resistance

Stencil 3 Builds

Build 11, 12 & 13: With the unexpected absence of open solder joints from all previous builds, the purpose of these assemblies was to study the tombstone trends between two Pb-free solder pastes on the effect of printing solder on only one of the two termination attachment pads for both 0201 and 01005 resistors using PCB Side B. The design of the 0201 pads reflected a combination that was found to produce the highest assembly yield from earlier testing.[4,5] Components that exhibited an angle of inclination and not laying flat on any of the non-printed pads was counted as a tombstone. Results are plotted in **Figure 14**.

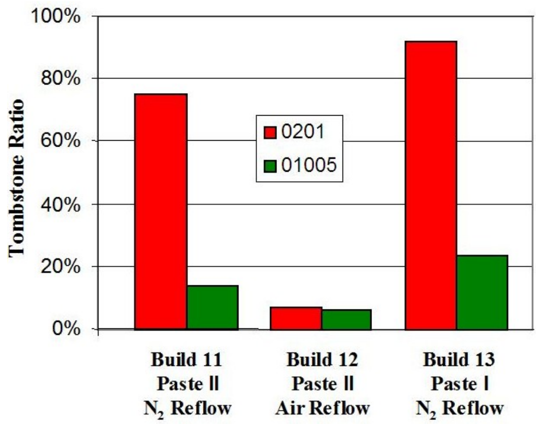


Figure 14 - Tombstone Trends Between 0201 & 01005 Resistors

The frequency of tombstones occurring on 0201 resistors is much greater than 01005s when the reflow atmosphere is nitrogen. This observation is consistent across both Paste I and Paste II materials using their respective unique thermal profiles. **Figure 15** shows a typical view of 0201s and 01005s from an assembled board that was reflowed in nitrogen. The occurrence of tombstones across both passive types is nearly the same level when the reflow atmosphere is switched air. Although both component types show greater resistance to tombstoning when reflowed in air, the effect of inerting the reflow atmosphere is significantly more influential on causing 0201s to tombstone. There was no correlation found to show the influence of aperture size on tombstone trends for either 0201 or 01005 components.

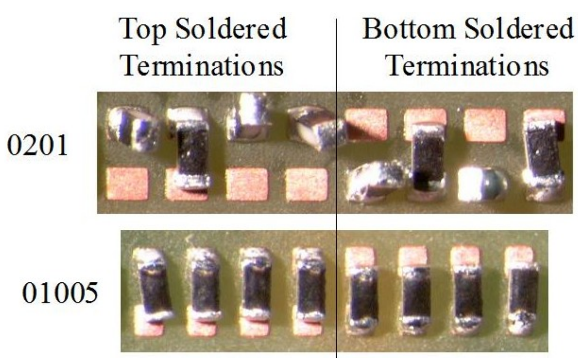


Figure 15 - Comparison Of Nitrogen Reflowed Passives

Samples of each component type were also assembled on pairs of printed pads as the experiment control. No tombstones were found under this assembly condition for either component across the three builds.

Discussion

Under the constraint of using a 4-mil thick stencil, the solder paste transfer efficiency prediction for 01005-scaled apertures is low due to low area ratio stencil designs used. Despite this challenge, several pad designs showed promise for achieving consistent enough solder deposit formation using "standard" aperture designs to produce promising assembly yield results. The replication of print success shown here will be influenced by the formation of a proper seal between apertures and pads, particularly using low area ratio stencil designs where solder adhesion is already biased more strongly on aperture walls. The effect of board topography (i.e. solder mask thickness and registration), which may vary significantly across applications, is expected to have considerable impact on 01005 process stencil gasketing.

Using Type 3 solder paste the print deposit shapes produced by < 0.6 area ratio apertures will not be well formed bricks, rather more resembling rounded mounds of solder that contour wider at their base. Placement machines with delicate nozzles that drop components onto printed solder deposits may be challenged to cope with such print deposit shapes since free falling components tend to land in odd positions on non-flat solder deposits. For conventional direct forced placement into printed solder deposits, tapered print deposit shapes are compatible.

Some other observations regarding 01005 resistor placement worth noting include nozzle design and ESD. Component insertion into the paste not only flattens the solder down, but it squeezes material out laterally and also upwards along the termination sides. Since the solder deposit thickness is nearly the height of the 01005 resistor itself, the solder paste can easily access the nozzle during placement. The nozzles used in this study (**Figure 2**) experienced periodic episodes of solder paste accumulation on the ends of the hard deck tip. The effect of solder debris contaminating the nozzles contributed significantly to pick yield loss and required regular nozzle cleaning to minimize rejected components. Pad designs with widest separation (S) and low stencil grab were found to have the highest influence on solder contamination on the nozzle. Design of the nozzle tip should consider suitable sizing to inhibit this.

Design of 01005 component placement should also consider implementation ample ESD controls. Selection of static resistant materials for nozzles and feeders that rapidly dissipate any charge produced due to removal of the component cover tape are strongly suggested.

Utilization of an air reflow process is considered the most attractive atmosphere choice from a cost perspective. This process produces lower molten solder wetting strength, which is favorable to minimize tombstones and bridging, but tends be more difficult generating the molten condition for 01005 size solder deposits, particularly for Pb-free solder. This study identified paste age as one factor that affected air reflow performance (i.e. Build 5 vs. Build 7). Evidence was found that the useful life of Pb-free solder paste could be extended when implementing a nitrogen reflow atmosphere. A difference in air reflow performance between two Pb-free materials, Paste I and Paste II, was also observed. It is unclear whether this difference is due to material ingredient variations alone, or if there is any thermal profile sensitivity. It has been suggested that improved air reflow results can be achieved using faster heating.[3]
This theory is supported by observations in this research showing better air reflow coalescence of Paste II, which uses more aggressive heating than Paste I.

Recommendations
The following designs are suggested for 01005 resistor assembly compatibility with stencil thickness of 4 mils and Type 3 solder paste. The ranking strategy for pad designs is shown in **Table 11**.

Table 11 - 01005 Resistor Pad Design Ranking

Pad	Area Ratio	Narrow	Symmetry	Tombstones	Demerits
ADG	0.40	Poor	OK	No	2
ADH	0.40	Poor	OK	No	2
ADI	0.40	Poor	OK	No	2
AEG	0.47	Poor	OK	No	2
AEH	0.47	Poor	OK	No	2
AEI	0.47	Poor	OK	No	2
AFG	0.51	Poor	Poor	No	2
AFH	0.51	Poor	Poor	No	2
AFI	0.51	Poor	Poor	No	2
BDG	0.45	OK	OK	No	1
BDH	0.45	OK	OK	No	1
BDI	0.45	OK	OK	No	1
BEG	0.53	OK	OK	No	0
BEH	0.53	OK	OK	No	0
BEI	0.53	OK	OK	Yes	1
BFG	0.59	OK	OK	No	0
BFH	0.59	OK	OK	No	0
BFI	0.59	OK	OK	No	0
CDG	0.49	OK	Poor	No	2
CDH	0.49	OK	Poor	No	2
CDI	0.49	OK	Poor	No	2
CEG	0.58	OK	OK	No	0
CEH	0.58	OK	OK	Yes	1
CEI	0.58	OK	OK	No	0
CFG	0.65	OK	OK	No	0
CFH	0.65	OK	OK	Yes	1
CFI	0.65	OK	OK	Yes	1

There are four categories used to evaluate the designs. "Standard" aperture sizes are considered only, with designs having print area ratio below 0.5 scored with a demerit, indicated by red font. The second category labeled "narrow" compares the pad width dimension to the 01005 resistor width. Due to all the pad width "A" designs being narrower than the component width, these are all designated "poor" as this scenario not generally advised. The third category is "symmetry" which assigns demerits to aperture designs that are most rectangular. The logic here is that more non-symmetrical apertures will have unequal print transfer results comparing 0° and 90° orientations. The final category of tombstones represents locations where this occurs on Build 3. This is the only PCB Side A assembly where such defects were observed. A count of all the demerits (red labels) is shown in the final column. The summation of demerits is used to determine the qualified pad and stencil designs. The top three pad and stencil designs are BF, CE, and BE. Ranking in this order follows area ratio from high to low, so BF should produce better print results than the other two designs. Of the three pad separations, level "G" did not show any tombstone defects and is considered the recommended design. At pad separation "G" the influence of aperture grab level was not found significant on causing defects. The highest aperture design grab level "H", where apertures are centered on the pads, is suggested as this should help to relax print and placement accuracy requirements.

Pad designs BFG, CEG, and BEG and "standard" size high grab stencil aperture designs are all capable of comfortably tolerating ±20μm print and ±60μm placement misregistrations while still maintaining component contact into printed solder deposits. The worst-case scenario is shown simulated in **Figure 16**. Larger pad separation and/or lower aperture grab designs will further challenge the ability of this offset scenario to recover in reflow.

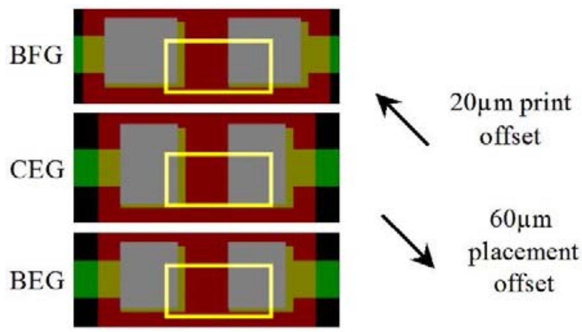


Figure 16 - Print / Placement Offset Scenario

With concern on achieving sustained consistent paste transfer results for low area ratio apertures, there are some additional changes that can increase printing success. The upgrade to electroformed stencils should increase print process stability offering lower surface roughness aperture walls to encourage highest paste release efficiency performance.[8] Another enhancement is replacing the squeegees with modern low friction metal blade options that are commonly available.[8] Such materials improve solder paste roll circulation during the print and also reduces the tendency for paste pulling off the stencil when the squeegee is retracted after completing the print. Enclosed print head technology can also be used in place of squeegees to provide a boost to transfer efficiency performance. [10]

The performance of "oversize" aperture designs was broadly successful in achieving repeatable high transfer efficiency solder deposits. However, the prevailing flaw with this stencil design strategy was the delivery of excess solder to pads causing bulging solder joints and the occasional floating component. Referring back to **Figure 5**, the only pad design recommend for use with the oversized aperture is pad design BD. Again the lowest pad separation "G" and highest grab aperture design are advised.

Conclusion

A successful 01005 resistor assembly process has been demonstrated using a 4-mil thick stencil with both Type 3 leaded and lead free alloy solder pastes across a variety of Cu/OSP pad sizes. A side-to-side pad spacing as low as 4-mil showed promising assembly yield results. The pad design BFG with 9.0 mil wide x 10.0 mil long apertures centered on pad is recommended. The 01005 resistors assembled with lead free solder are found to be significantly less sensitive to tombstoning when reflowed in nitrogen compared with 0201 resistors assembled in the same manner. Similar defect ratios are found between the two component types if reflow is conducted in air. A lead free air reflow 01005 assembly process is considered the most desirable industry practice and proof of achieving this has been realized. However, this work also reveals observations of incomplete solder coalescence suggesting that air reflow compatibility challenges may exist with small lead free solder deposits.

Acknowledgements
Thanks to Ashok Viswanathan from Binghamton University for assistance with the component assembly process and many constructive discussions that helped shape this work. Also compliments to the staff of the Unovis Advanced Process Laboratory for supporting the assembly and inspection equipment requirements to complete this project.

References
[1] Mattsson, F., Geiger, D., Shangguan, D., and Castello, T., "Design and Assembly of 01005 Passives Using Pb-Free Solder", Circuits Assembly, 2005, 16, 5, pp 22-27.
[2] Grundy, P. and Magnell, M., "Process Optimization of an 01005", Surface Mount Technology, 2006, 20, 5, pp 26-28.
[3] Borkes, T. and Groves, L., "Process Characterization of the 01005 (English) Component Package", *Proceedings from the SMTA International Conference*, September 2006, Chicago, IL.
[4] Adriance, J. and Schake, J., "Mass Reflow Assembly of 0201 Components", *Proceedings from the IPC APEX Conference*, March 2000, Long Beach, CA.
[5] Westby, G., Adriance, J., Prinz von Hessen, W., Schake, J. and Barbini, D., "0201 Issues and Process Window", September 2000 webcast, http://webevents.broadcast.com/cahners/universal0900/home.asp.
[6] Viswanathan, A., Schake, J., and Srihari, K., "Process Characterization for the Assembly of 01005 Components", *Proceedings from the SMTA International Conference,* September 2006, Chicago, IL.
[7] Area ratios are calculated as per IPC-7525 "Stencil Design Guidelines".
[8] Coleman, W.E. and Burgess, M.R., "Choosing a Stencil," Surface Mount Technology, 2006, 20, 7, pp 14-17.
[9] Mukadam, M., Borgesen, P., and Srihari, K., "Assembly Process Results for 0.4mm Pitch CSPs", *Area Array Consortium 2003 Report*, Universal Instruments Corporation, Binghamton, NY.

