# DFM Rules for BGA's

Presented By: Dale Lee
E-mail: Dale.Lee@Plexus.Com

August 2012

PLEXUS
The Product Realization Company

# DFM Guidelines for BGA's and LGA's

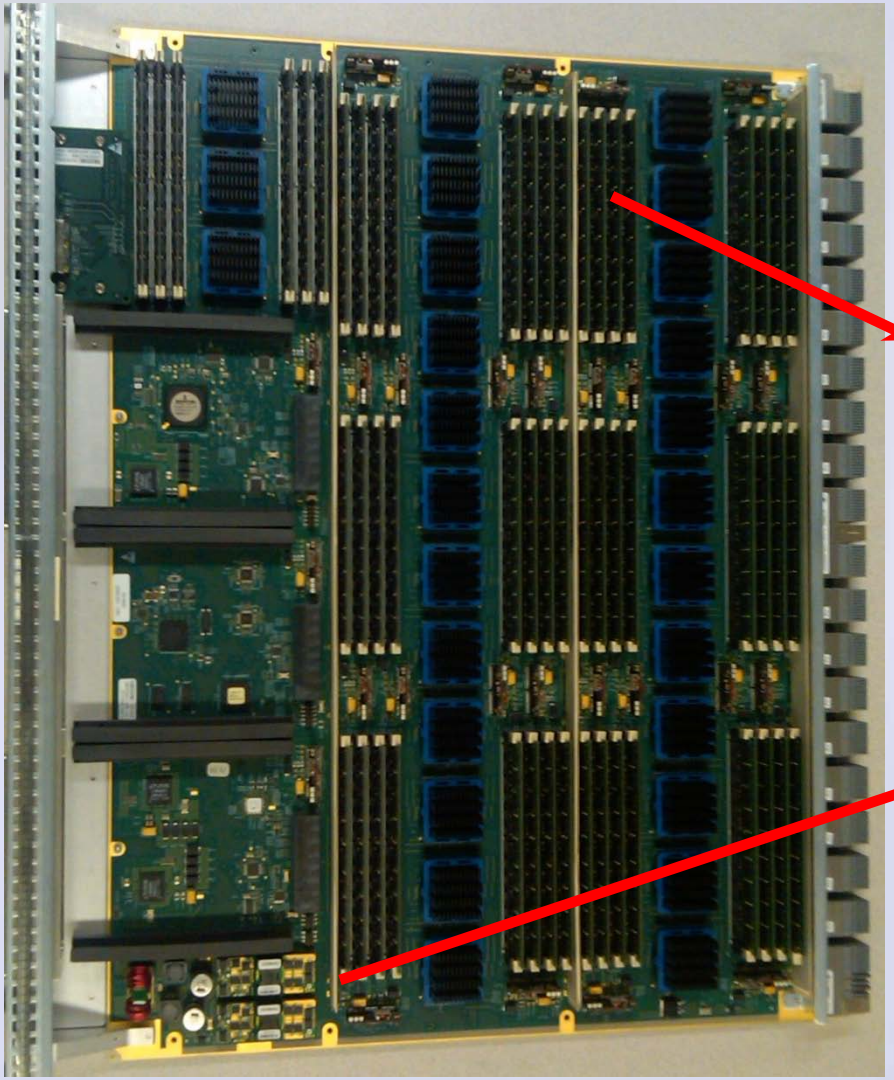
Presented By: Dale Lee
E-mail: Dale.Lee@Plexus.Com

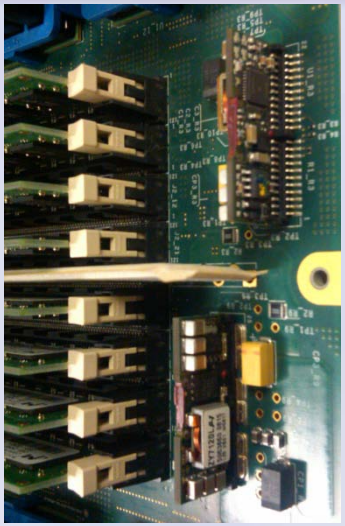
August 2012

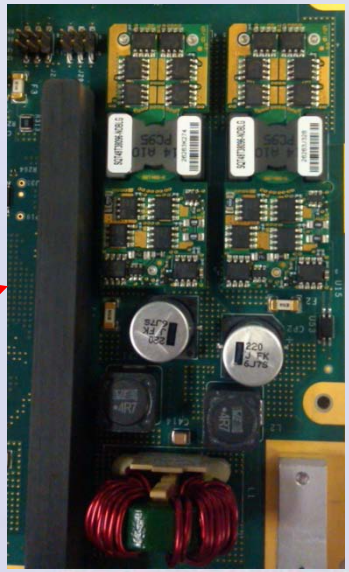
PLEXUS
The Product Realization Company

# Today's Electronics

- High Layer Counts
- Wide Range Of Component Package Sizes
- Soldered Other Soldered Assemblies
- Mixed SMT & PTH Technology
- Increased Component / Interconnection Density
- Higher Number Of Components (10,000 Plus)

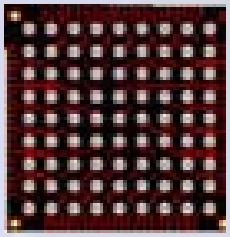






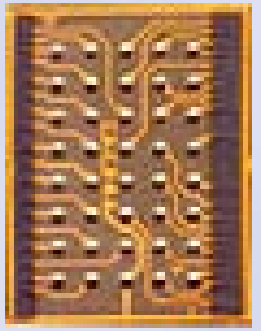
PLEXUS
The Product Realization Company

# Area Array Packages



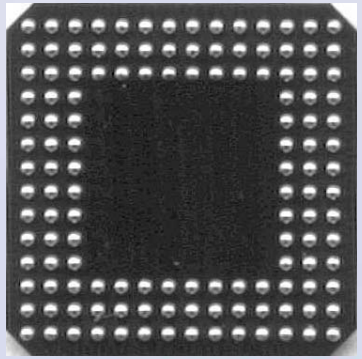
0.8mm Flex CSP

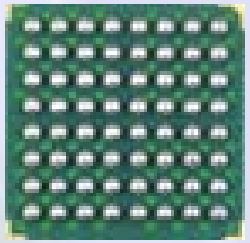
0.5mm u-SMD



0.75mm Flex CSP

1.0mm Flex CSP



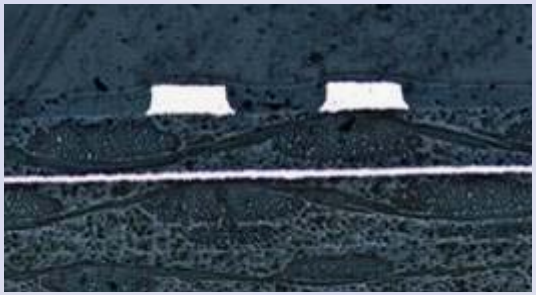


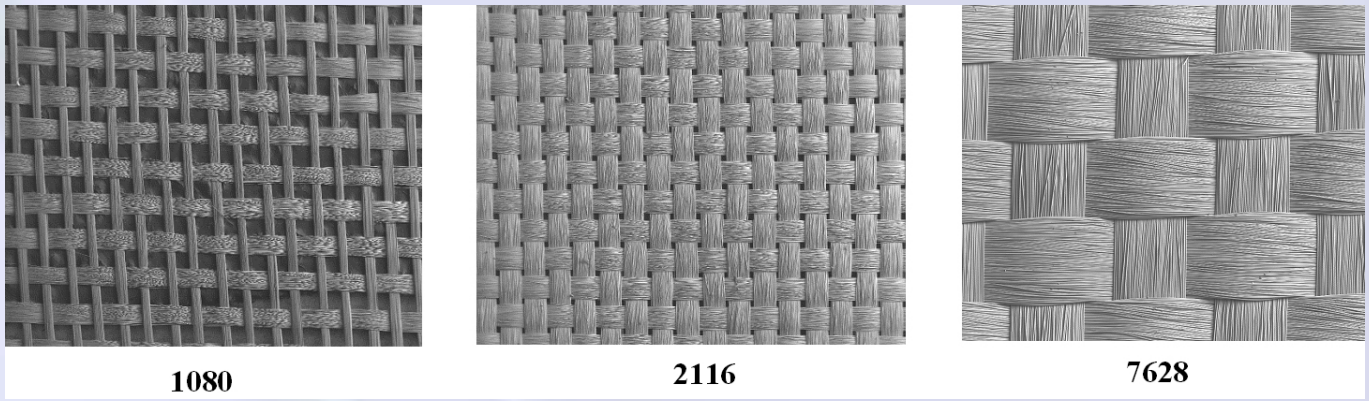
1.0mm Laminate CSP

PLEXUS
The Product Realization Company

# PCB Laminate Construction

- Laminate Made From Mix of Resin & Glass
- Most Common Types: 7628HR, 7628,2116, 2113,1080,1080LD,106
- Location of Trace to Glass (Dk~6) Bundle or Resin (Dk~3.5) Pocket Can Impact Electrical (Impedance) And Mechanical Performance

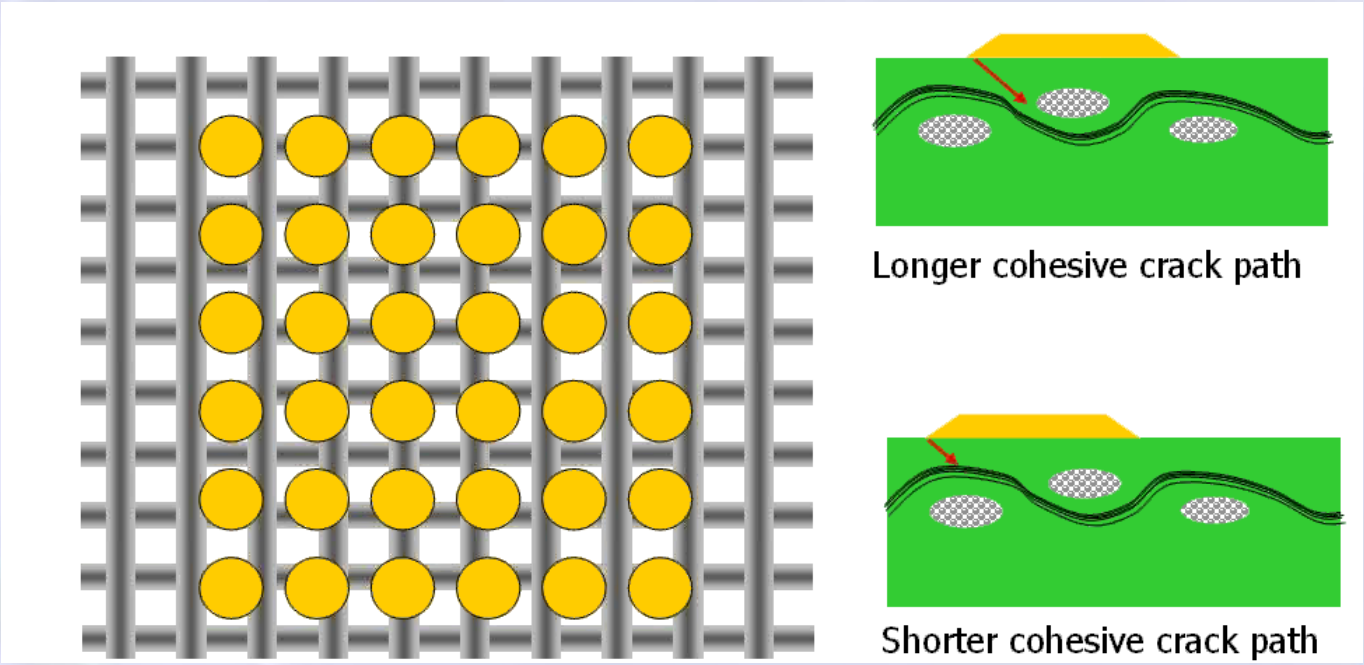





PLEXUS
The Product Realization Company

# Pad To Glass Bundle Interaction

- Location of Pad Relative to Glass Has An Impact On:
  - Crack Location & Formation
  - Functional Performance
- Size And Shape Of Pad Also Important Factor




Courtesy of Universal Instruments

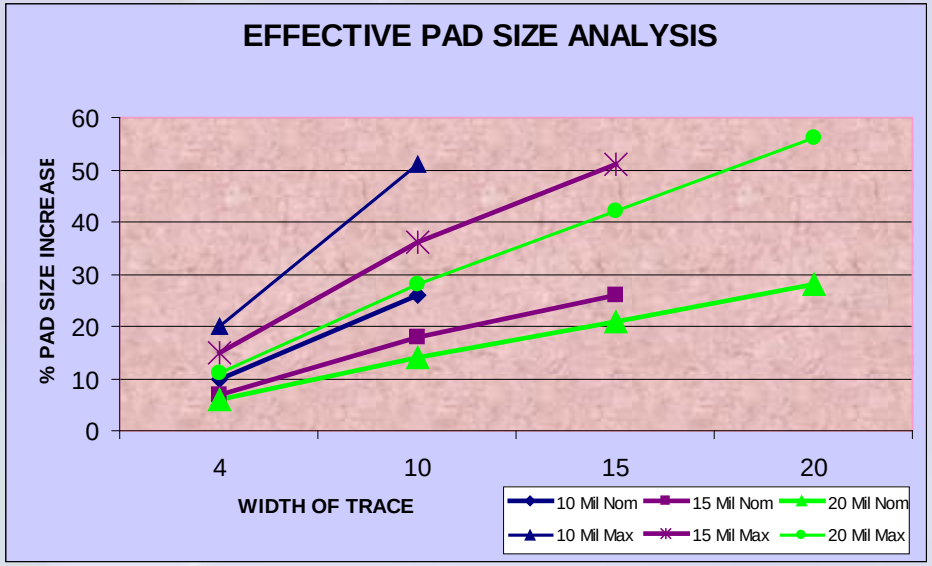
PLEXUS
The Product Realization Company

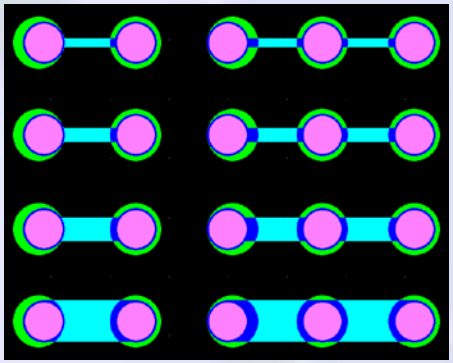
# SMT Component Land Pattern

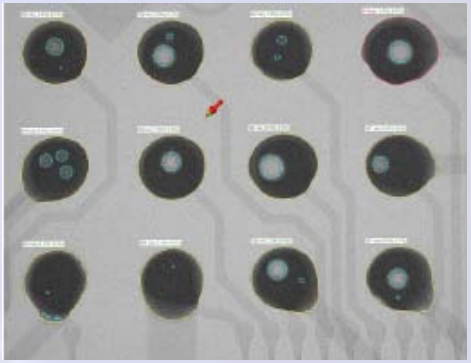
When developing a new land pattern design, utilize pad design recommendations from suppliers with volume production experience first. If no recommendation is available refer to IPC-7351 (Surface Mount Design and Land Pattern Standard) as a comprehensive land pattern design guide starting point.

All new land pattern designs should be tested in an actual production environment prior to release as a production design standard.

# Trace Routing Impacts Solder Joint





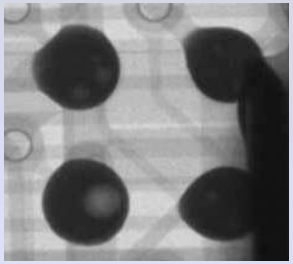
## Increased Mounting Pad Size Affected By:

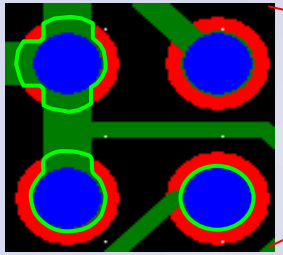
- Number Of Trace Connections To Each Pad
- Width Of Trace Connections To Each Pad
- Size of Pad
  - Small Pads Have Less Margin
- Uniformity Of Trace Egress Direction
  - Some Package Types Are More Sensitive Than Others
- Uniformity Of Trace Sizes

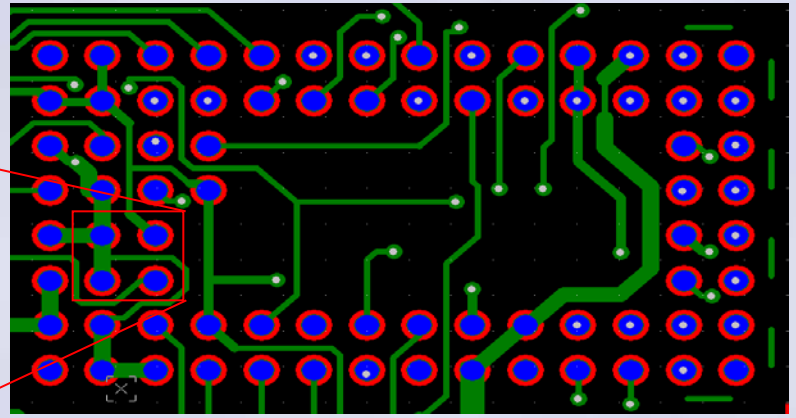
PLEXUS
The Product Realization Company

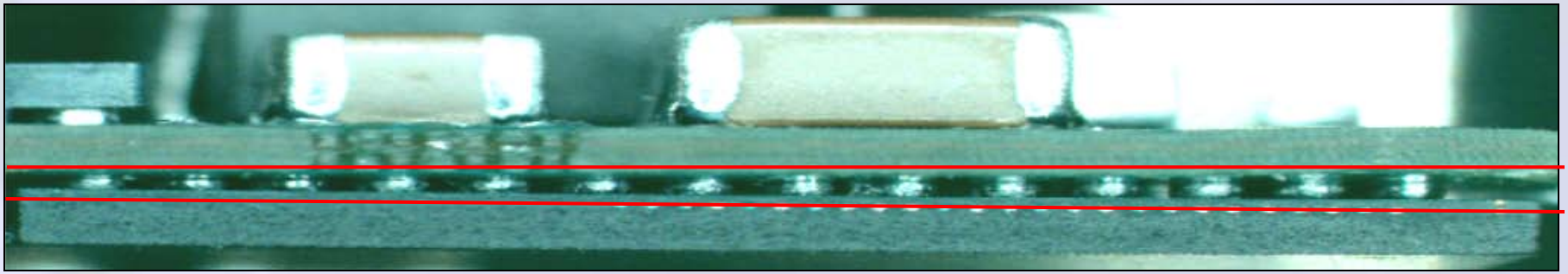
# Trace Routing Impacts Solder Joint

- Gradient Of Different Trace Sizes
- Localized Concentrated Large Trace Connections Increase Defect Potential









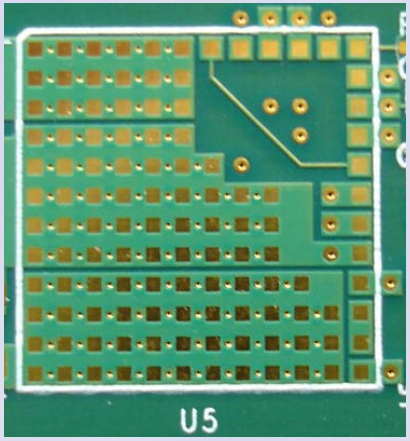
- Concentrations Of Design Variability Can Create:
  Solder Bridge, Open Connection, Insufficient Solder, Tilted Components

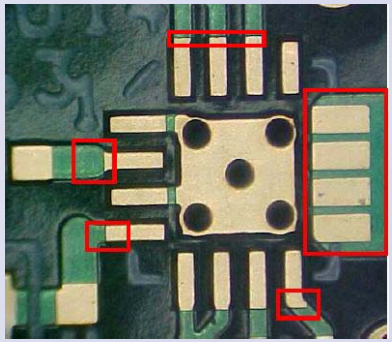
PLEXUS
The Product Realization Company

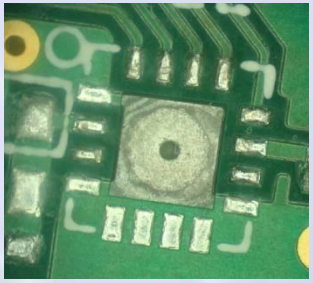
# Component Issues – LGA & QFN

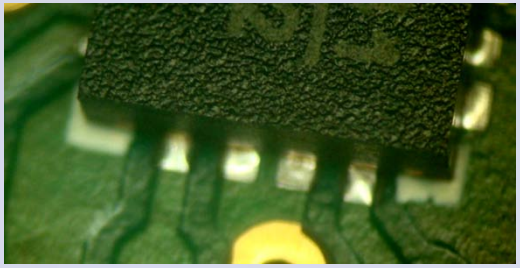
**Potential Issues:**

- Land Pattern Design
  - Pad Size Uniformity (SMD vs NSMD)
- Paste Volume Control
  - Pad to Pad Volume
  - Pad to Design Defined Volume
- Component/PCB Flatness
  - Internal Split Plane
  - NFP Removal Impacts
- Component/PCB Warpage
- Decrease Component Standoff Height
  - Decreased Reliability







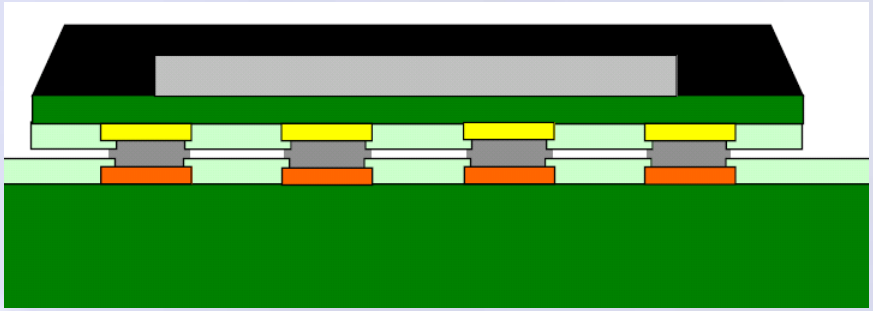
PLEXUS
The Product Realization Company

# LGA Pad Design

Solder Mask Defined

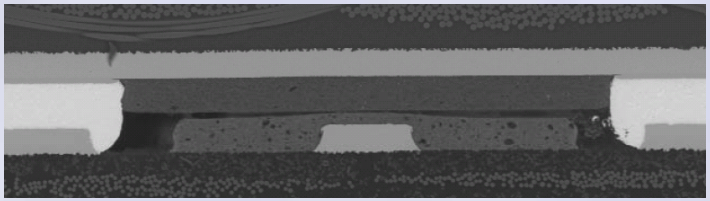
Non-Solder Mask Defined

- Solder wicking around NSMD pads produce significantly lower molten solder height.
- Solder mask defined pads should be used for LGA and 0.4mm & smaller pitch BGA/CSP packages.



# LGA/QFN Package Assembly

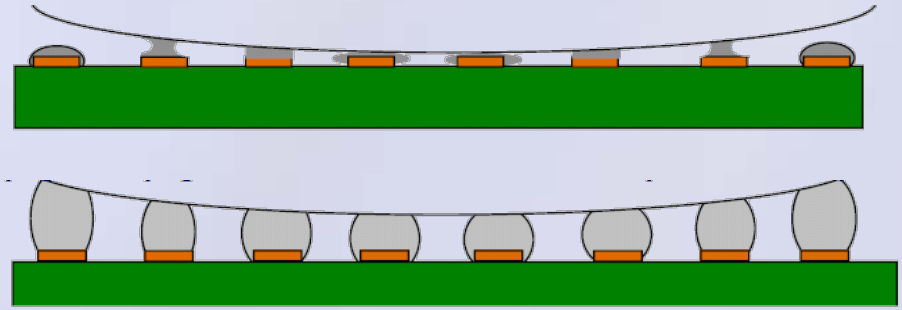
- Trace Routing Under Component Create Localized Height Variations
  - Standoff Height Variation



- Leadless Devices Are More Sensitive To PCB/Component Flatness/Warpage
  - Received Condition
  - In-process Condition (During Reflow/Rework Solder Process)

LGA Package

BGA Package

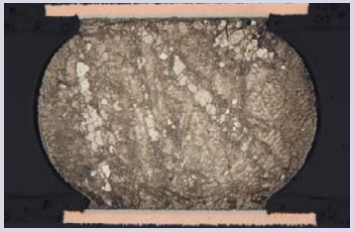


PLEXUS
The Product Realization Company

# Component Issues – Decreasing Pitch

- 0.2 mm

- 0.4 mm



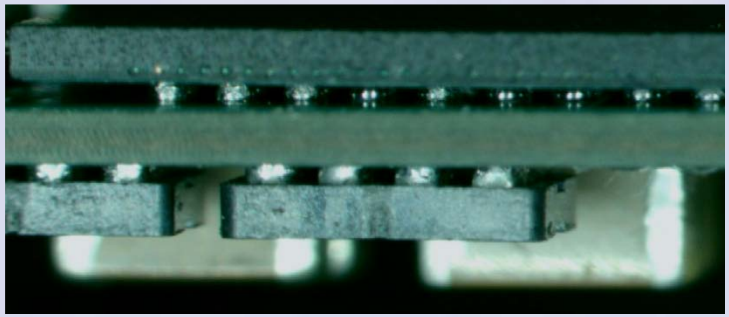
- 0.8 mm

- 1.0 mm

- 1.27 mm

## Potential Issues:

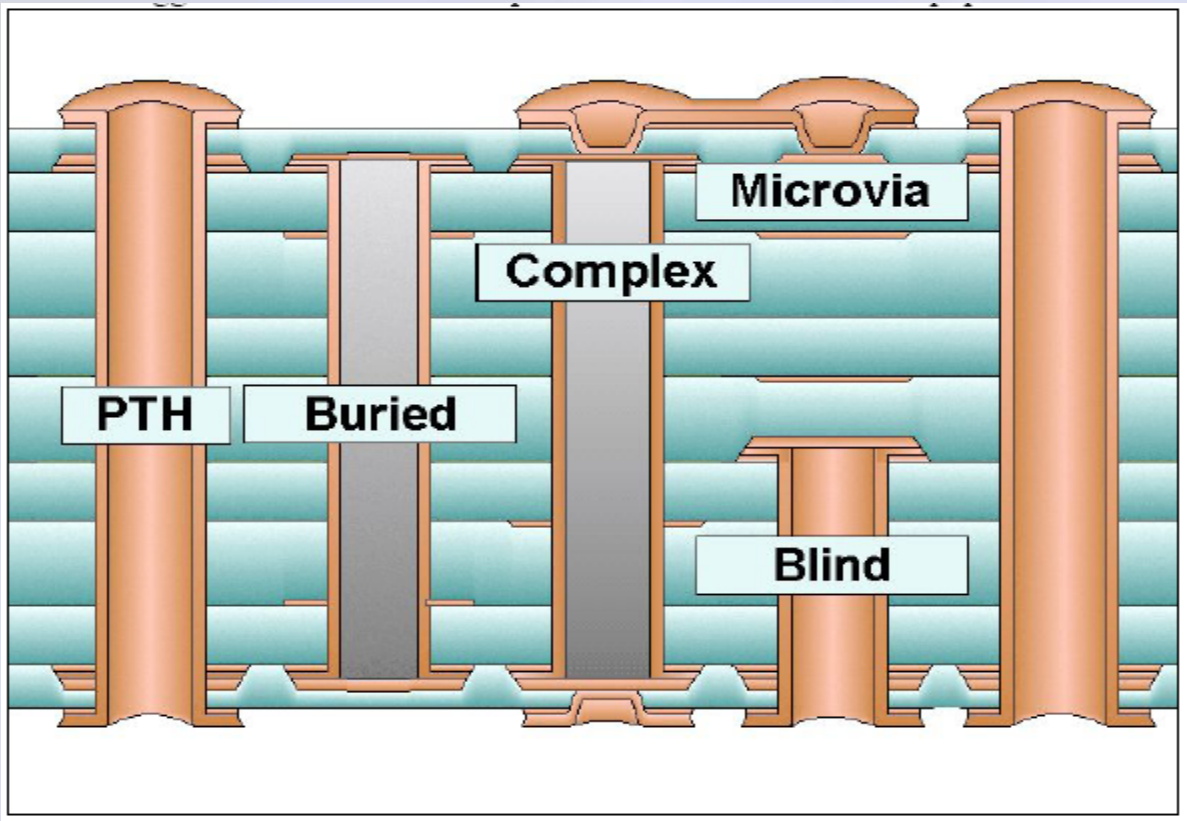
- Paste Volume Control
- Component/PCB Flatness
  - Internal Split Plane
  - NFP Removal Impacts
- Component/PCB Warpage