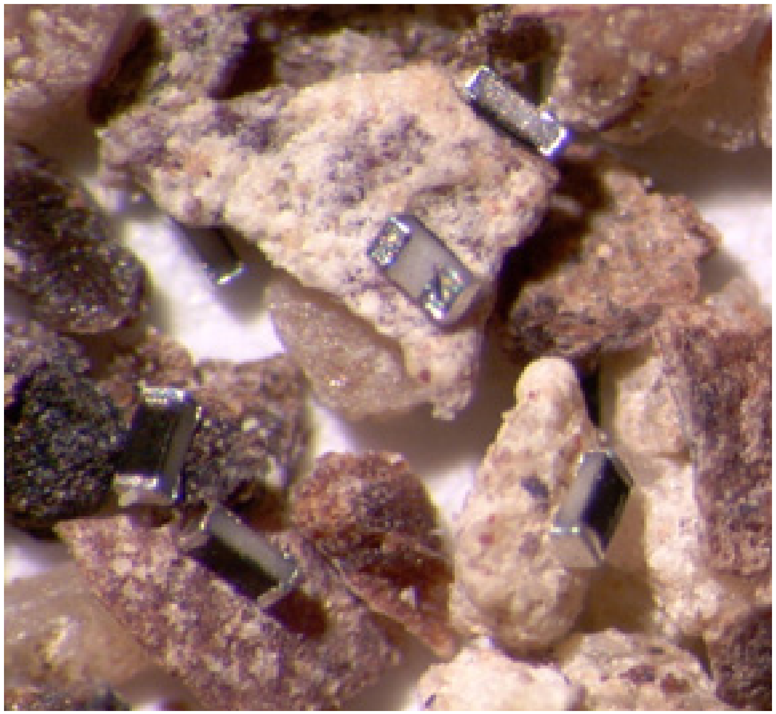
Mass Reflow Assembly of 01005 Resistor Components

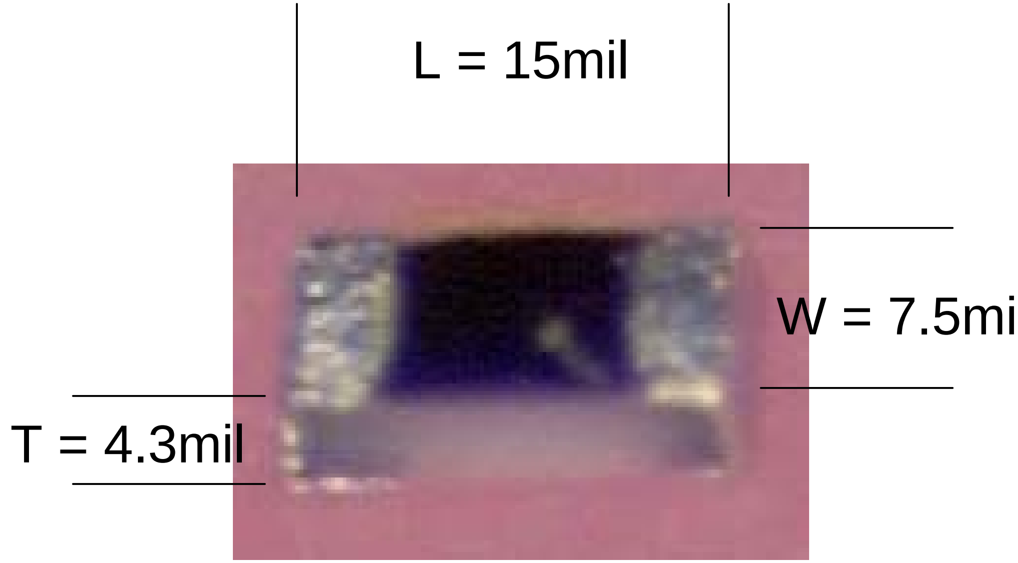
Jeff Schake

DEK USA inc.

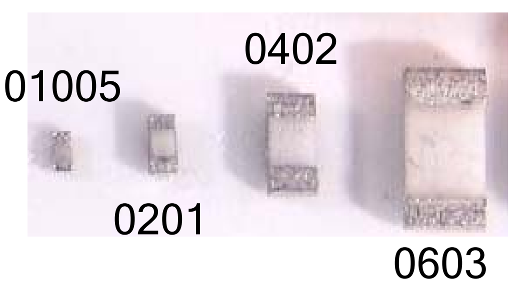
How Small Are They?

01005s mixed with black pepper.





More resembling a metric 0402 at 0.4 x 0.2 mm.



Bottom View

	Nominal (mm / mil)	Tolerance	Lower (mm / mil)	Upper (mm / mil)
L	0.4 / 15.75	+/- 0.02mm	0.38 / 14.96	0.44 / 17.32
W	0.2 / 7.87	+/- 0.02mm	0.18 / 7.09	0.22 / 8.66
T	0.13 / 5.12	+/- 0.03mm	0.10 / 3.94	0.16 / 6.30

Source: Panasonic Thick Film Resistors

Anticipated Challenges

- Printing: Small apertures
  - Stencil clogging, insufficient deposits, deposit size variation, alignment.
- Placement: Small passives
  - Rejects, nozzle contamination, ESD, speed, accuracy.
- Reflow: Tolerance to above imperfections?

Assembly Test Strategy

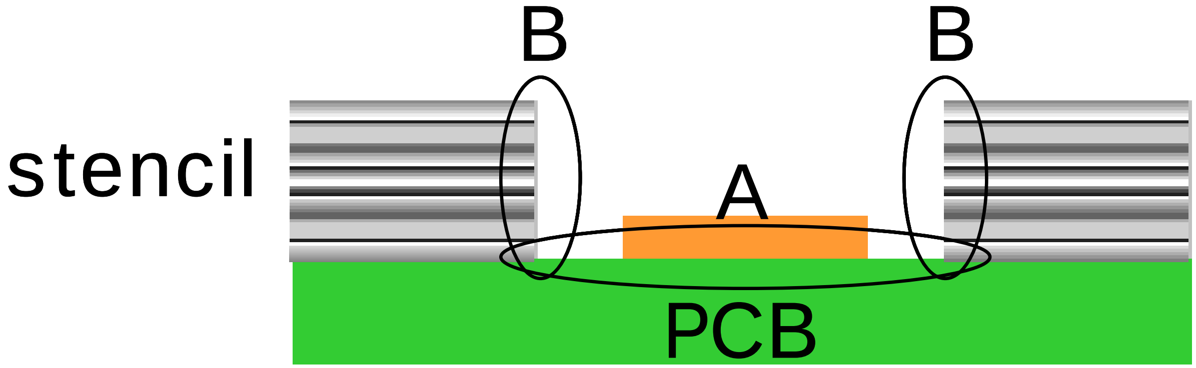
- Printing: Small apertures
  - Stencil clogging, insufficient deposits, deposit size variation, alignment
- Placement: Small passives
  - Rejects, nozzle contamination, ESD, speed, accuracy
- Reflow: Tolerance to above imperfections?

01005 Literature Suggestions	**Benefits**	**Concerns**
3 mil thick stencil	Compliant area ratios.	Paste volume reduced for other components.
Type 4 or 5 solder paste	Improved transfer efficiency.	Material price increase. Paste requalification.
Electroformed stencil	Paste release.	Premium price.

- Generate assembly yield data to compare against published recommendations using:
  - 4 mil thick stencil
  - Type 3 solder paste
  - Laser cut stencil

Area Ratio

- Area Ratio: comparing areas of paste contact inside the aperture = A ÷ B.

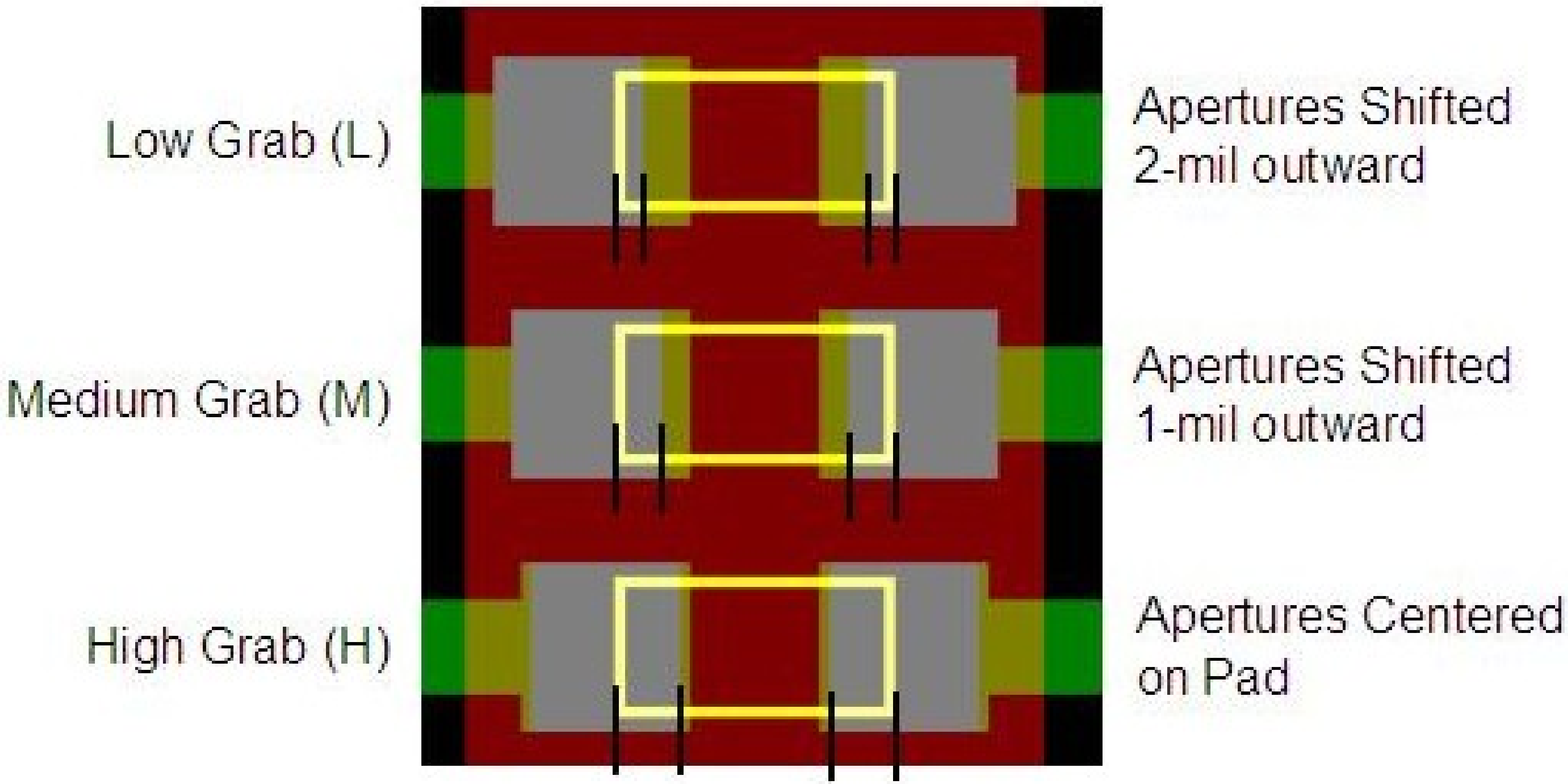


- IPC–7525: Complete solder transfer achieved at Area Ratios > 0.66.

Objectives / Takeaways

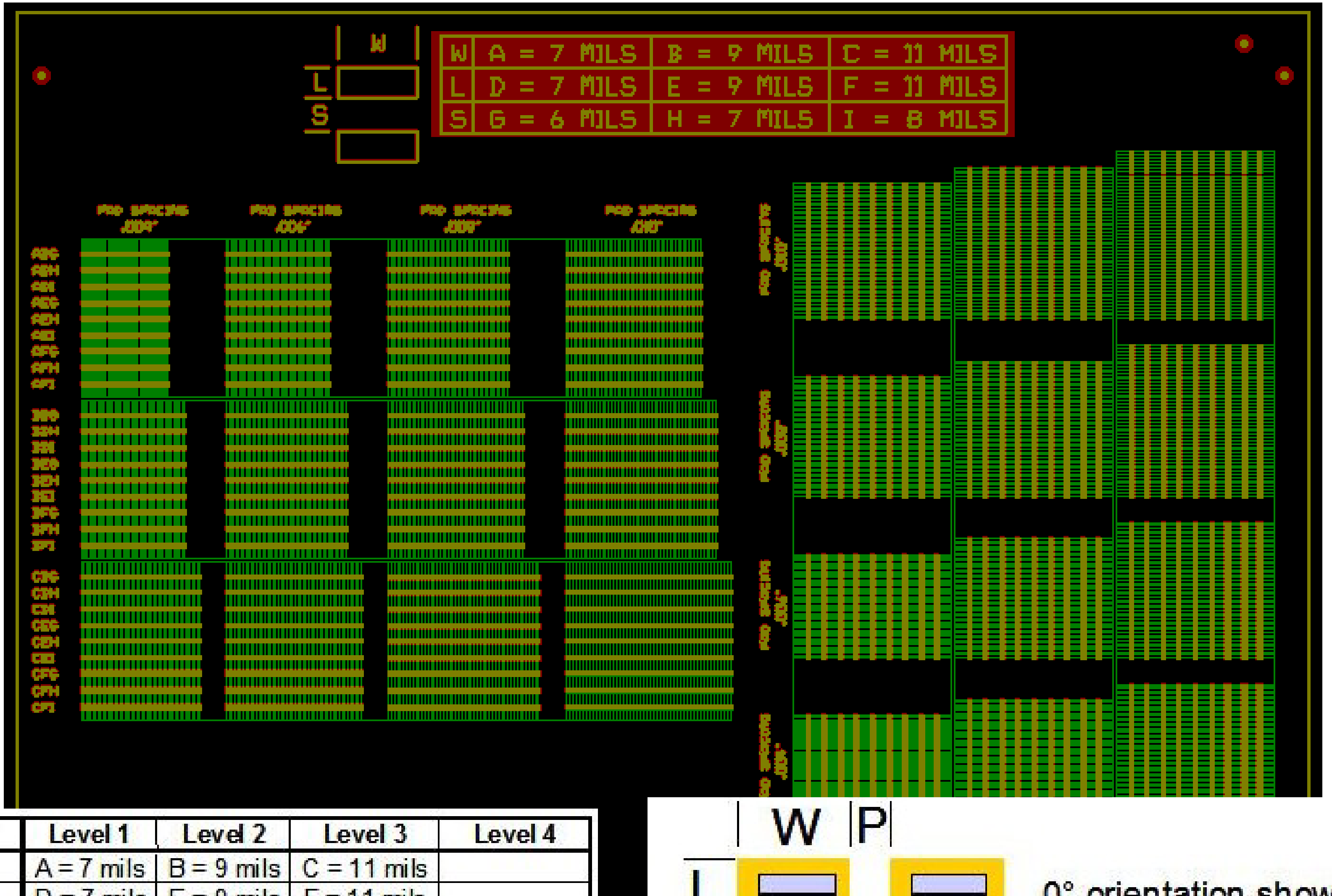
- Recommend best pad / stencil design combination.
- Report assembly yield trends for:
  - Grab (balanced / imbalanced) *See next slide…*
  - Print mis-registration tolerance
  - Reflow atmosphere (N_2 vs. air)
  - 0201 vs. 01005

What is “Grab”?