PLEXUS
The Product Realization Company

# Impacts of Via Design on Assembly

- Placement And
  - Types Of Vias In Pad Can Affect Assembly Solder Joint Formation
    - More Of An Impact On Smaller Components And/Or Lower I/O Count

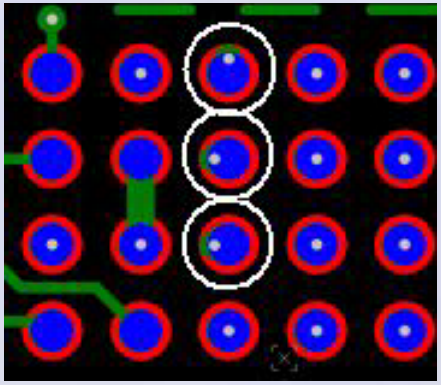


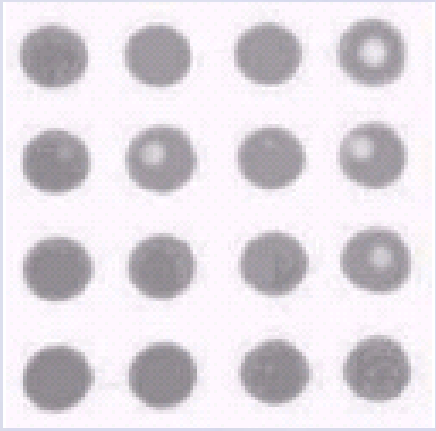

Design and Construction Affects on PWB Reliability, Paul Reid, IPC Apex Proceedings 2012.

PLEXUS
The Product Realization Company

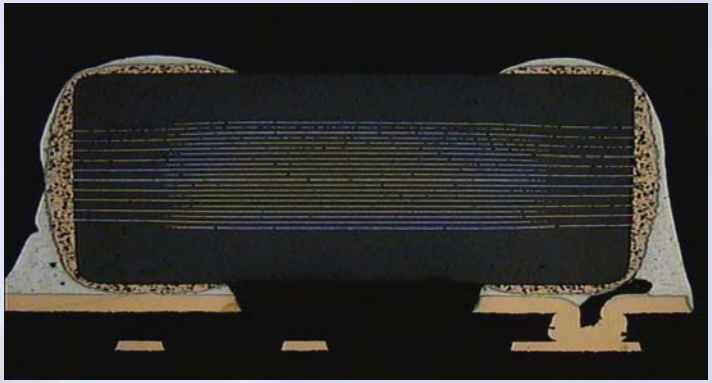
# Micro-Via in Pad

- Micro-Via Locations Should Be Consistent
  - Area Array Components - Center of Pad
  - Leadless (Chips, Chip Array, LCC, QFN, etc) and Leaded (QFP, SOIC, SOT, etc.) Components – Near Toe Fillet





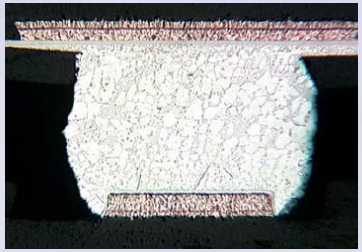
X-ray Image



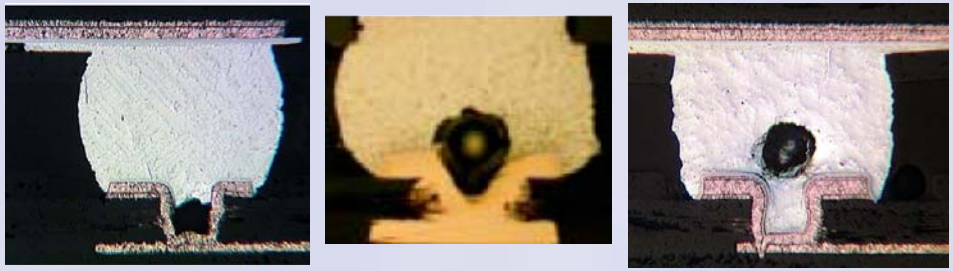
PLEXUS
The Product Realization Company

# Micro-Via in Pad

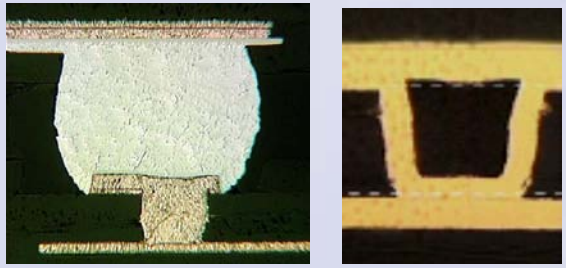
- Voids in Solder Joints
  - Unfilled Via in Pad
- Provide Flat Pad With Filled/Plated Closed Via
- Solder Joint Formation
  - Thermal Connection
    - Plane Connection
    - Multiple Connections
    - Stacked Via
  - Solder Volume
    - Via Location - Edge



No Via



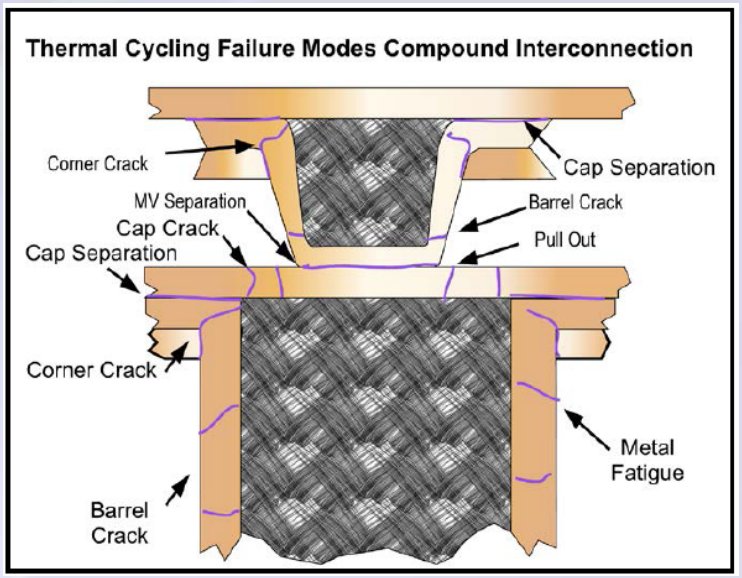
Unfilled Via's



Plated Closed & Filled Via

PLEXUS
The Product Realization Company

# Micro-Via Design – Reliability Impacts



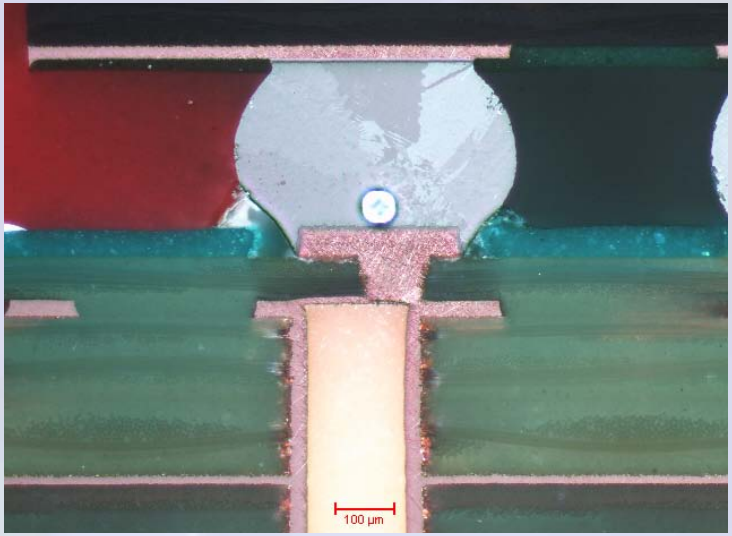

| IST Cycles to Failure - Tested at 190°C | | | | | | |
|---|---|---|---|---|---|---|
| Precon | As Received | | | Predoncitioning 6X260°C | | |
| Type | Buried | Microvia | | Buried | Microvia | |
| Circuit | Power | Staggered | Stacked | Power | Staggered | Stacked |
| Mean | 1000 | 1000 | 79 | 1000 | 887 | 3 |
| StDev | 0 | 0 | 32 | 0 | 182 | 0 |
| Min | 1000 | 1000 | 52 | 1000 | 582 | 3 |
| Max | 1000 | 1000 | 138 | 1000 | 1000 | 3 |
| Range | 0 | 0 | 86 | 0 | 418 | 0 |
| Coef Var | 0% | 0% | 41% | 0.0 | 0.2 | 0.0 |

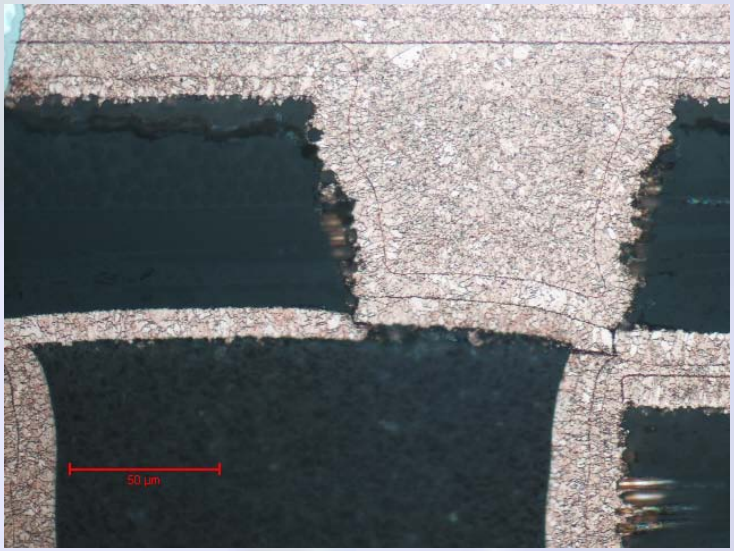
Different Via Hole Structures Impact Resistance To Assembly, Rework or Handling Damage

Design and Construction Affects on PWB Reliability, Paul Reid, IPC Apex Proceedings 2012.

PLEXUS
The Product Realization Company

# Stacked Via In Pad Failure Mode



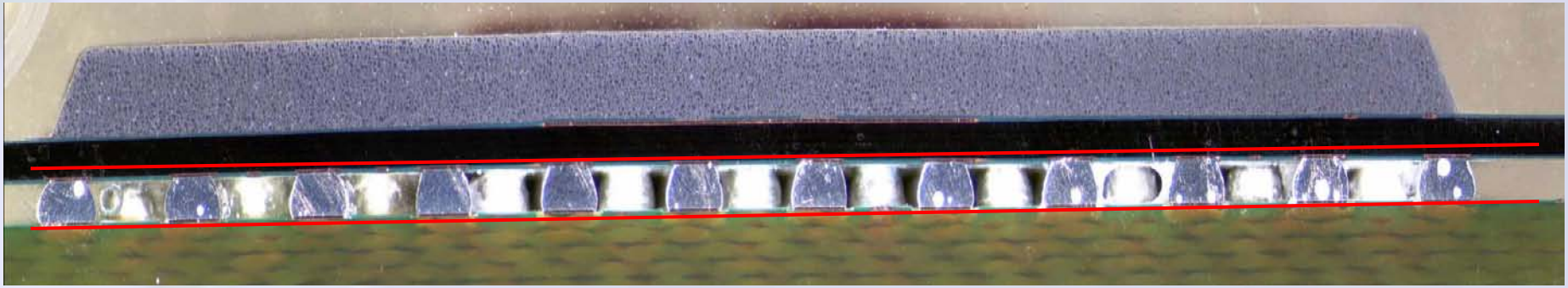

- Lead Free Laminate Is Less Ductile
- Lead Free Solder Is Less Ductile
- Increased Stress Transmission To Internal Connections
  - Test Fixture
  - Assembly Fixtures
  - System Integration
  - Environmental

PLEXUS
The Product Realization Company

# Component Issues - Warpage

- Higher Lead Free Solder Solidification and Process Temperatures, Increases The Amount Of Thermal Expansion Mismatch Of Components Which Can Increase Amount Of Component Warping During Assembly Process

- May Require Redesign Of Package (Material Selection) For Thermal Mass And Expansion Balance.