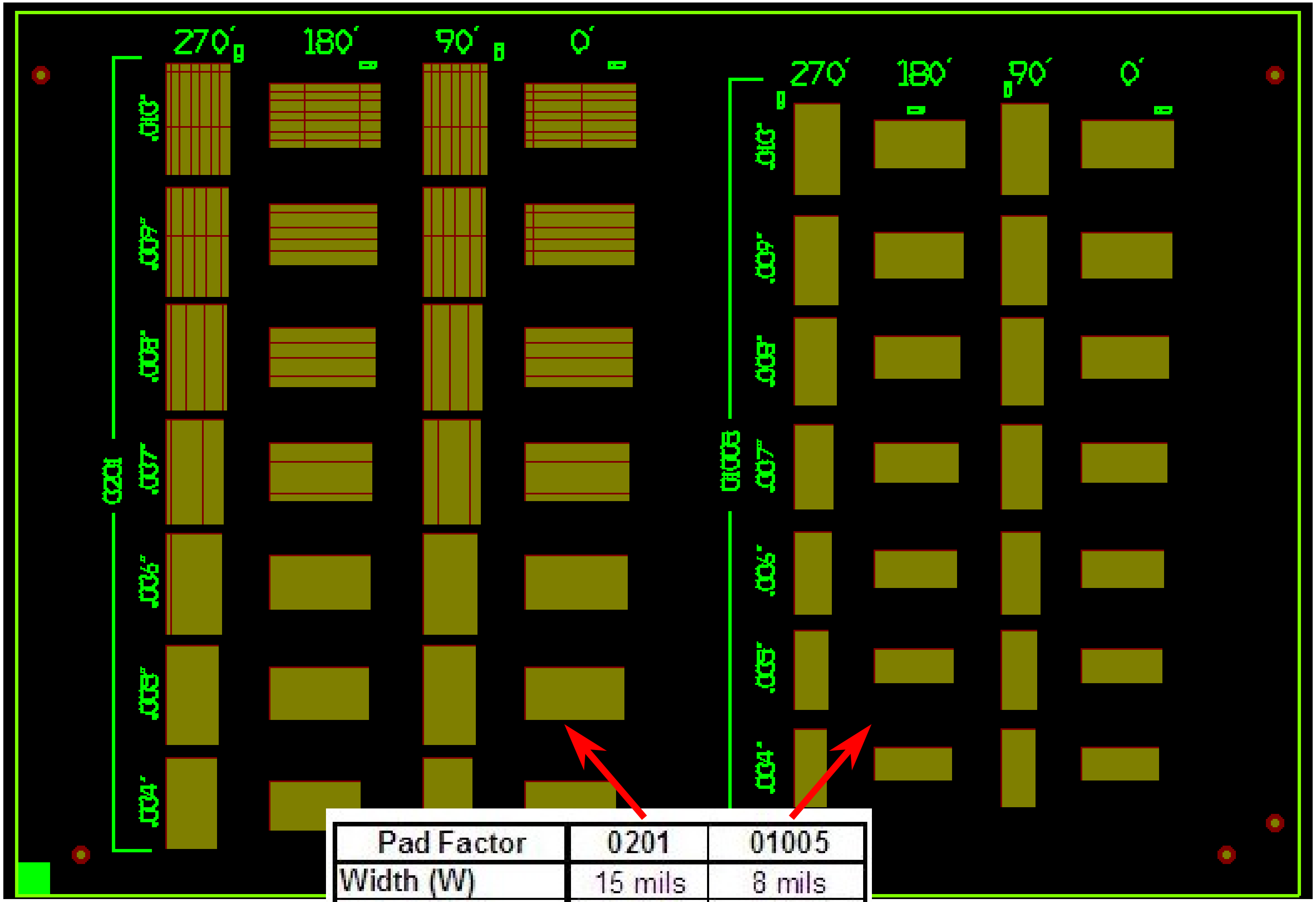
- Grab = overlap of termination on paste.

PCB Design – Side A



Pad Factor	Level 1	Level 2	Level 3	Level 4
Width (W)	A = 7 mils	B = 9 mils	C = 11 mils	
Length (L)	D = 7 mils	E = 9 mils	F = 11 mils	
Separation (S)	G = 6 mils	H = 7 mils	I = 8 mils	
Pad to Pad (P)	4 mils	6 mils	8 mils	10 mils
Orientation (O)	0 deg	90 deg		

PCB Design – Side B



Pad Factor	0201	01005
Width (W)	15 mils	8 mils
Length (L)	12 mils	9 mils
Separation (S)	6, 9 mils	6, 7.5 mils
Pad to Pad (P)	6 mils	6 mils
Orientation (O)	0, 90 deg	0, 90 deg

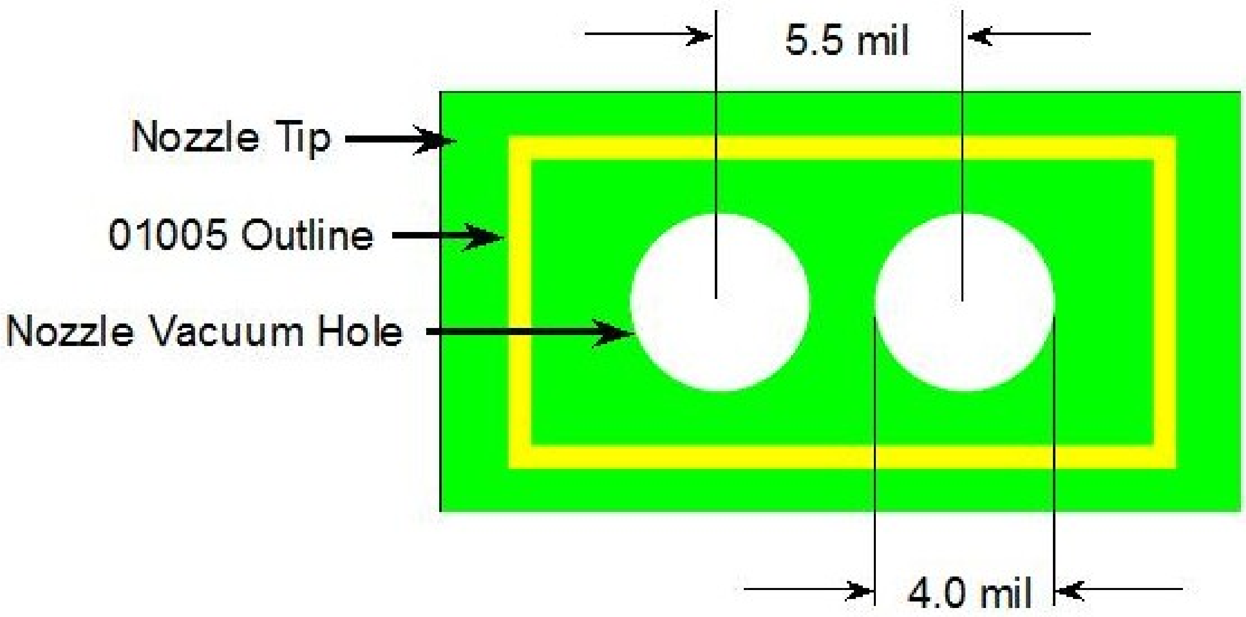
Assembly Equipment

- Fully automatic stencil printer

Parameter	Setting
Print Direction	Rear to Front
Squeegee	Metal, 60 deg, 250mm long
Print Speed	50 mm/sec
Print Pressure	5.0 kg
Separation Speed	20 mm/sec
Print Gap	0 mm
Prior Dummy Prints	3
Stencil Clean	Once before dummy prints

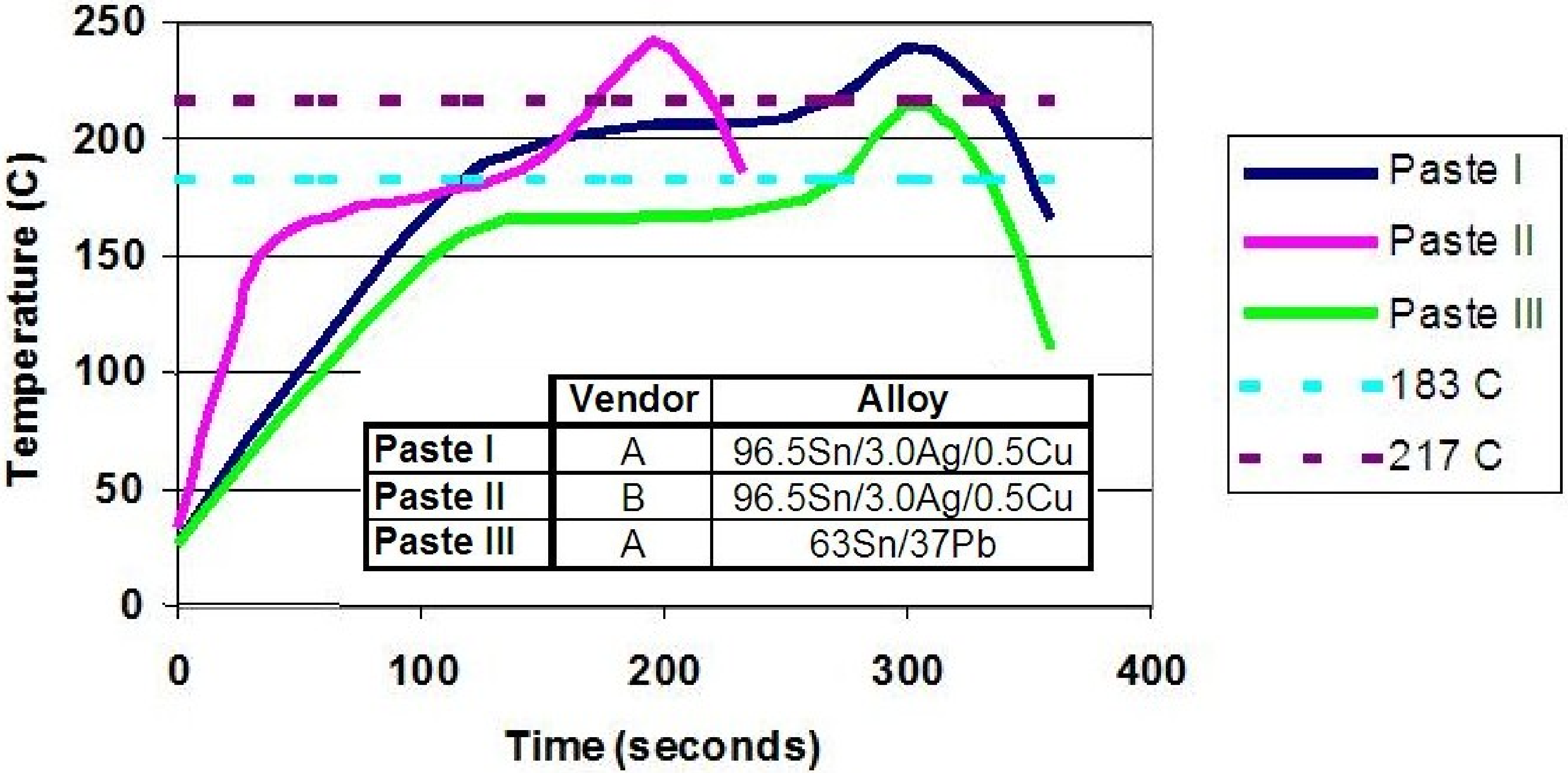
Assembly Equipment (cont.)

- 4 Spindle placement machine



Represents nozzle design actually used, not optimized.

Assembly Equipment (cont.)



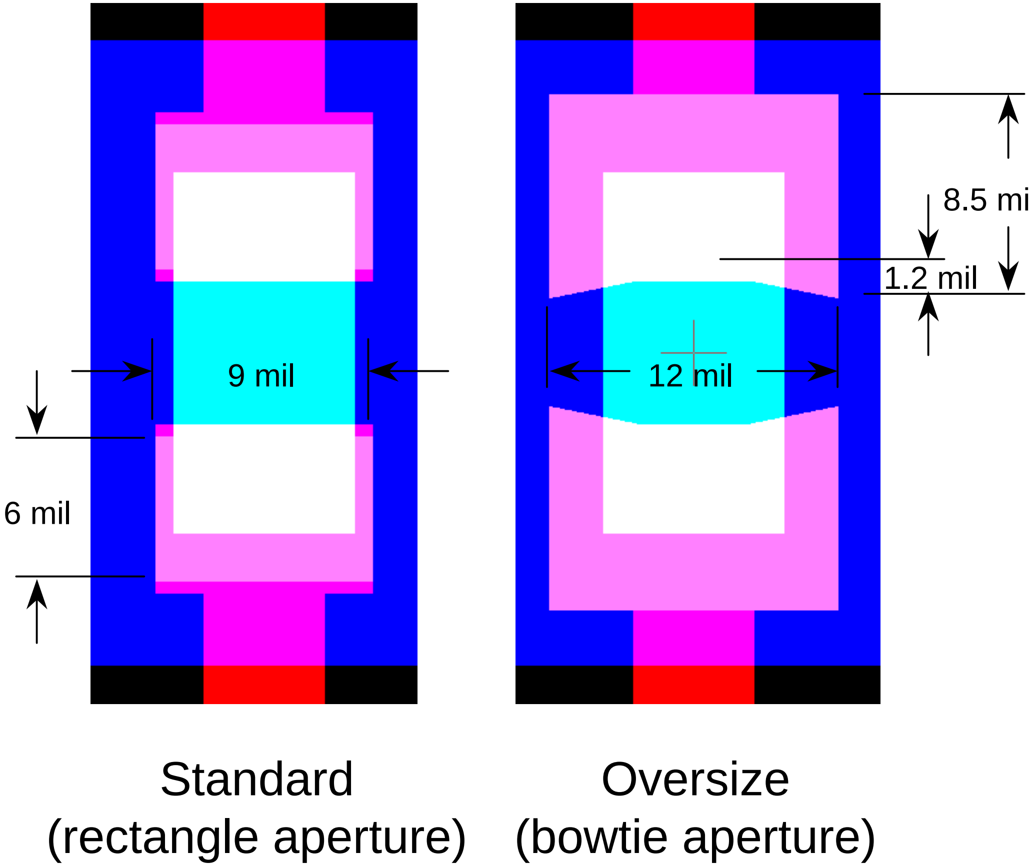
	Vendor	Alloy
Paste I	A	96.5Sn/3.0Ag/0.5Cu
Paste II	B	96.5Sn/3.0Ag/0.5Cu
Paste III	A	63Sn/37Pb

- 8-zone forced convection oven.

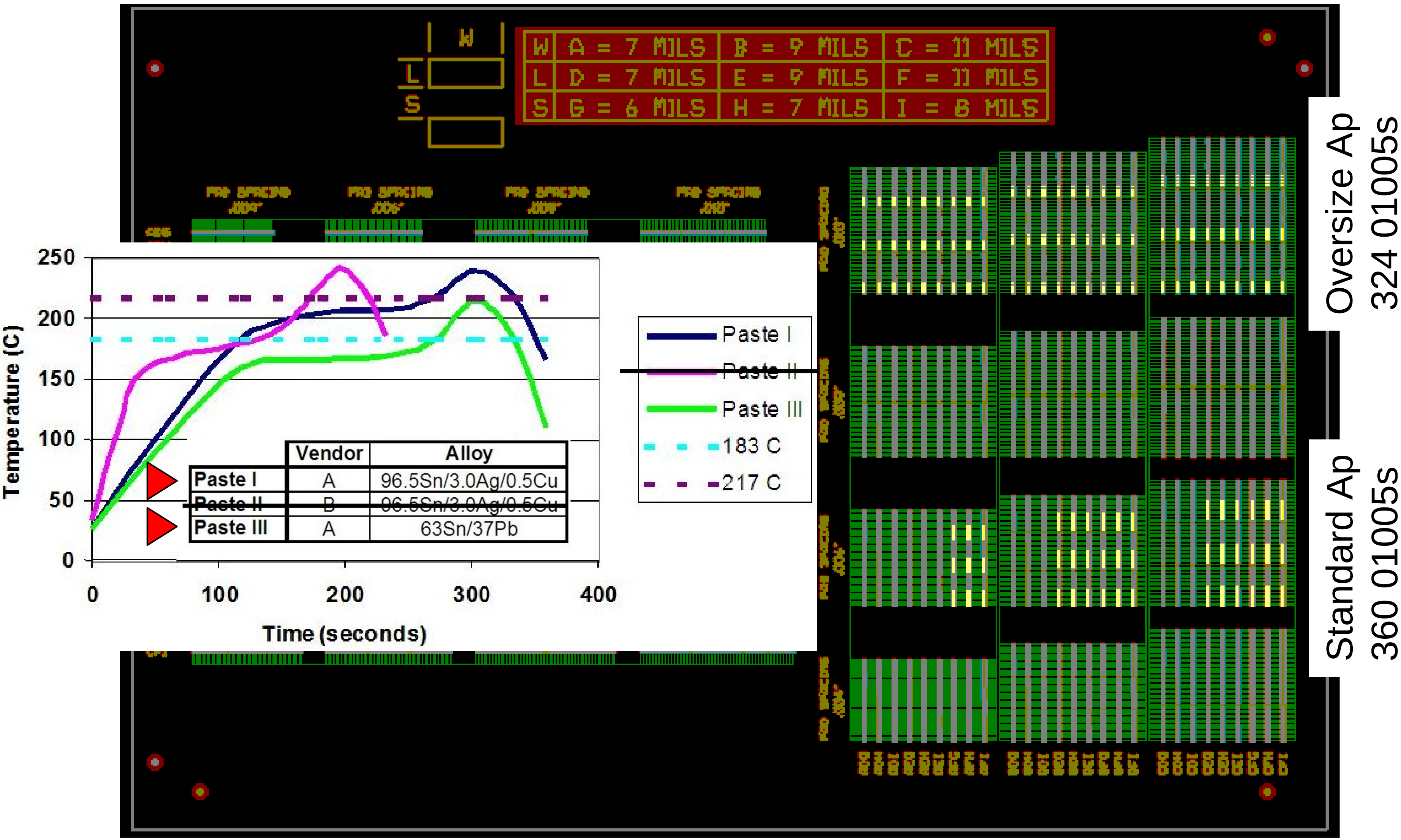
01005 Stencil Apertures

- Standard and overprint aperture designs.

Example: Pad Design BDG

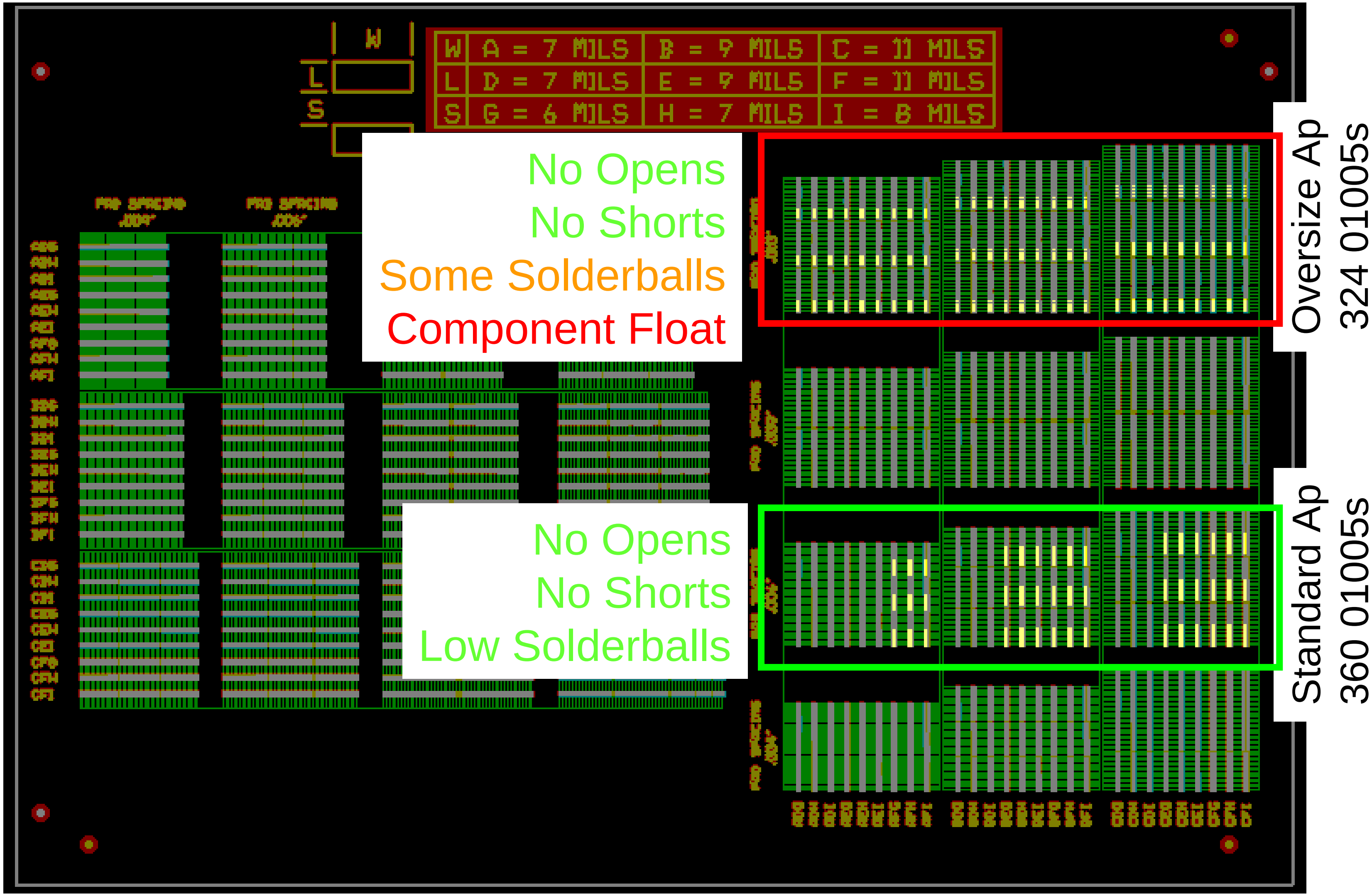


Build 1 & 2 Setup



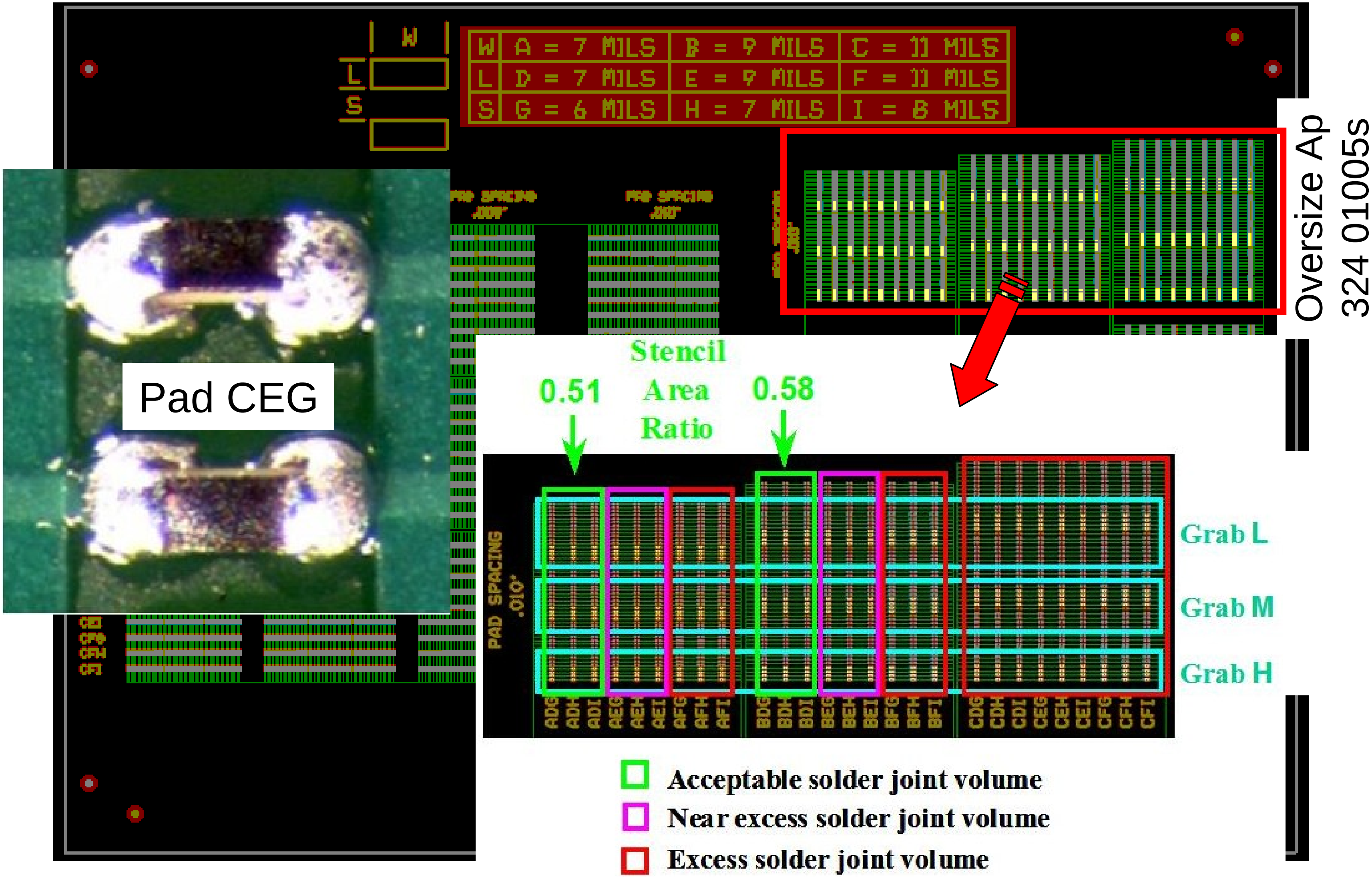
- AR > 0.5, 3 grabs, N_2 reflow.