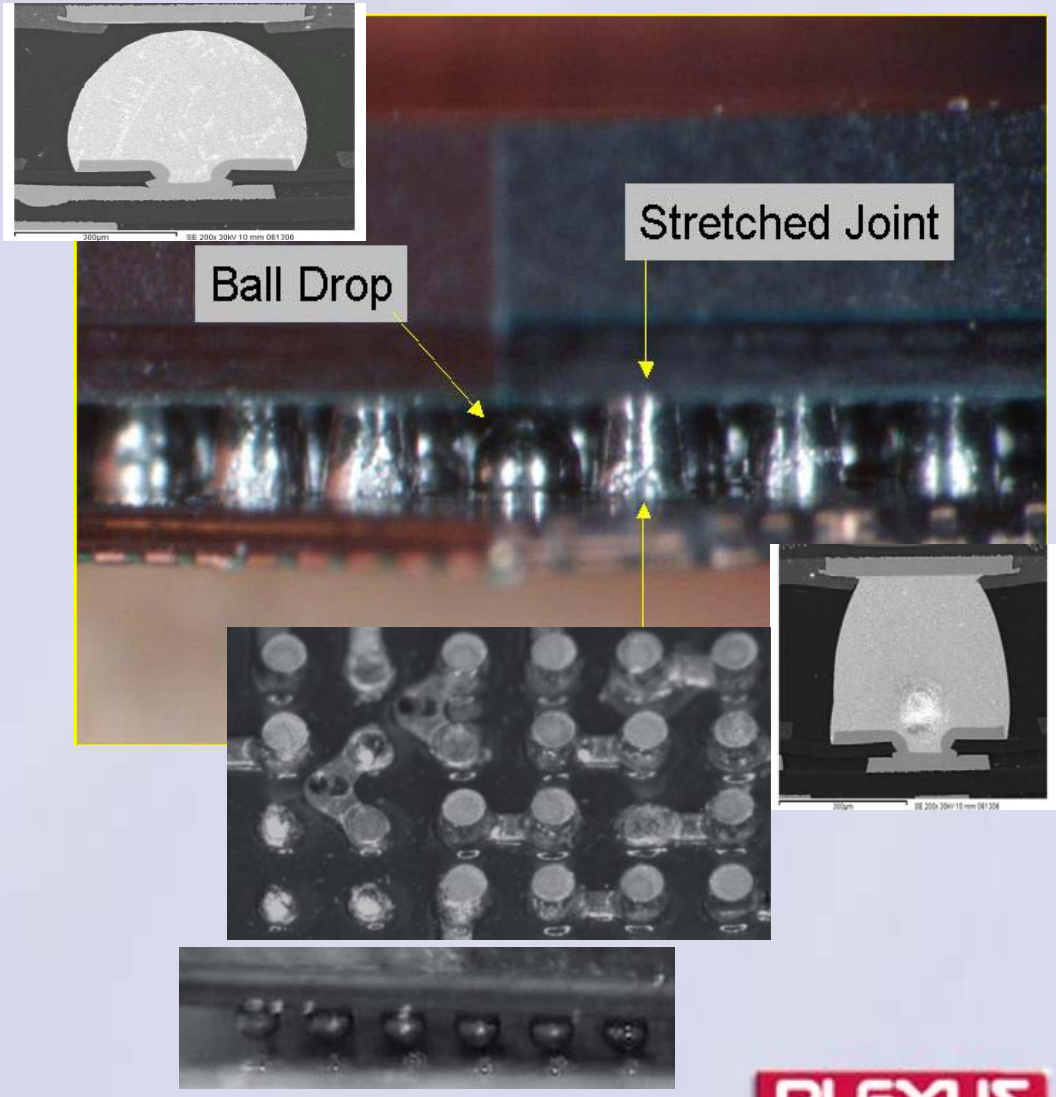


PLEXUS
The Product Realization Company

# Component/PCB Warpage Impacts

## Split Planes/Unused Pad Removal:

- Localize Changes In Thickness/Coplanarity Of PCB
- Potential Opens From Tilted Components (Teeter-Totter Effect)
- Potential Opens From “Dropped” Solder Connection
- Potential Reduced Reliability From Stretched Solder Joints

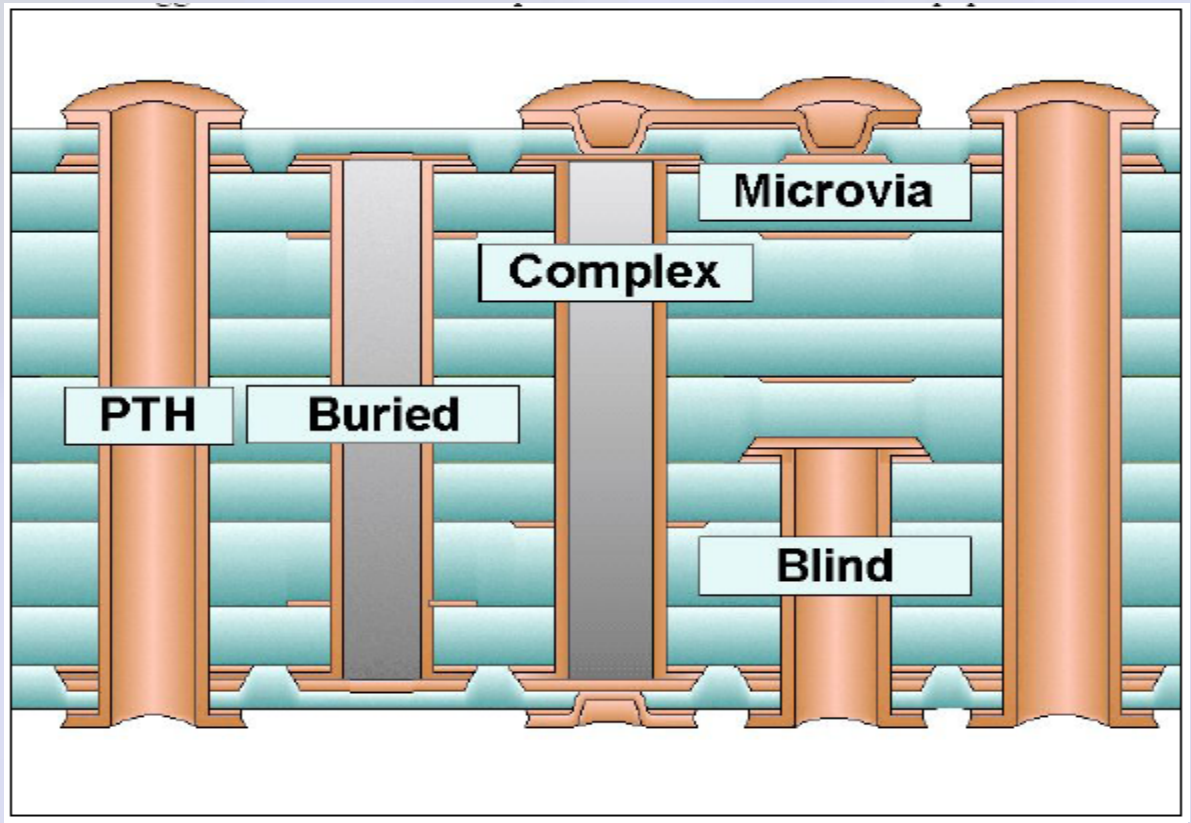



Some Photos Courtesy of Amkor

PLEXUS
The Product Realization Company

# Impacts of Via Design on Assembly

- Placement And
  - Types Of Vias In Pad Can Affect Assembly Solder Joint Formation
    - More Of An Impact On Smaller Components And/Or Lower I/O Count

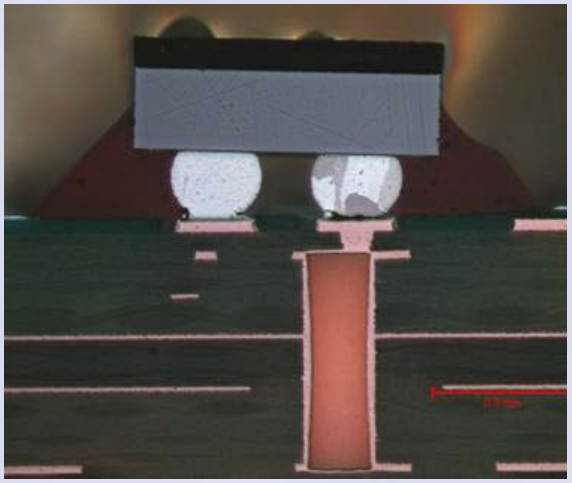


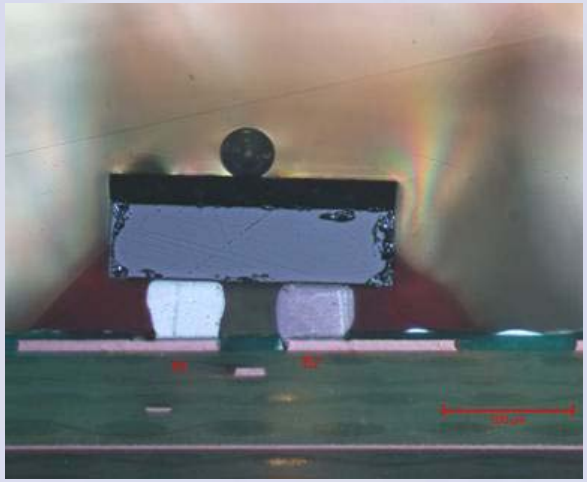

Design and Construction Affects on PWB Reliability, Paul Reid, IPC Apex Proceedings 2012.

PLEXUS
The Product Realization Company

# Stacked Via Holes

- Placement Of Stacked Vias Under Devices May Create Slight Mounting Pad Height Differences
  - More Of An Impact On Smaller And/Or Lower I/O Count





PLEXUS
The Product Realization Company

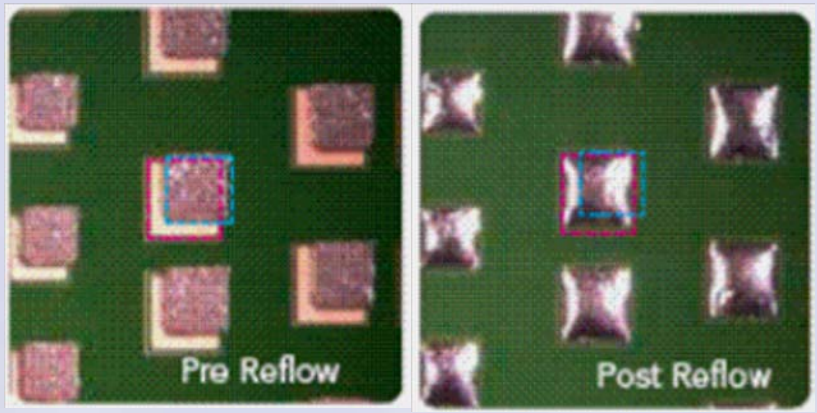
# PCB Finish Coatings

- **Hot Air Solder Level (HAL or HASL)**
  - ❖ Immerse PCB in Molten Solder, Hot Air Stripped Excess Solder.
  - ❖ Tin-Lead Coating (Sn63/Pb37, Sn60/Pb40)
  - ❖ Lead Free Coating (SnCu, SnCuNi)
  - ❖ Vertical, Horizontal Process
- **Immersion Gold (Gold Flash)**
  - ❖ 0.05 µm - 0.13 µm ( 2 - 5 µ") Electroless Gold over
  - ❖ 2.5 µm - 5.0 µm ( 100 - 200 µ") Nickel
- **Organic Surface Preservative (OSP)**
- **Immersion Tin**
- **Immersion Silver**
- **Immersion Palladium**

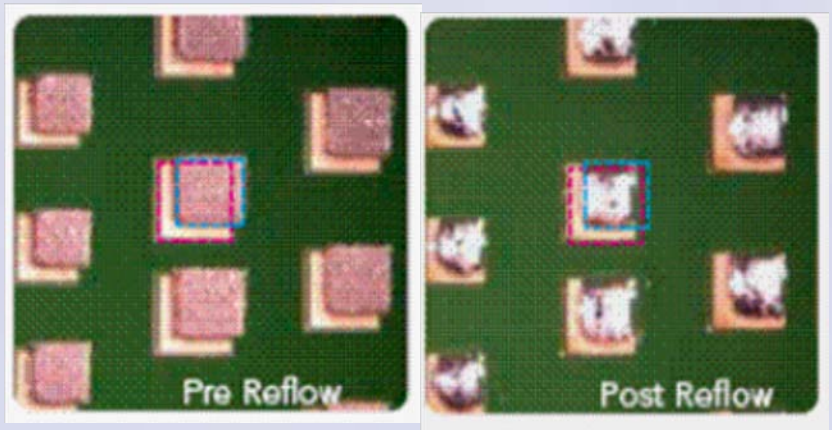
PLEXUS
The Product Realization Company

# Lead Free Solder Spread

- Stencil Alignment of Solder Paste To Pad Tolerance May Be Critical To Good Manufacturing Yields
  (Dependant Upon PCB Surface Finish)
- Example – OSP Finish