Build 1 & 2 Result



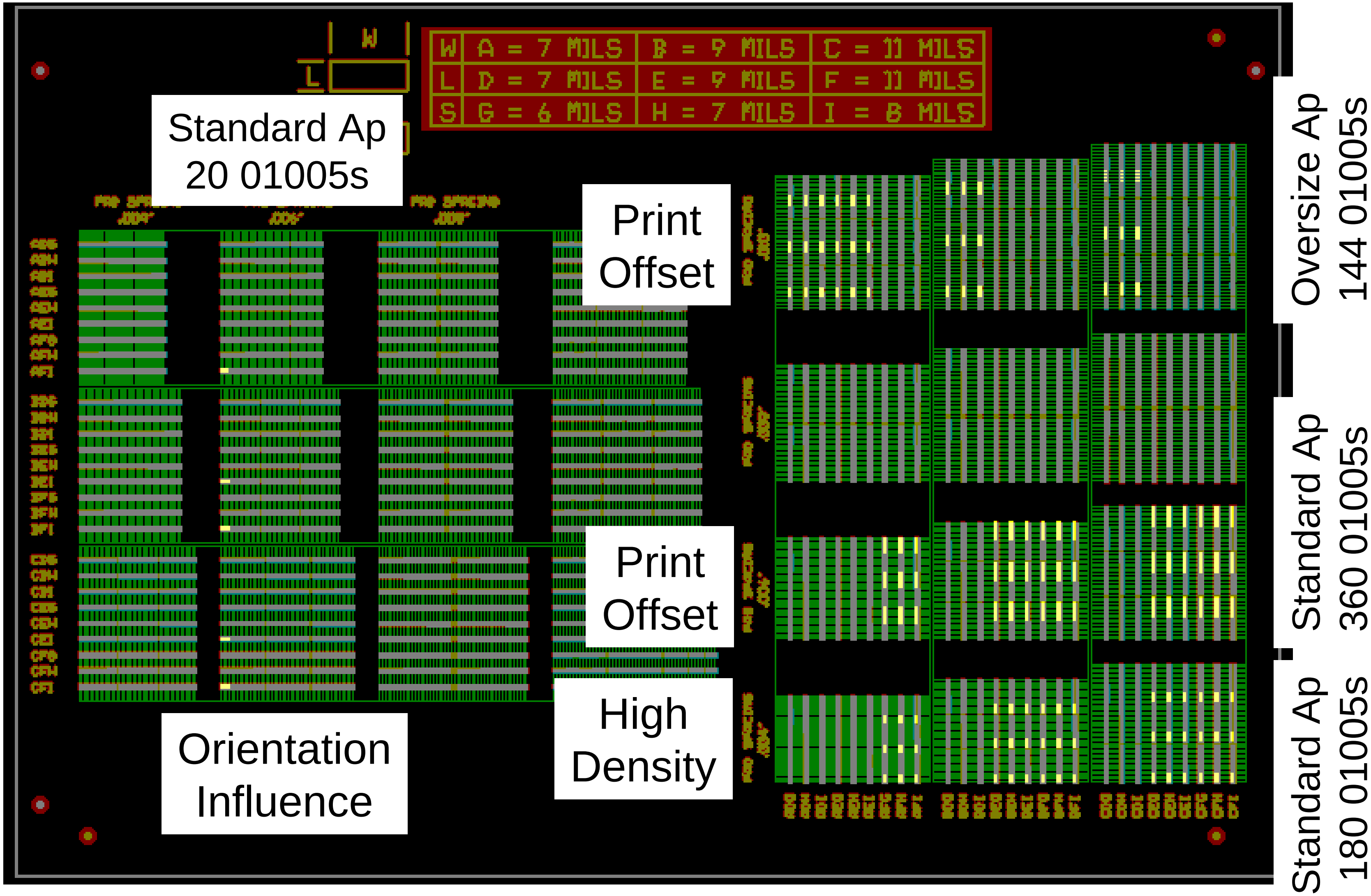
- Sn/Pb wets better, fewer “floats”.

Build 1 & 2 Result



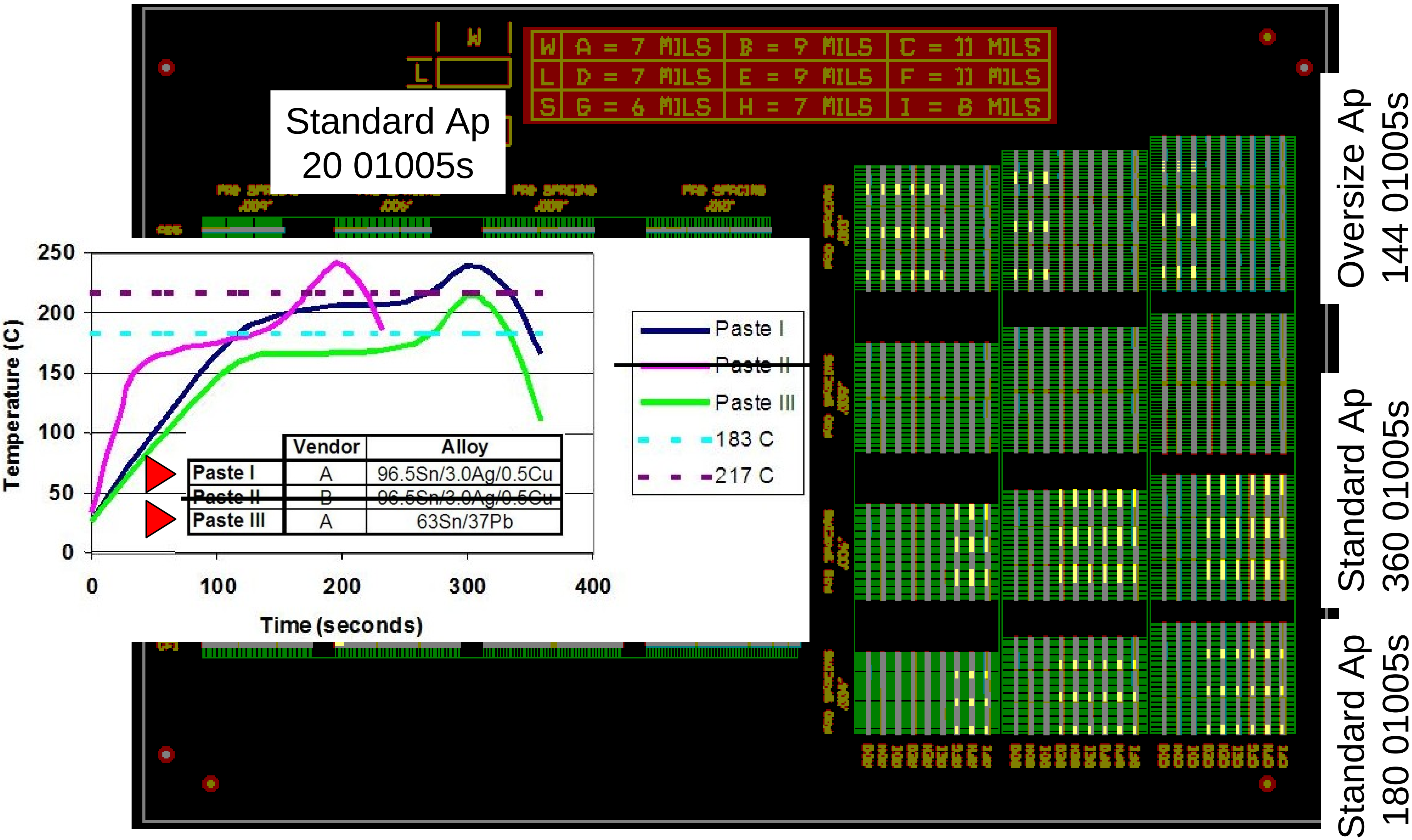
- Sn/Pb wets better, fewer “floats”.

Build 3 & 4 Setup



- AR > 0.5, 3 grabs (90° only), N_2 reflow.

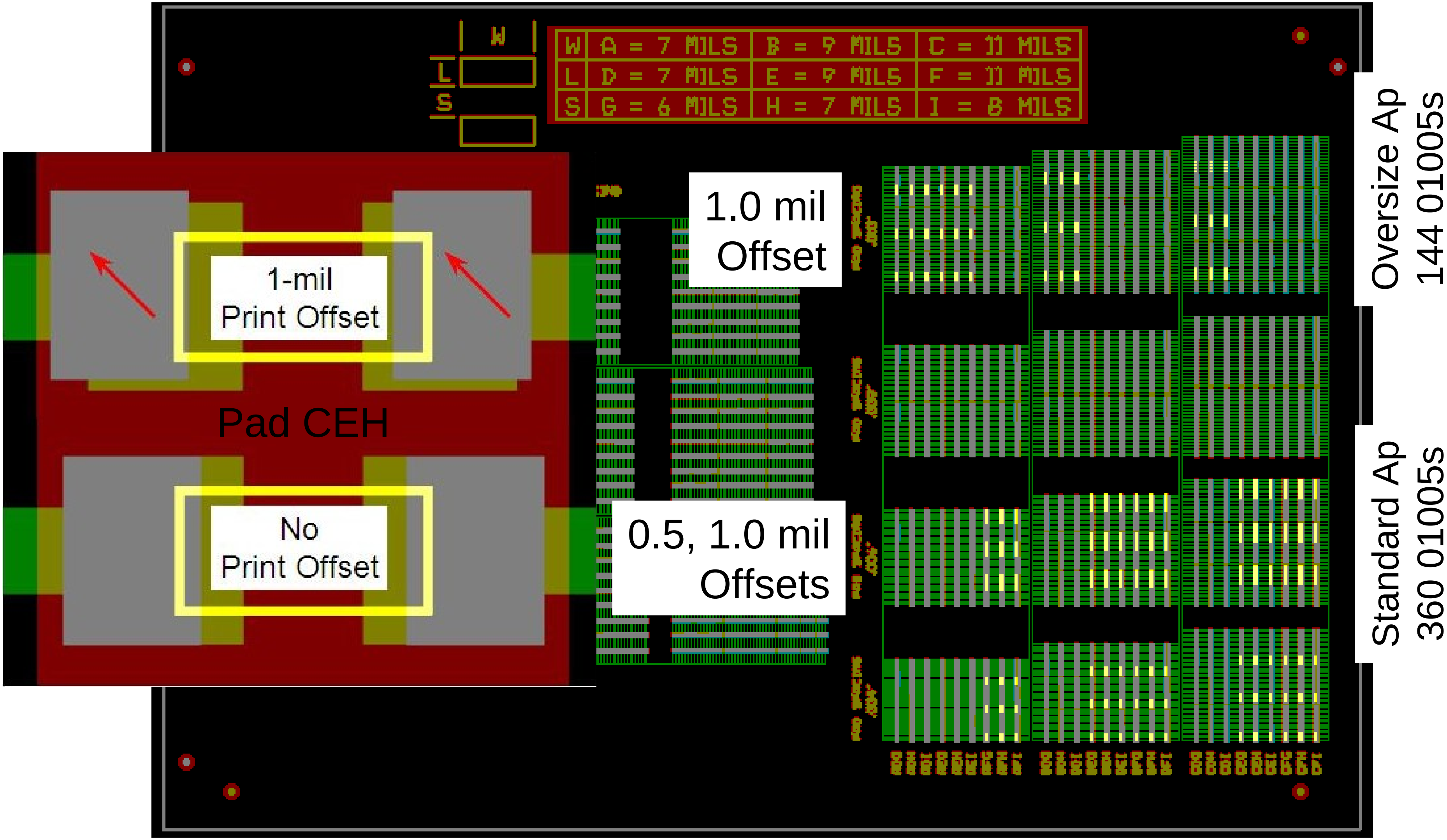
Build 3 & 4 Setup



	Vendor	Alloy
Paste I	A	96.5Sn/3.0Ag/0.5Cu
~~Paste II~~	~~B~~	~~96.5Sn/3.0Ag/0.5Cu~~
Paste III	A	63Sn/37Pb

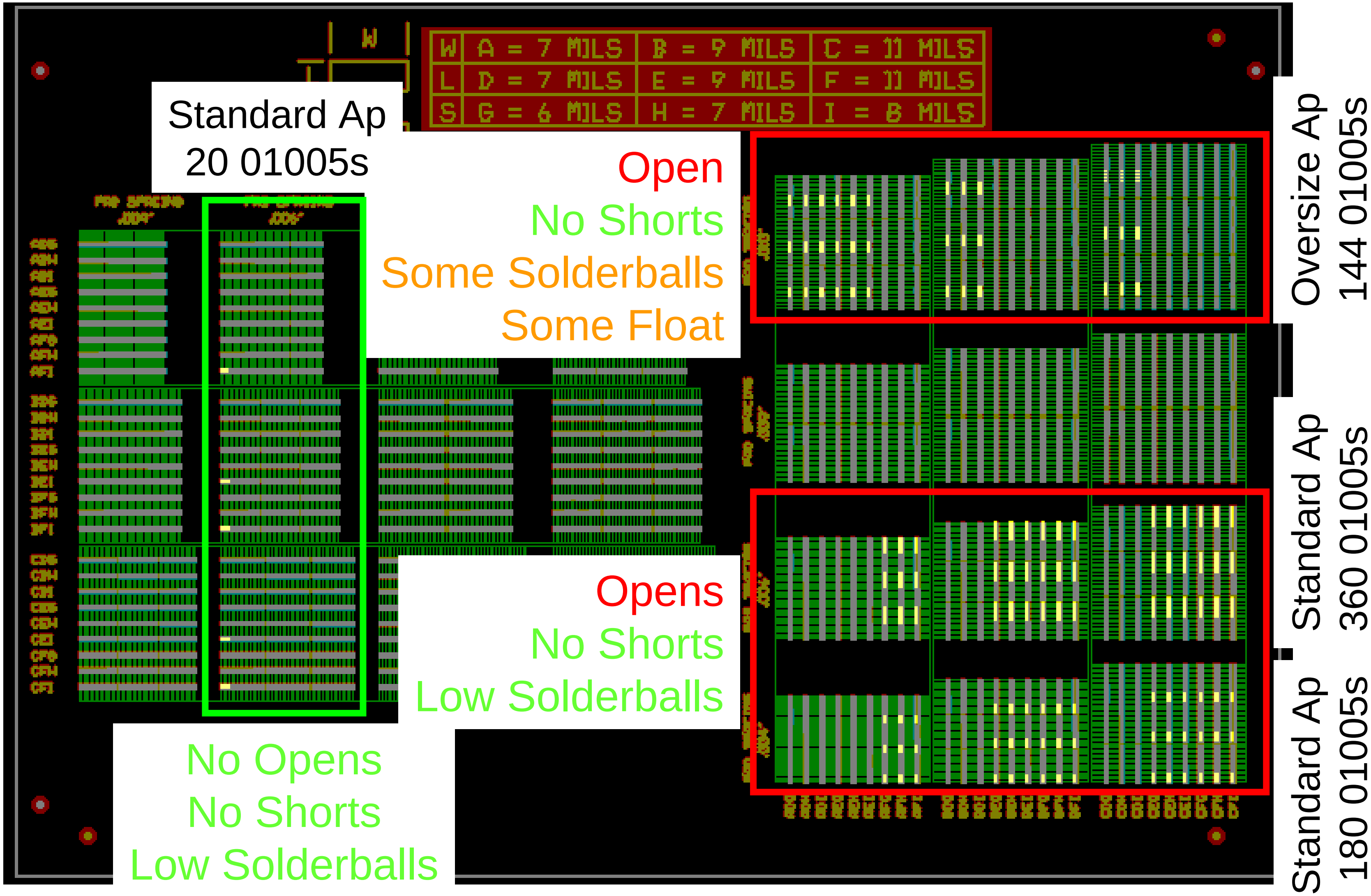
- AR > 0.5, 3 grabs (90° only), N_2 reflow.

Build 3 & 4 Setup (cont.)



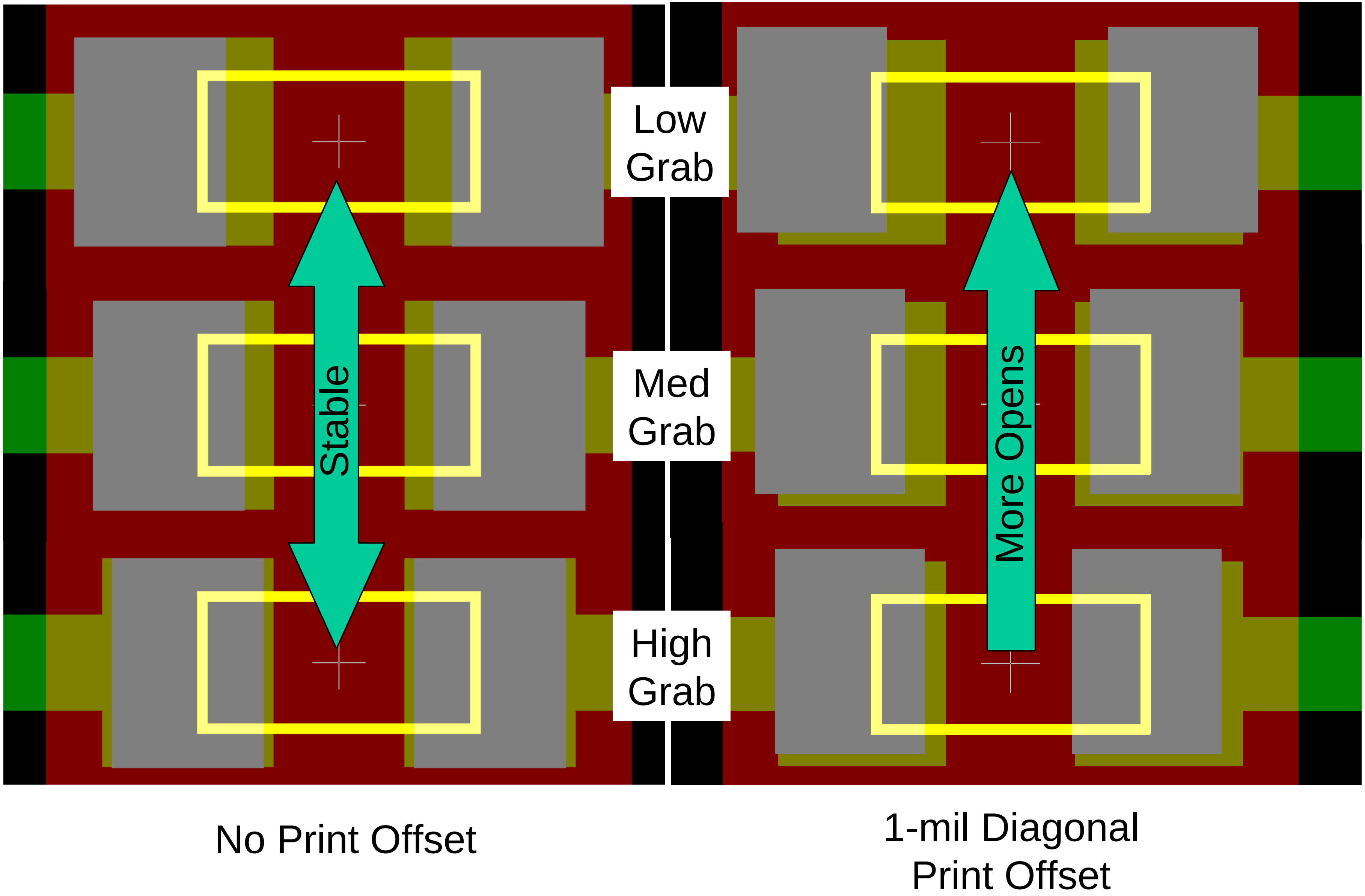
- Print mis-registration test areas.

Build 3 & 4 Result

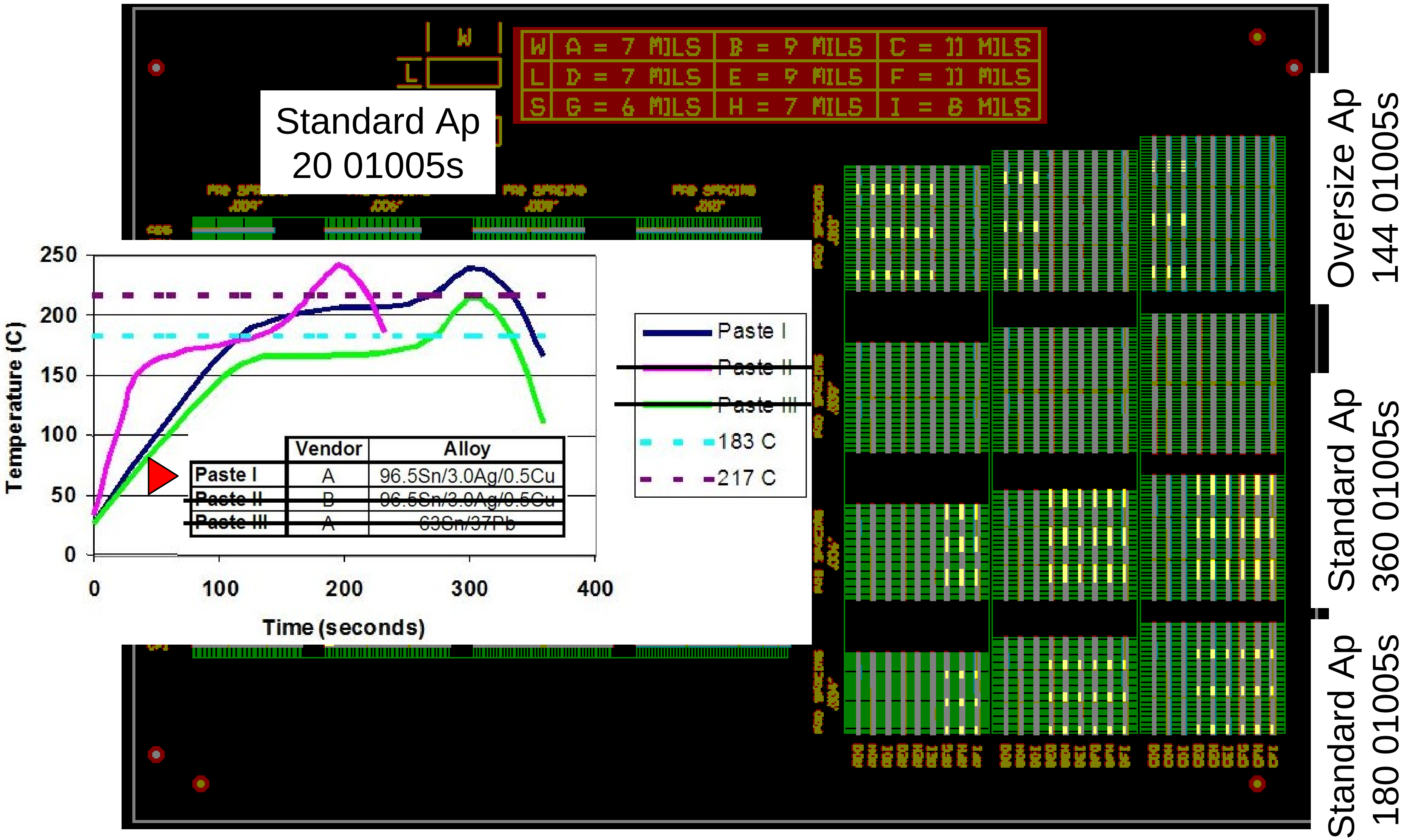


- Opens observed for Sn/Pb only (Build 3).

Build 3 Grab / Print Misregistration Effects

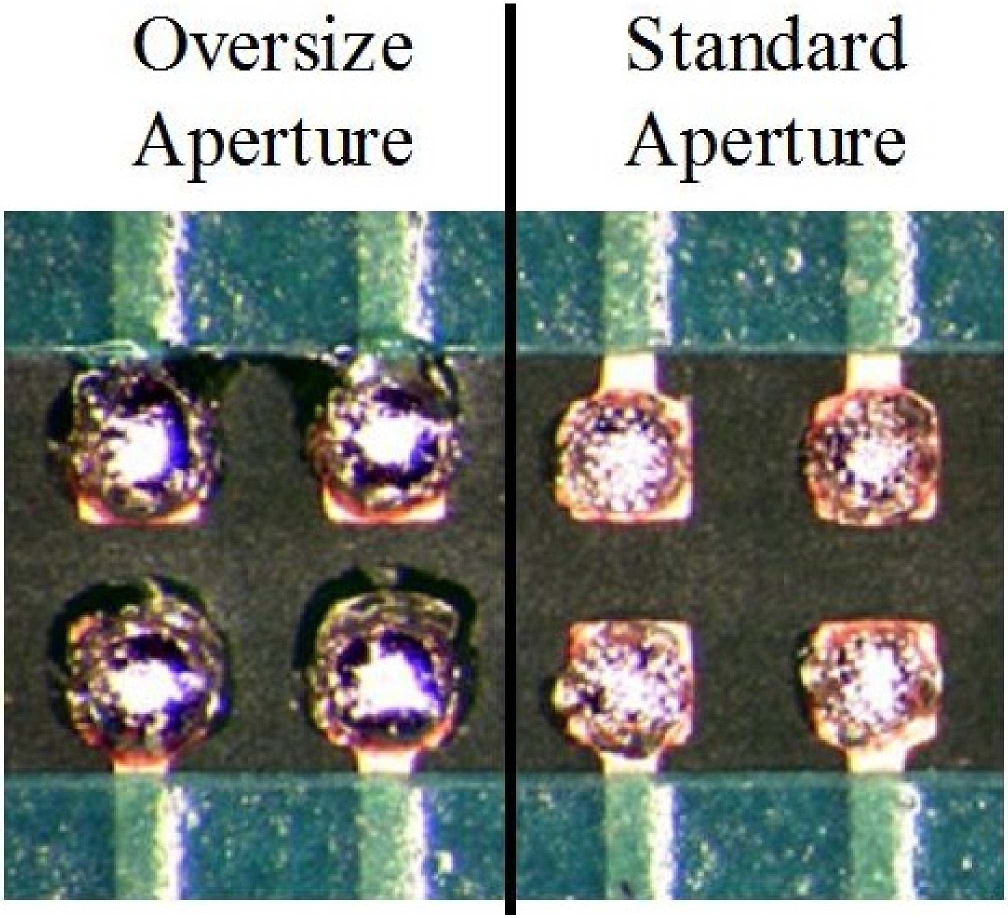


Build 5 Setup (same locations as previous)