Tin-Lead Paste

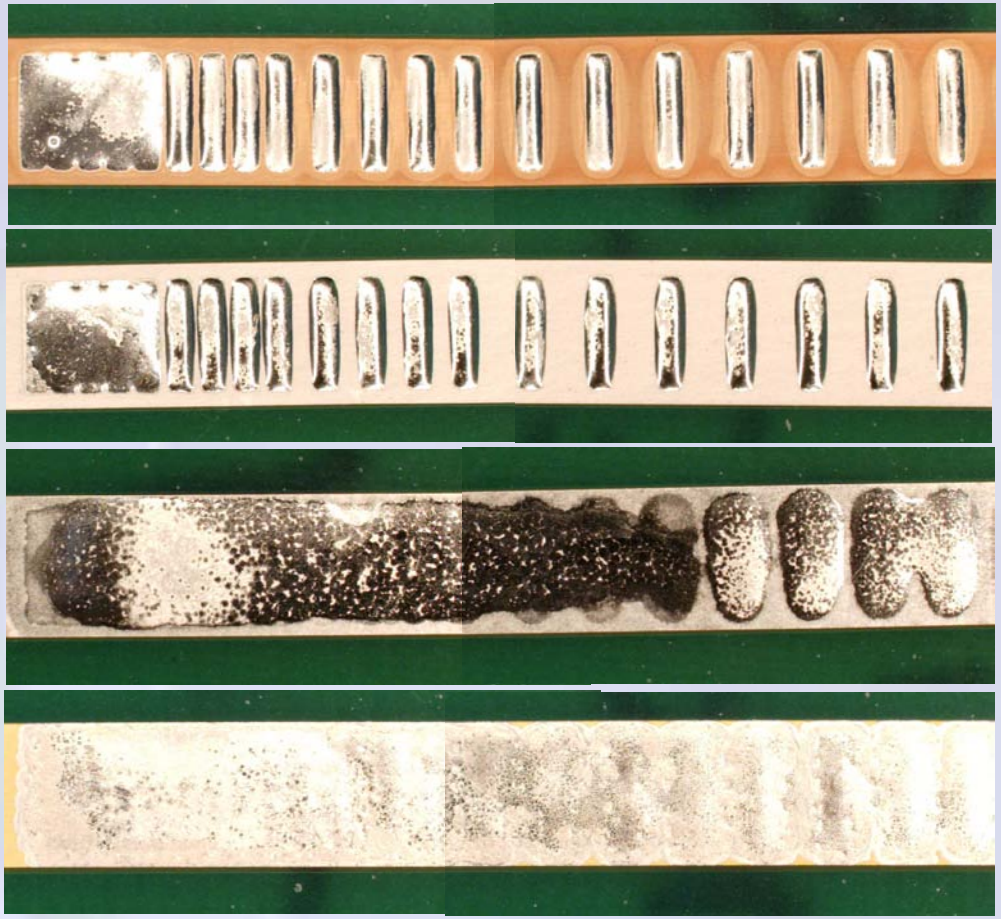



Lead Free Paste

Alpha Metals, SMT Mag Webcast, Jan 2006

PLEXUS
The Product Realization Company

# PCB Finish Vs Solder Spread



– OSP

– Immersion Silver

– Immersion Tin

– ENIG

**Amount Of Lead Free Solder Wicking Is Dependant Upon Finish**

PLEXUS
The Product Realization Company

# Process Control Issues

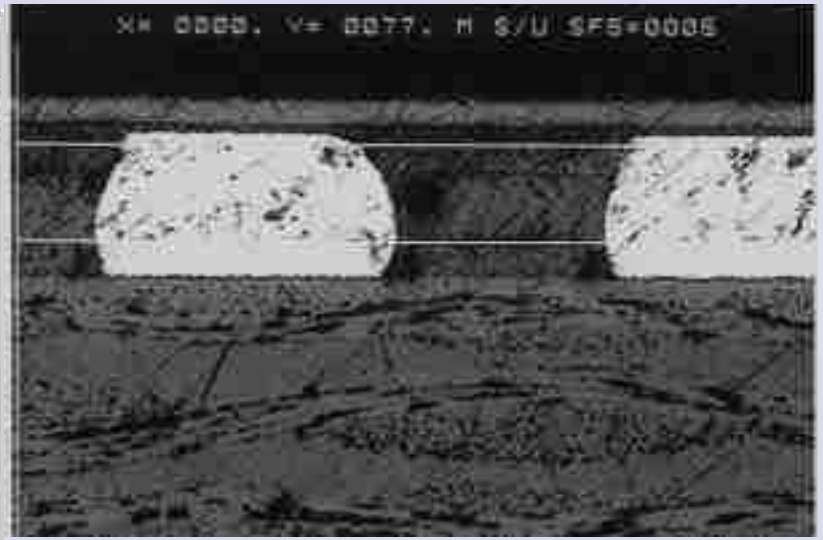
Component Attachment, Height Critical

- Solder Volume
- Land Pattern Design
- Component Interconnect Type
- Component Mass
- Component Soldered in Compression
- Component Soldered in Tension

# BGA vs CSP Z-Height

## Comparison

**256 I/O  1.27mm pitch vs.  46 I/O  .75mm pitch**



Potential Issues:

Paste Volume

Part Flatness

Board Warpage

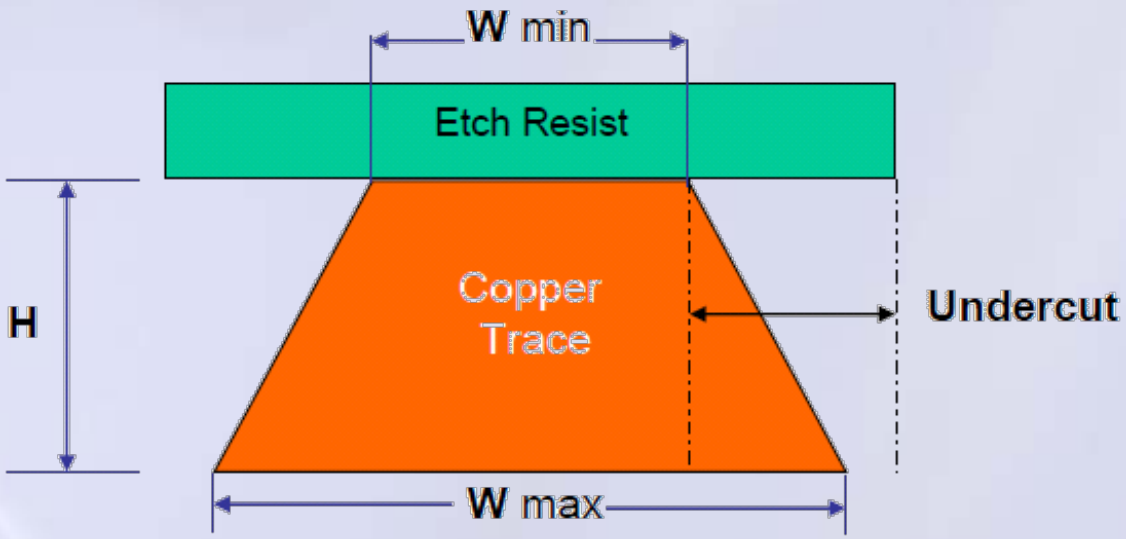
Same Scale

PLEXUS

The Product Realization Company

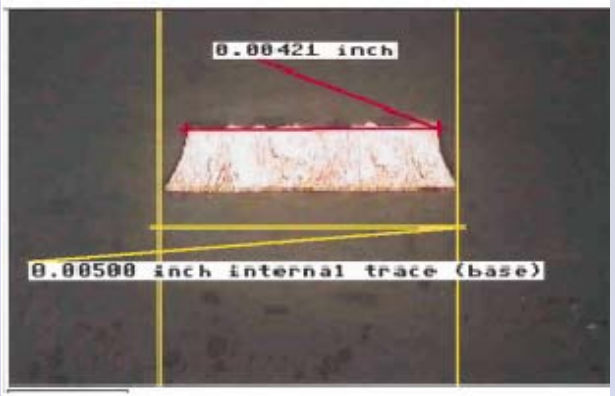
# Copper Etch Process

- **H = Height of Copper Trace**
- Etch Factor = 2H / (**W max - W min**)
- Etch Factor Should Be Larger Than 1.0

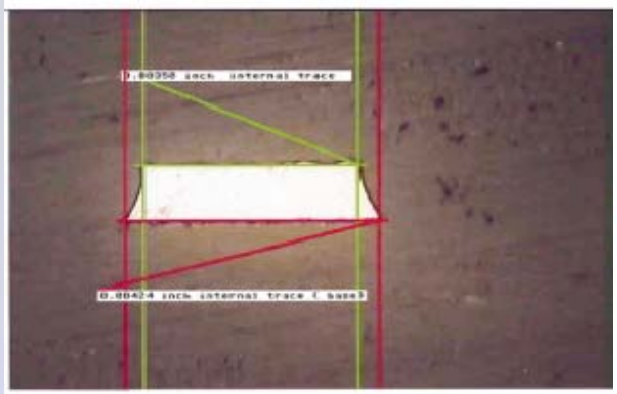



PLEXUS
The Product Realization Company

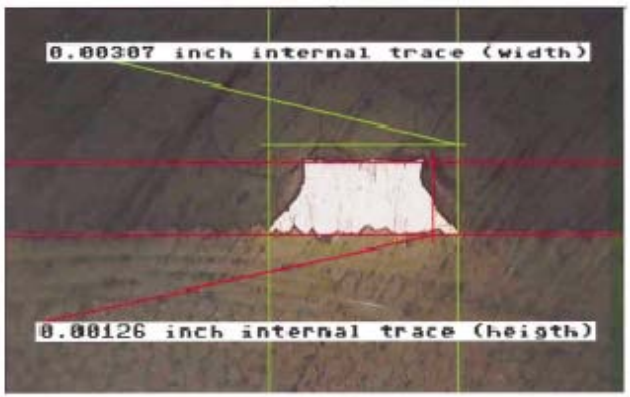
# Etched Trace Design Considerations




0.005” Trace 1 oz copper



0.004” Trace 1 oz copper




0.003” Trace 1 oz copper

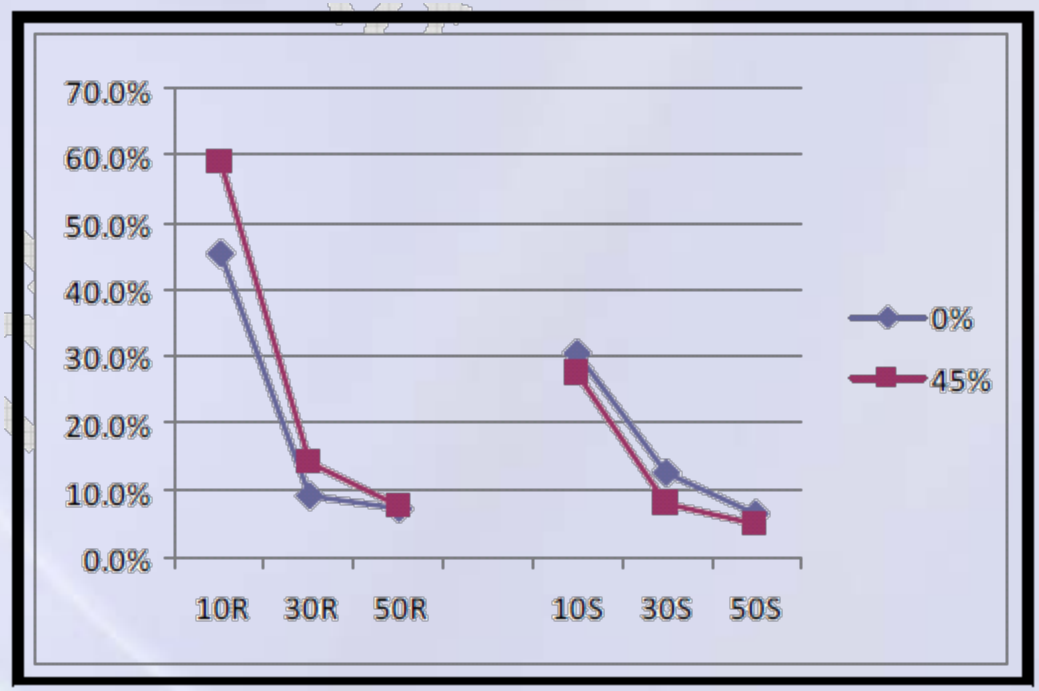
- Thinner Copper Produces More Precise Geometries on Lines Less Than 0.005” in Width
- Actual Conductor Shape is Close to a Trapazoid
- Copper Thickness is Slightly Reduced After PCB Processing

PLEXUS
The Product Realization Company

# Pad Size Reduction

Current Procedures For Applying Uniform Etch Compensation Values Across All Surface Features Are Inadequate.

Below Illustrate A Near Exponential Reduction In Pad Size As The Pad Gets Smaller, For Both Round And Square Pads In Either Orientation.