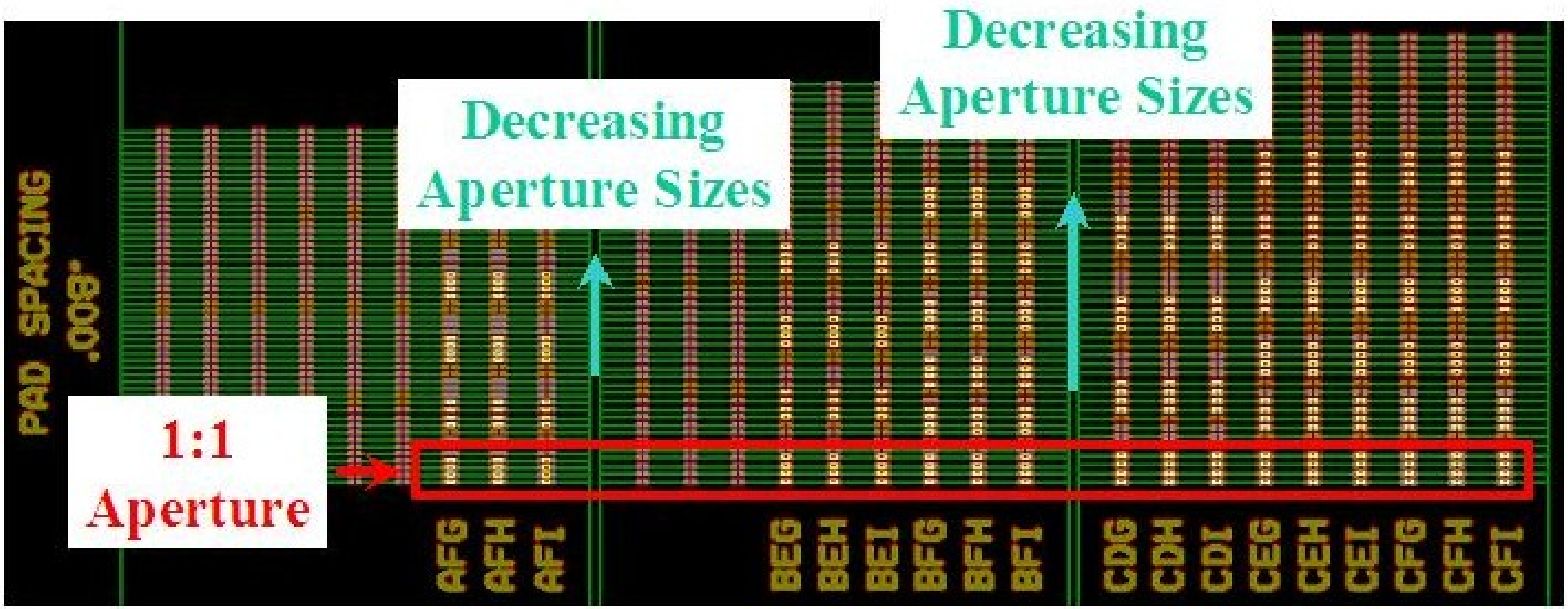
- AR > 0.5, 3 grabs (90° only), <u>Air</u> reflow.

Build 5 Result

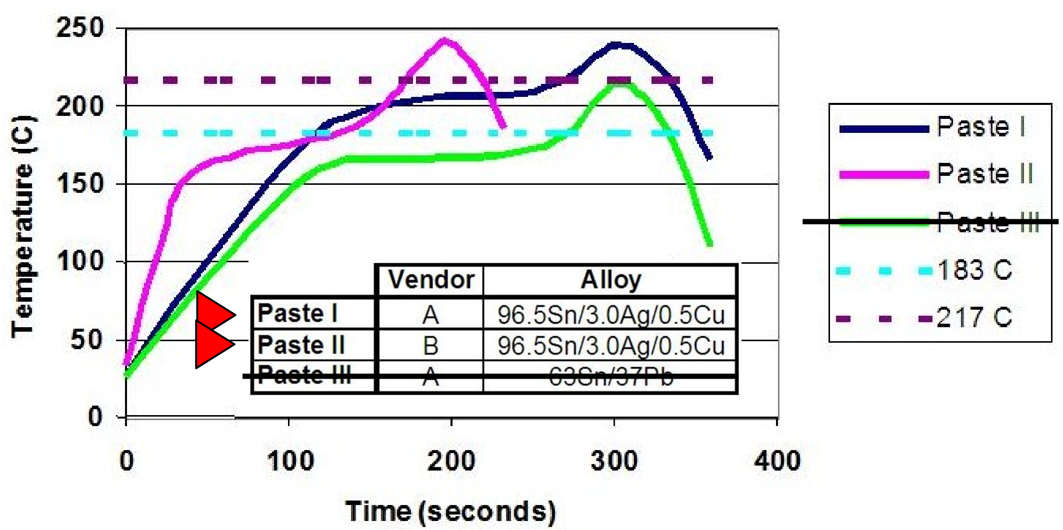


- Incomplete solder coalescence prevalent.
- Inconclusive defect trend.

Build 6, 7, 9, 10 Setup

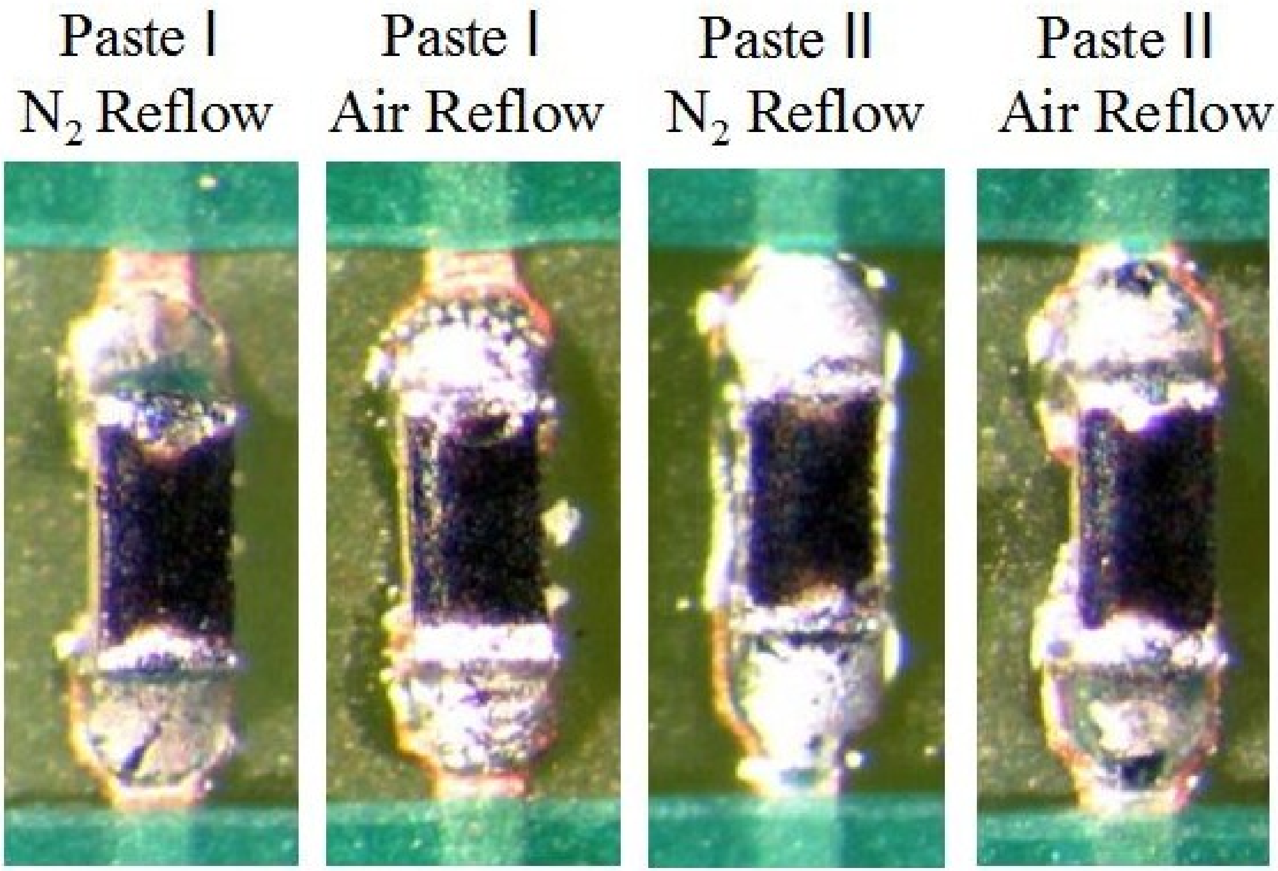


- Assembled on some area ratios < 0.5.
- All apertures centered on pads.



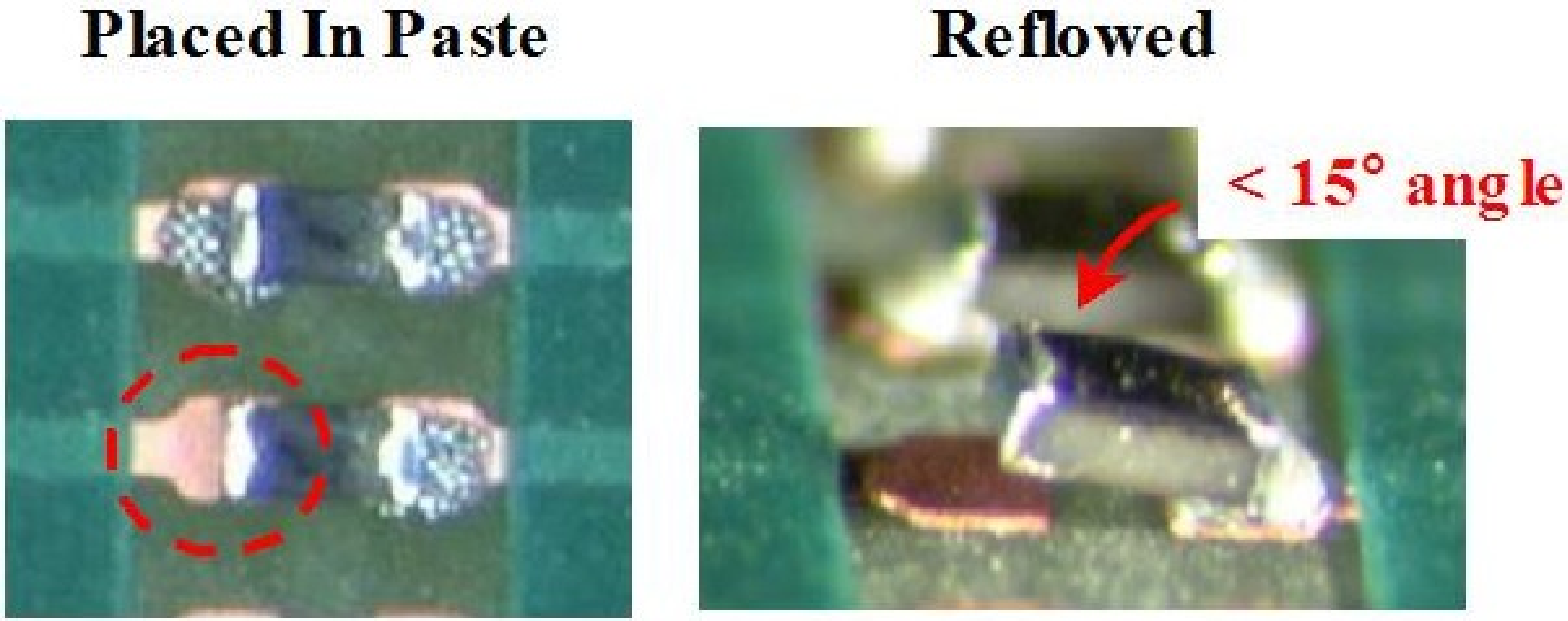
Build 6, 7, 9, 10 Result

SAC305
Paste Only



- Paste II better in air, due to profile or chemistry?