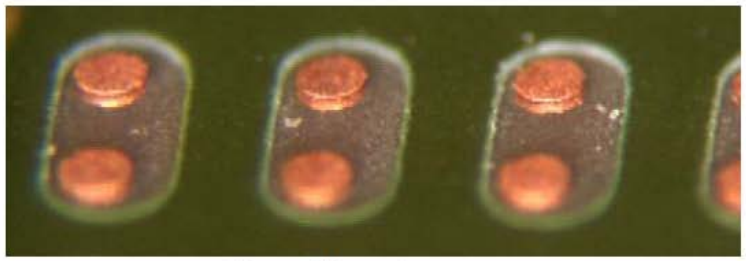

*Percentage Reduction by Pad Size*

PLEXUS
The Product Realization Company

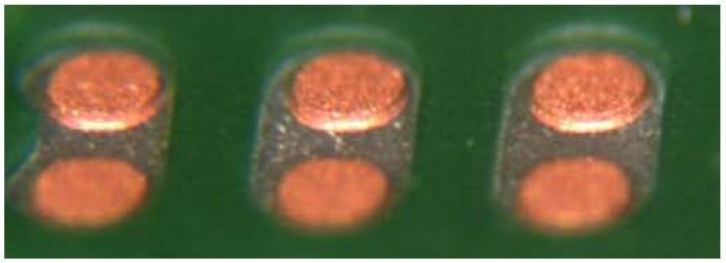
# PCB Fabricator Variation

## Pad Size Comparison

– Same design data may not yield same PCB pads sizes.



Supplier A



Supplier B

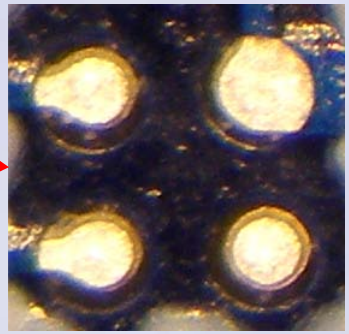
DFM Rules for Smartphones: An Analysis of Yield on Extremely Dense Assemblies, Jimmy Chow et al., IPC Apex Proceedings 2012.

PLEXUS
The Product Realization Company

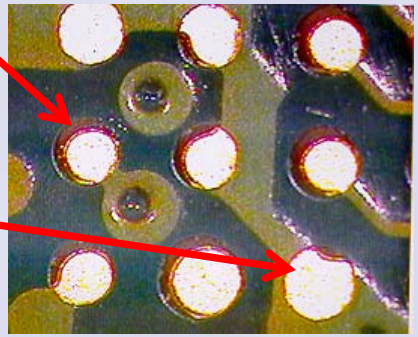
# Solder Mask Opening Design

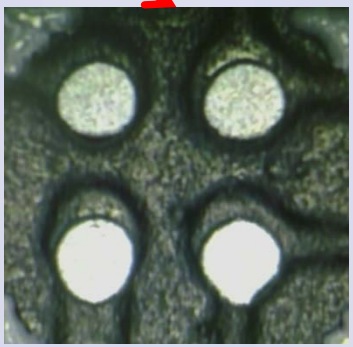
## Pad Geometries

- Non-Solder Mask Defined (NSMD)
  - Size of Pad Defined By Copper Pad and Interconnections (Variable Size)
  - Solder Encapsulates Pad
  - Limited to Components With Lead Pitch Greater Than 0.4mm



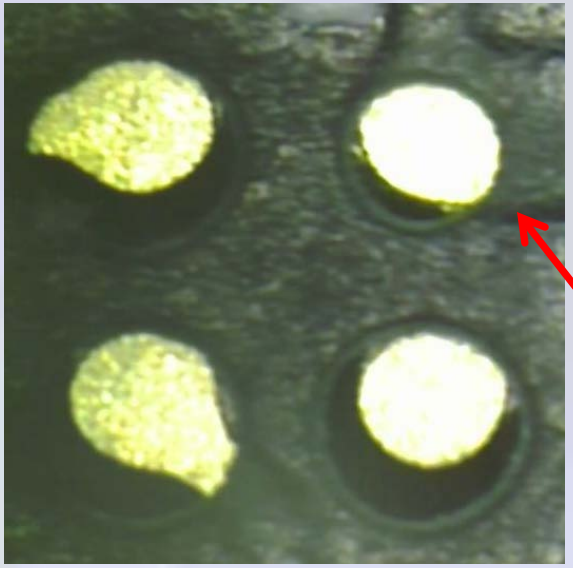
- Solder Mask Defined (SMD)
  - Size of Pad Defined By Solder Mask Opening (Uniform Size)
  - Solder Covers Exposed Pad (Fills Opening)
  - Required For Components With Lead Pitch 0.4mm or Less.
  - Preferred for Leadless Array Devices Like LGA's, Multi-row QFN's, etc.

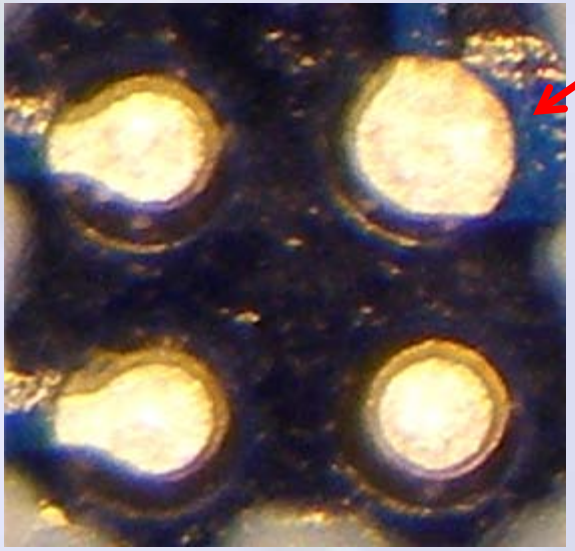




PLEXUS
The Product Realization Company

# PCB Fabricator Variation





- Solder Mask Fabrication Consistency
  - Fabrication Note Interpretation
    - “No Solder Mask Permissible On Pads Unless Provided This Way On Supplied Artwork ”
    - (Selective Solder Mask Opening Changes Between PCB Suppliers)

PLEXUS
The Product Realization Company

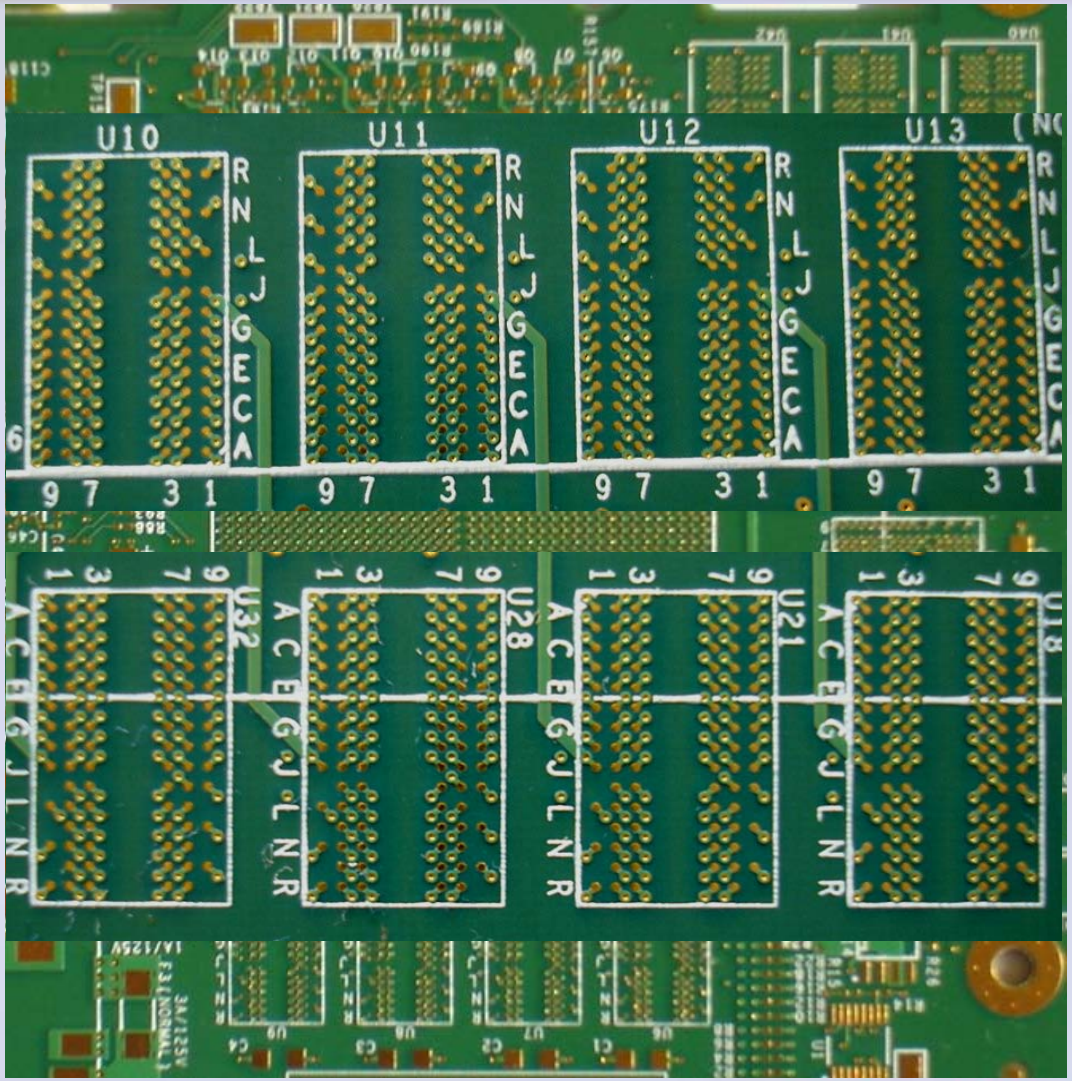
# Silk Screen Design

## Low Component Stand-off Height

- Tilted Component
- Open Joints (standoff from PCB)
- Misalignment

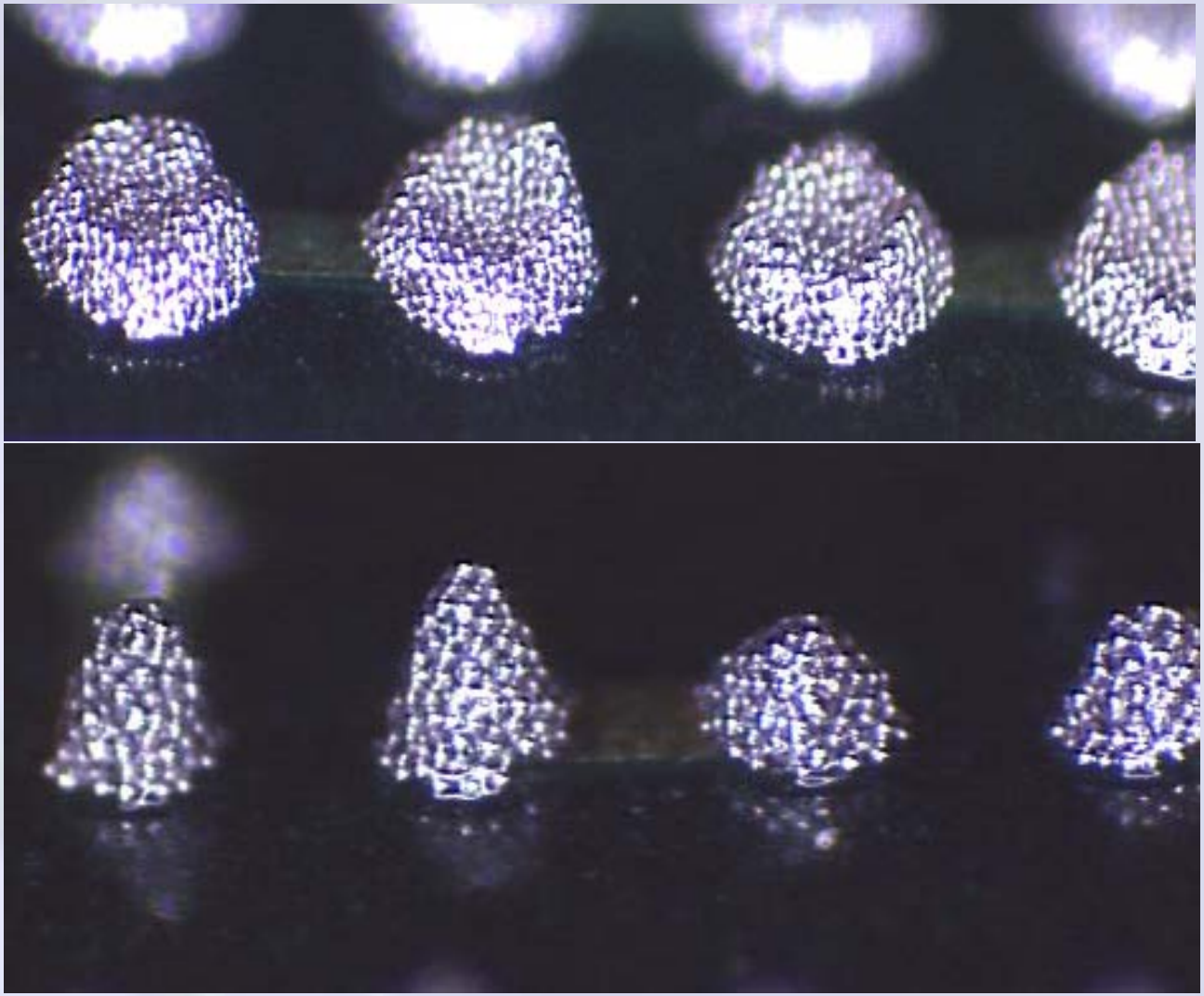
## Component Types

- Leadless
  - QFN, DFN, Passives, etc.
- Fine Pitch Area Array
  - BGA, WL-CSP, CSP, etc.