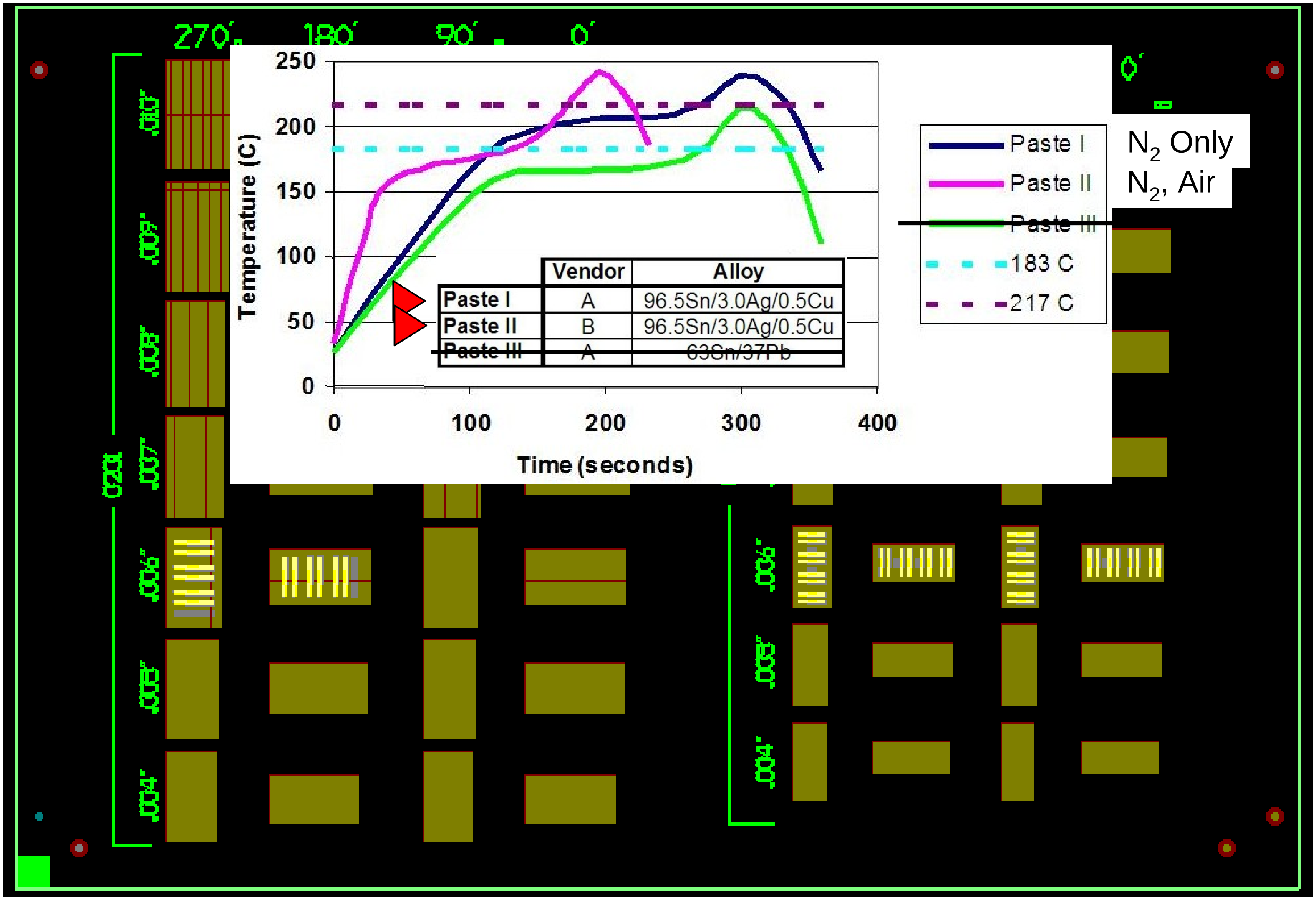
Build 6, 7, 9, 10 Result (cont.)



SAC305 Paste I, N_2 reflow.

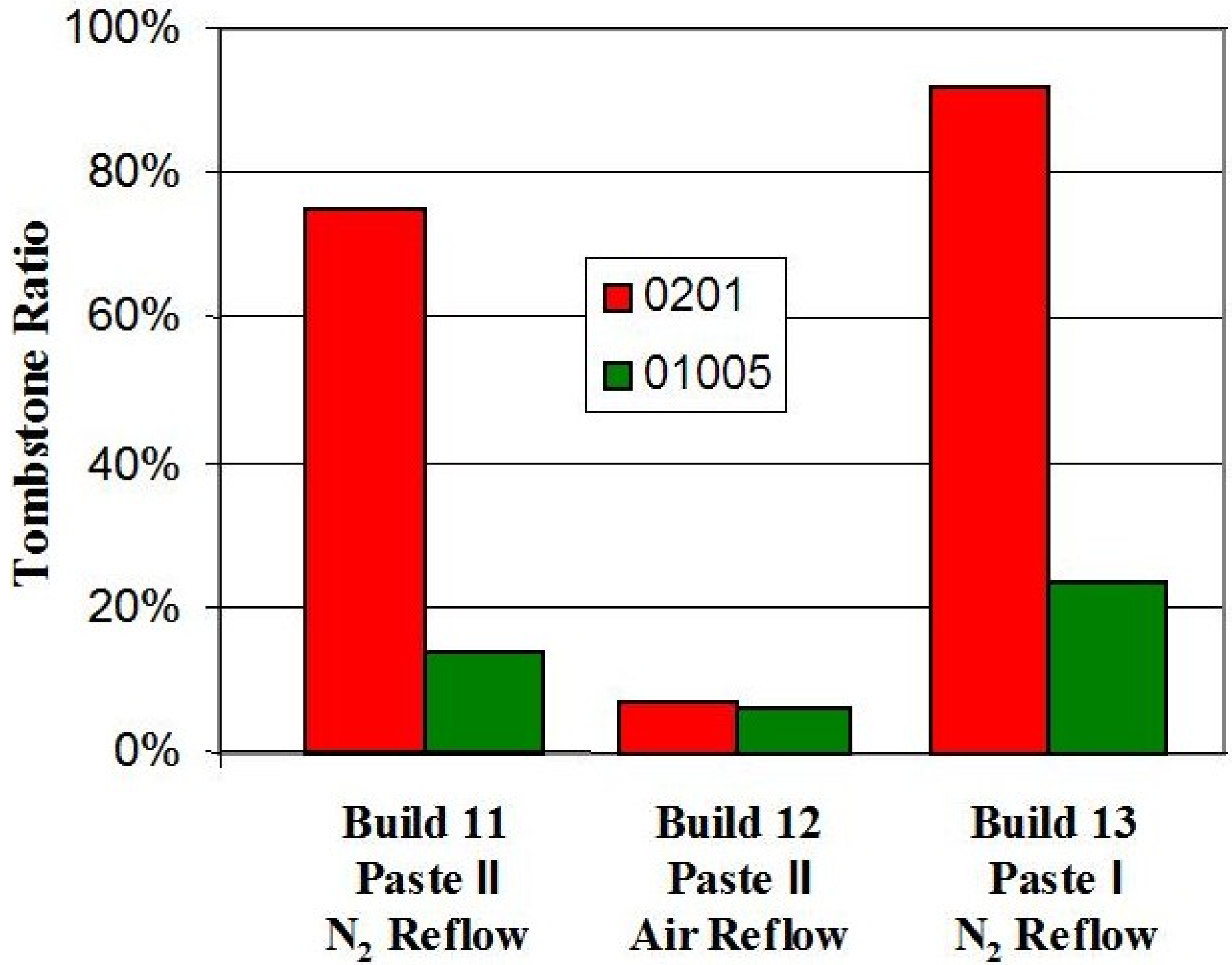
- When both pads received solder, no defects observed.
- Small apertures clogged, imbalance condition.
- Evidence of tombstone resistance.
- Do 0201s behave the same?

Build 11, 12, 13 Setup



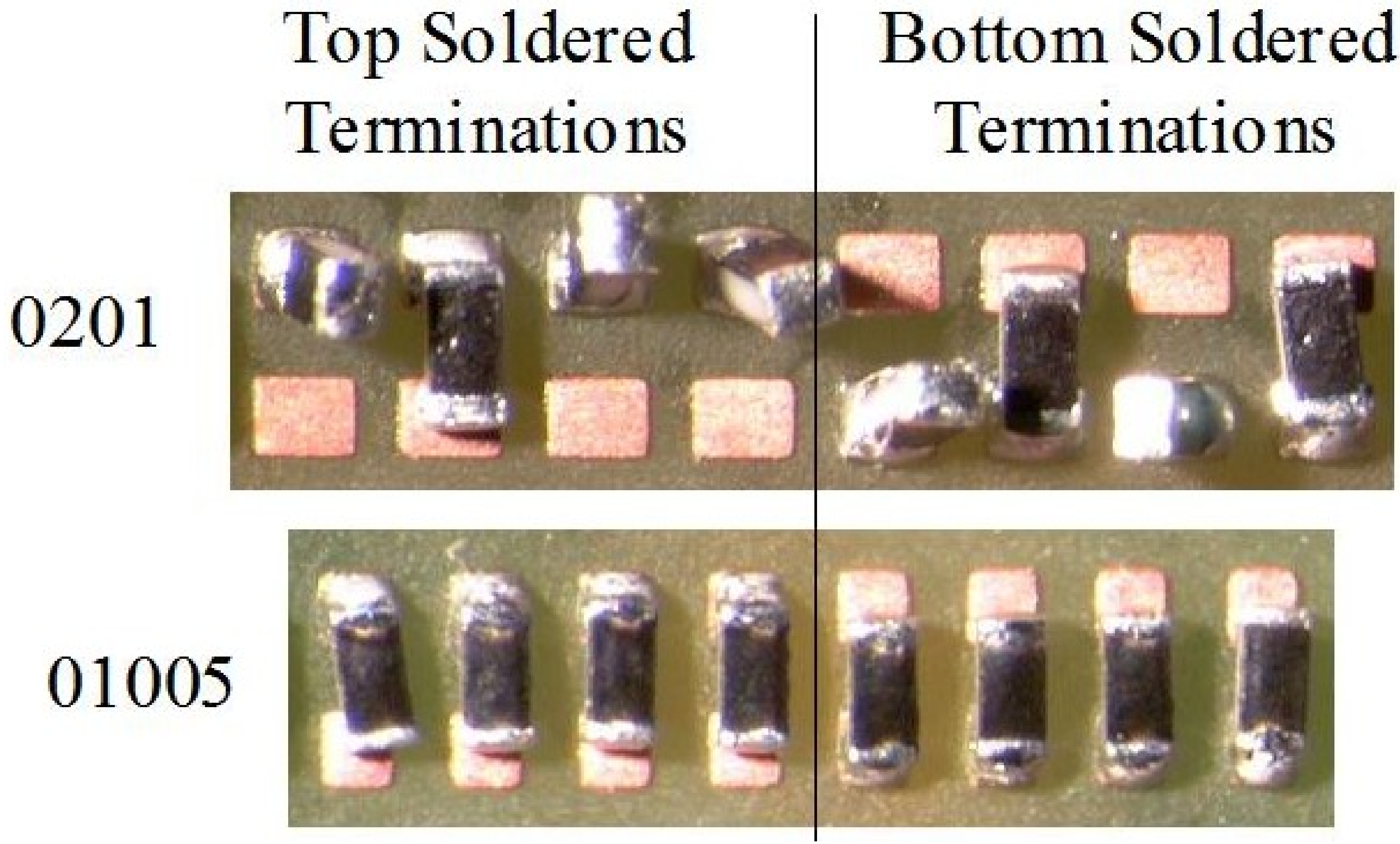
- Print one termination attachment pad only.

Build 11, 12, 13 Result



- N_2 reflow more significant on <u>0201</u> defects.

Build 11, 12, 13 Result (cont.)



- N_2 reflow comparison.

Key Points

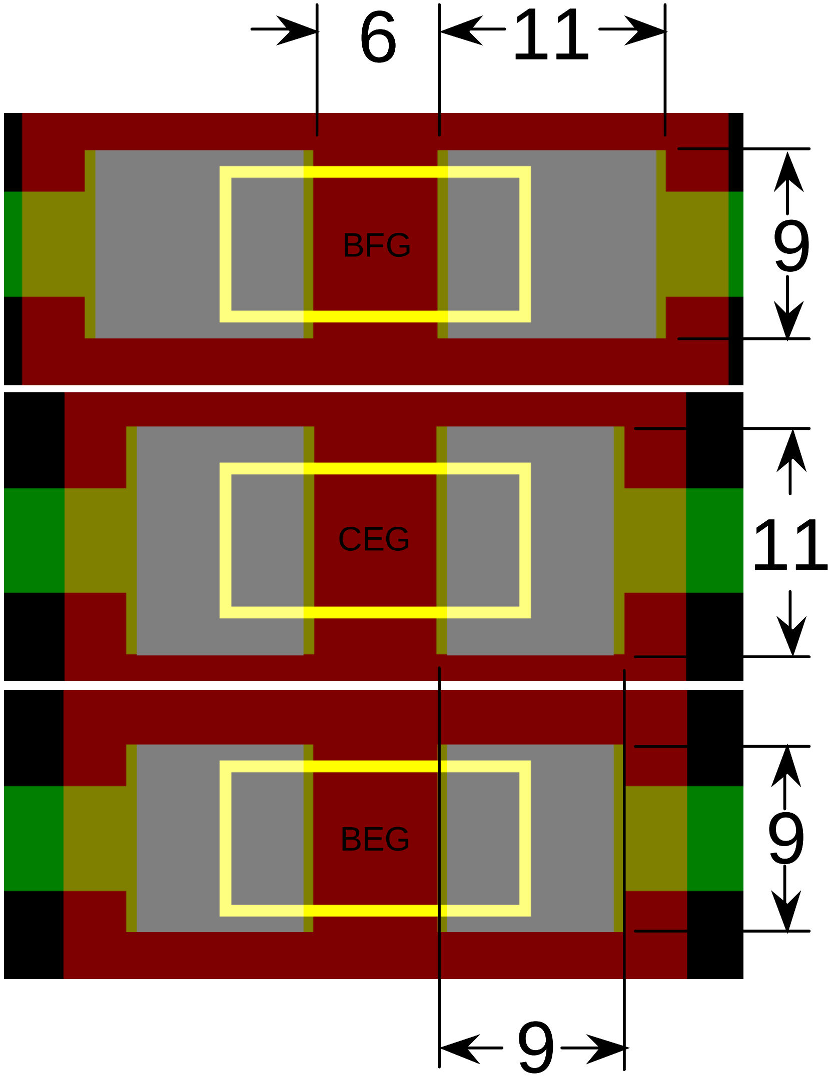
- 4-mil thick stencil / Type 3 SAC305 worked better than expected.
- Low area ratio apertures give enough solder, but process should be monitored.
- Print = 01005 thickness: Nozzle hygiene.
- Profile optimization required for air reflow.
- 01005 less tombstone sensitive than 0201.
- Reflow orientation appears not critical.
- Fewer defects with SAC305 compared to Sn/Pb.
- Low “grab” designs require greater print/place accuracy.
- Results for Cu/OSP pads only!
- 01005 <u>capacitors</u> could yield differently!

Pad / Stencil Design Recommendation

Choice 1
Pad BFG

Choice 2
Pad CEG

Choice 3
Pad BEG



- “Standard” apertures centered on pad.

Thanks for attending