PLEXUS
The Product Realization Company

# Solder Paste Printing



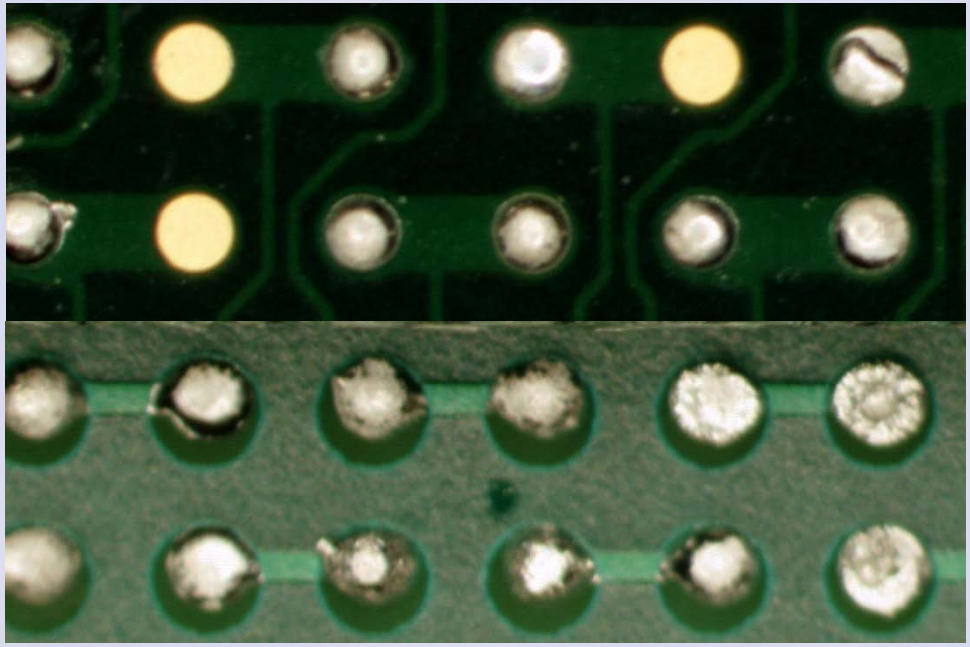
- Low Aspect Area Ratio Printing
- High Aspect Area Ratio Printing

PLEXUS
The Product Realization Company

# Solder Paste Printing Volume

Leadless Device Usage Increase (DFN, QFN, LCC LGA) & Ultra-fine Pitch Components Impacts:

- Tighter Tolerance On Solder Paste Volume – Thinner Stencil
- Increased Uniformity Of Paste Volume Across Component (Pad to Pad)
- Paste Volume/Pad Trace Egress Direction Impact
  - Some Package Types Are More Sensitive Than Others



**Open/Unwetted LGA Solder Connection**

PLEXUS
The Product Realization Company

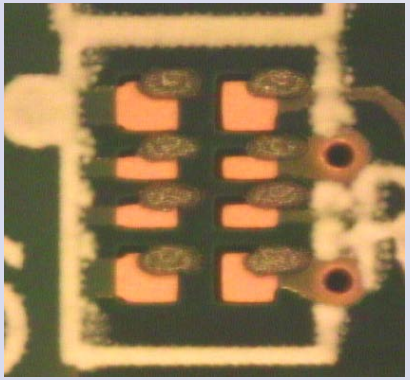
# Assembly Issues

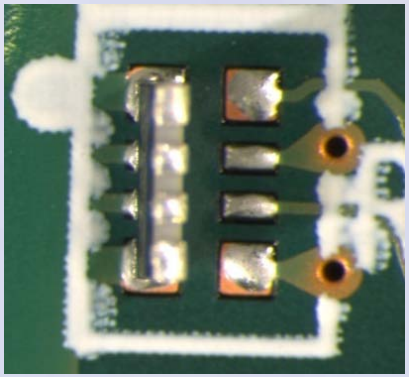
- Stencil Design
  - Registration To PCB
    - Decrease PCB/Panel Size
    - Modify Pad/Aperture Size
  - Component Mounting Restrictions
    - Minimum Solder Paste - Minimal Solder Pedestal
    - Paste Location on Pad - Improve Component Alignment

# Stencil Tolerances

## Artwork Feature Positional Tolerances Increase

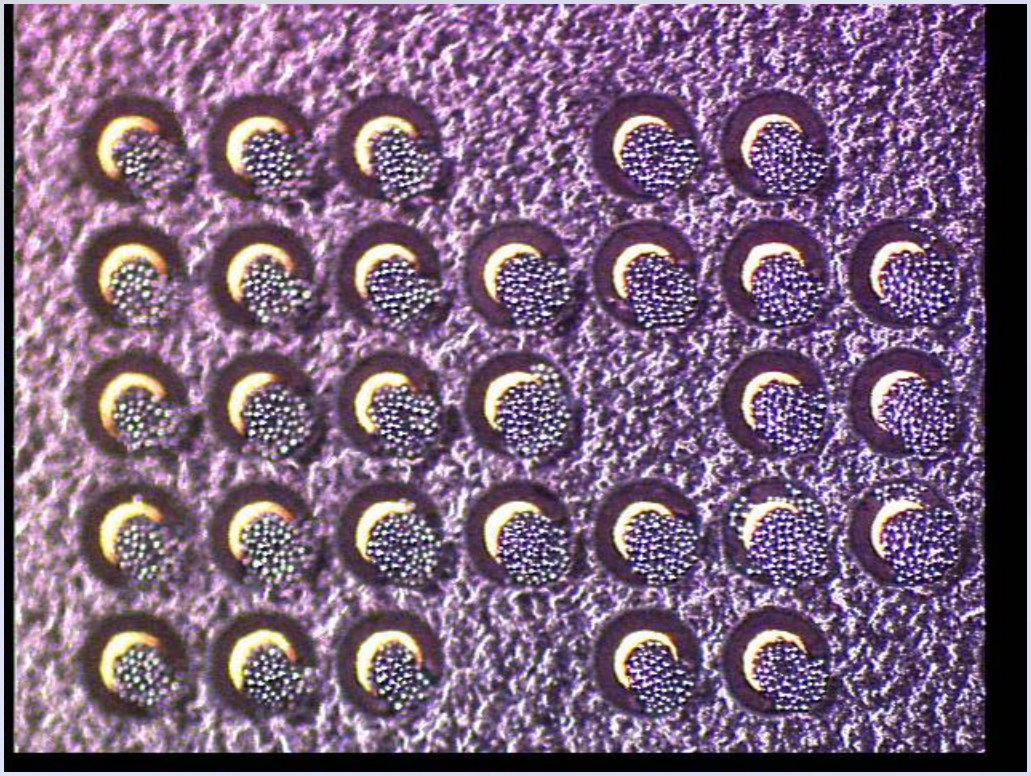
- Fabrication Tolerances Artwork Registration
  - Etched Feature Position
  - Etched Feature Size
  - Etched Feature Quality
  - Etched Feature Directional Etch
- Stencil Print Directional Compensation Orientation





PLEXUS
The Product Realization Company

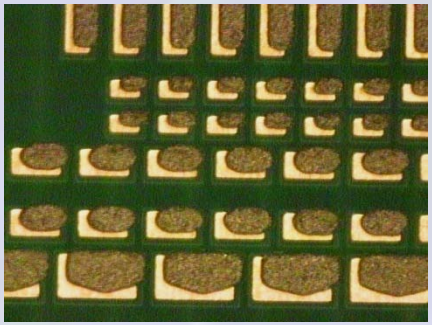
# Printed Paste Offset



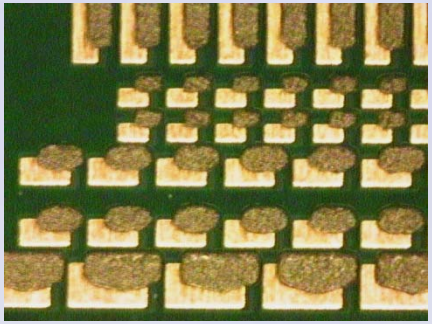
•Printing Process Positional Tolerances
Windows Decrease With Decrease in CSP Pitch

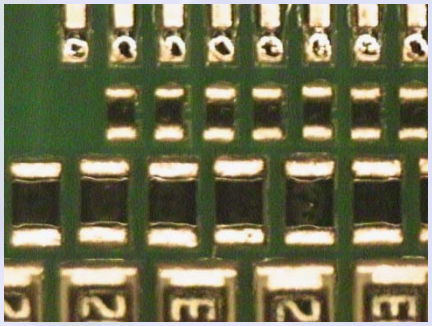
PLEXUS
The Product Realization Company

# Stencil To PCB Alignment

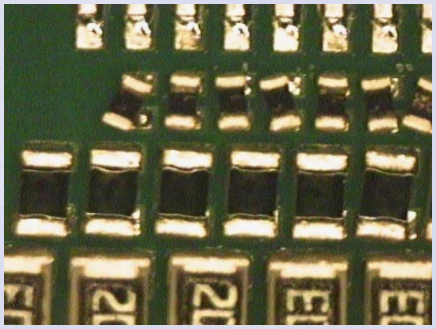


- Smaller Components Decrease Total PCB & Assembly Process Tolerance
- Minor Misalignment Can Impact Process Yields





3 mil



6 mil

PLEXUS
The Product Realization Company

# Offset Paste – Normal Placement

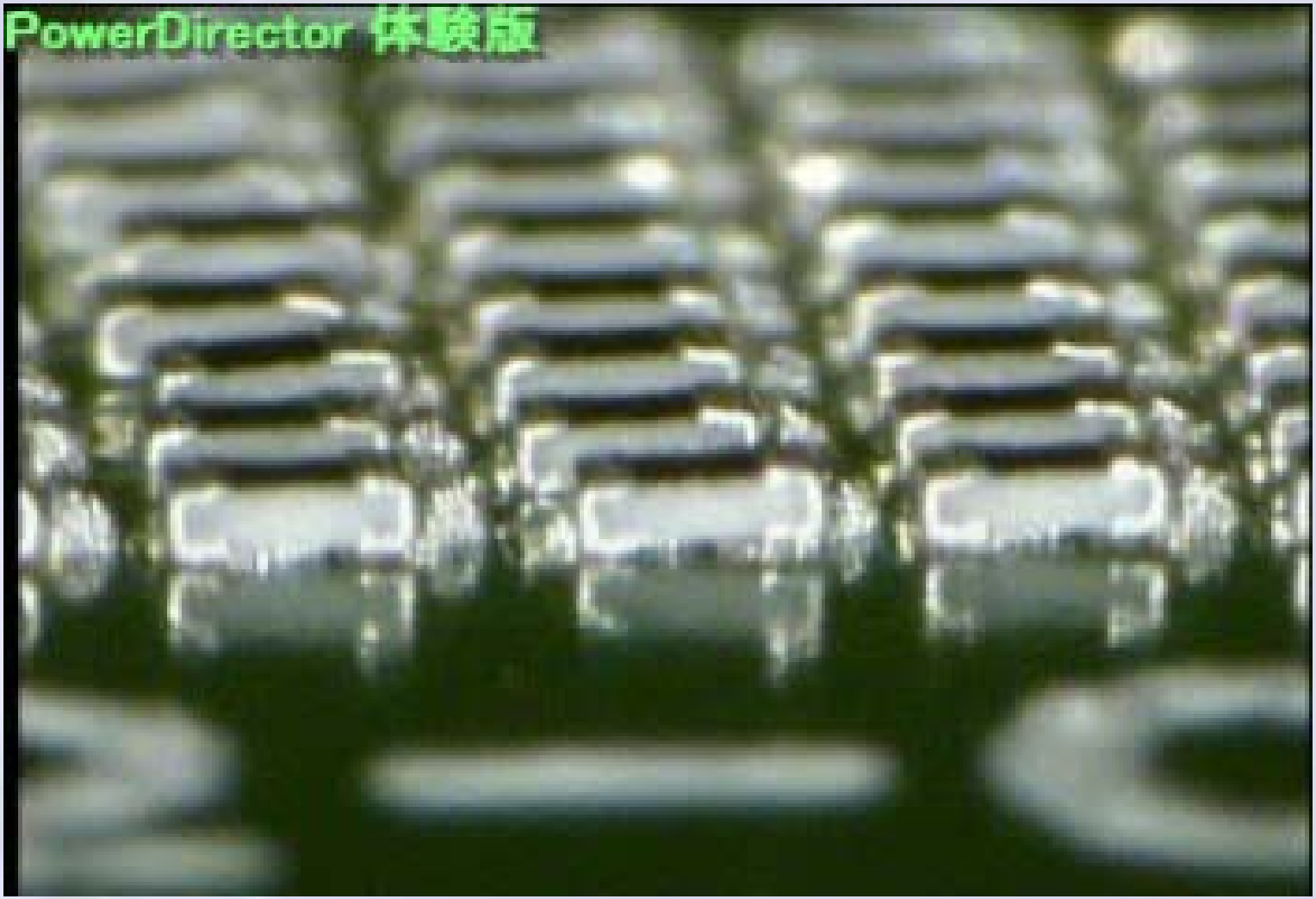
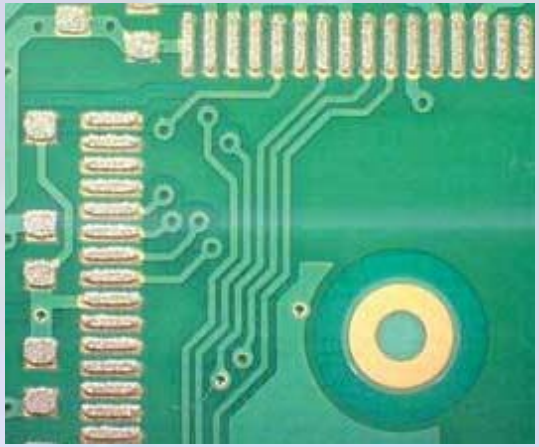



Photo Courtesy of Juki Automation

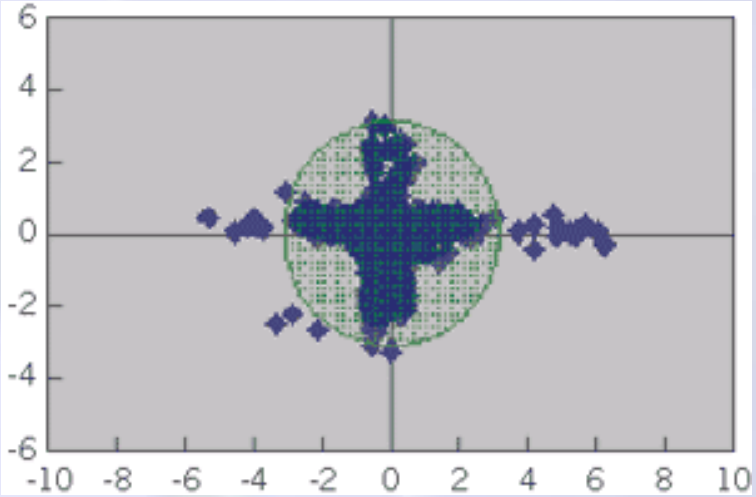
PLEXUS
The Product Realization Company

# Match Tooling Design To Parts

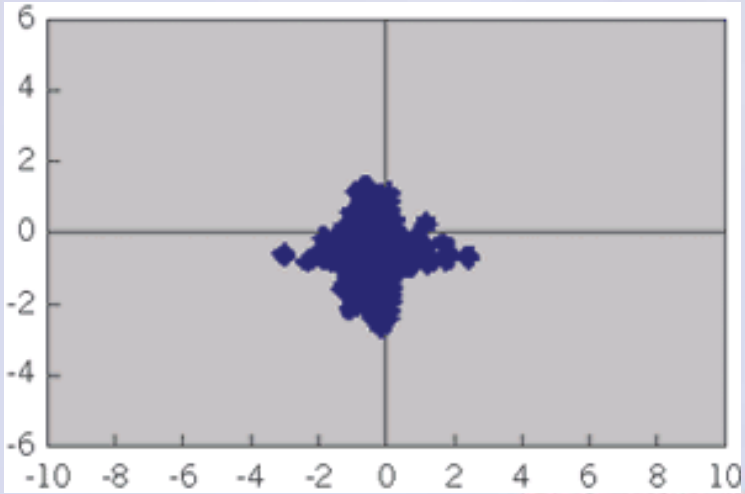


- Normal Manufacturing Process Variability May Exceed Allowable Assembly Process Tolerance For High Yield, Reliable Assembly
- Matched Tooling (Stencils) To Materials (PCB) May Be Required

PCB to Stencil Pad Positional Deviation Measurements No Adjustments

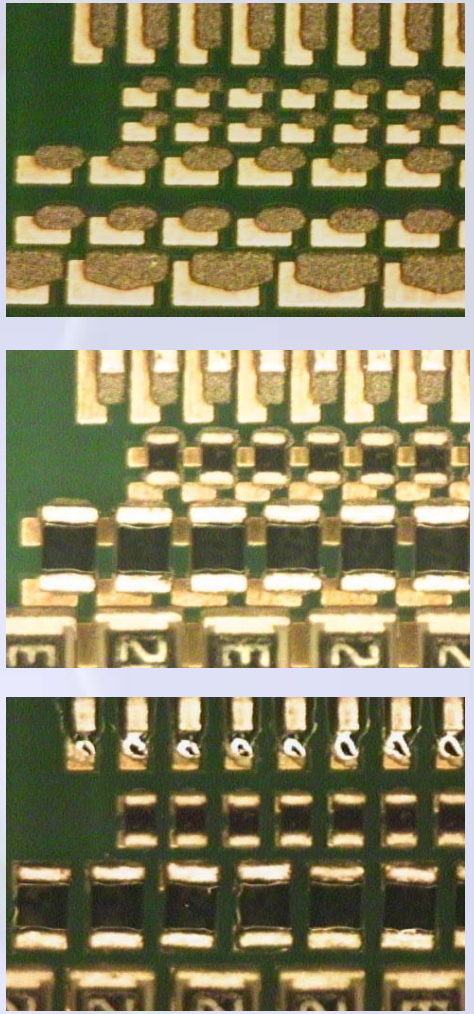



PCB to Stencil Pad Positional Deviation Measurements With Scaling Adjustments




PLEXUS
The Product Realization Company

# Match Placement To Paste



- Slight Offset Of Solder Paste And Component Placement May Improve Soldering Yields
  - Paste and Placement Must Have Same Offset
  - Tombstone Passives
  - BGA Voiding

PLEXUS
The Product Realization Company

# Matched Offset – Paste & Part

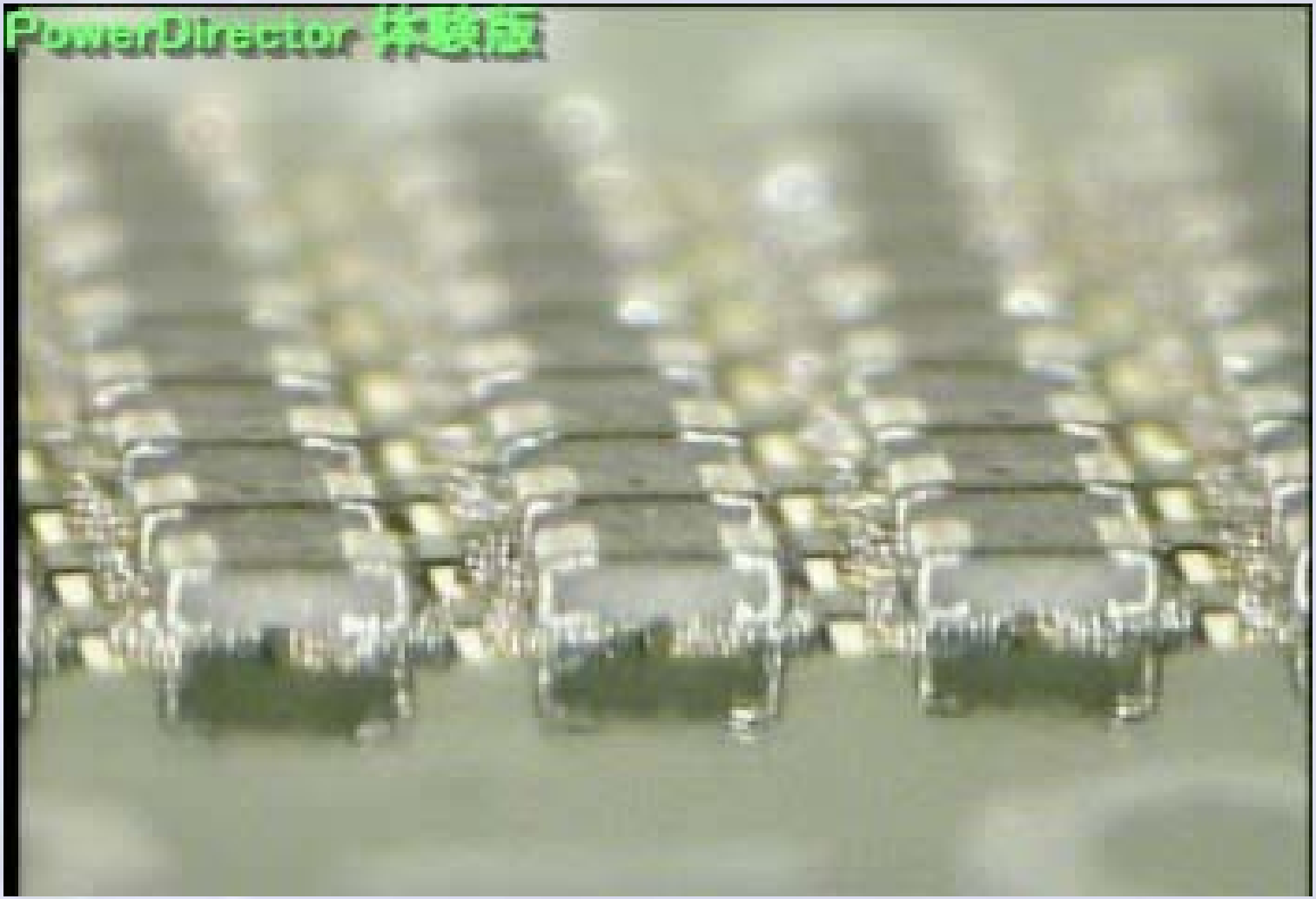
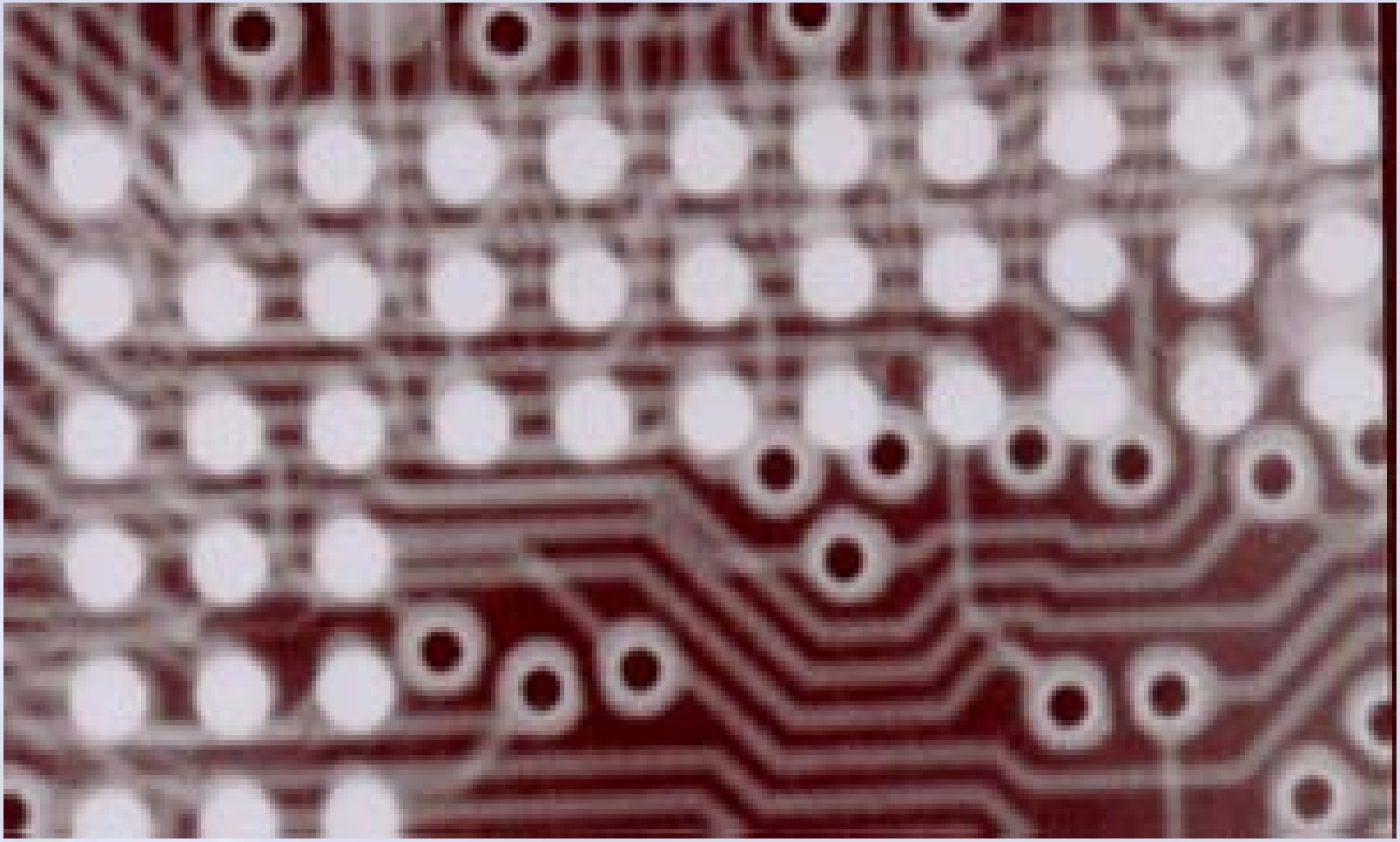


Photo Courtesy of Juki Automation

PLEXUS
The Product Realization Company

# Part Offset



X-ray of Component Offset in Solder Paste

PLEXUS
The Product Realization Company

# Warpage & Thermal Profile Issues

## May Require Change In Production Process

Reflow Profile To Bridge PCB Warpage Gap.

(Decreased Thermal Change Rate And Delta T Vertically In PCB – Reduce Surface To Cooler Location Temperature Delta - TCE Induced Warpage)

Large ΔT across Board

PLEXUS
The Product Realization Company

# Closing Thought

"We can't solve problems by using the same kind of thinking we used when we created them."

Albert Einstein

**Don't Forget About Reflow Process Induced Warpage/Coplanarity Issues.**

PLEXUS
The Product Realization Company