Root Cause Failure Analysis of Printed Circuit Board Assemblies

Presented By: Dale Lee

8 March 2011

Wisconsin Chapter

SMTA

Surface Mount Technology Association

PLEXUS

The Product Realization Company

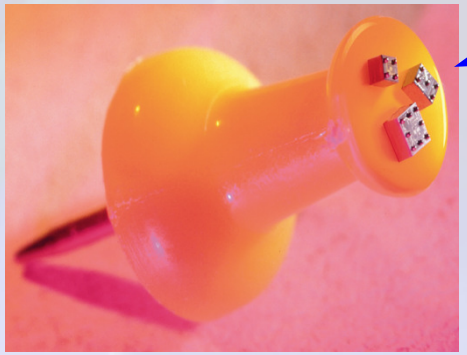
What do you mean 0% first pass yield?

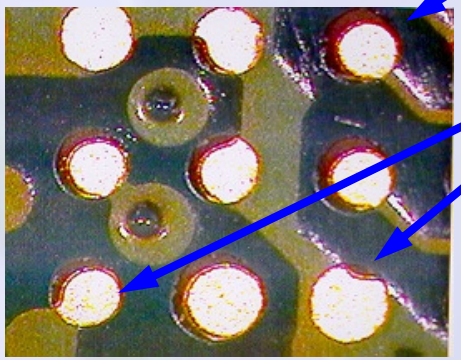
My process is in control , so what happened?

PLEXUS
The Product Realization Company

Photo Courtesy of G. Toren

Today's Electronic Designs





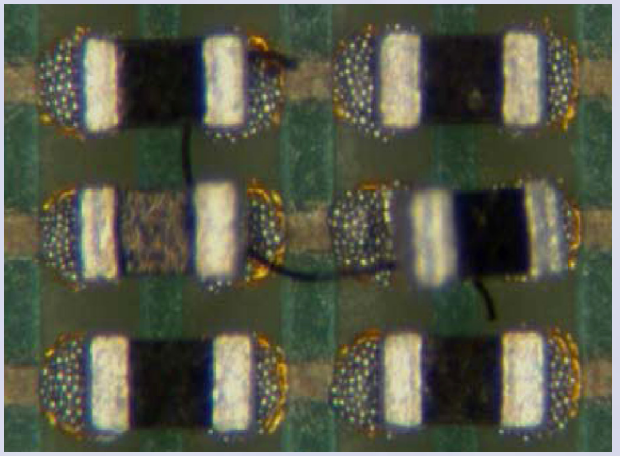
- Component Packages Are Getting Smaller
  - 0603 > 0402 > 0201>01005
  - BGA > CSP > WL-CSP > ?
- Assembly Design Density Is Increasing
  - Tighter Component to Component Spacing (Spacing Smaller Than 0.020" Common)
  - Smaller Copper (Pad) Interconnections
- Assembly Process Margins Are Tighter
  - Thermal Balance @ Pad Level Is Critical (Trace/Via Connection Size to Pads)
  - Component Placement Accuracy

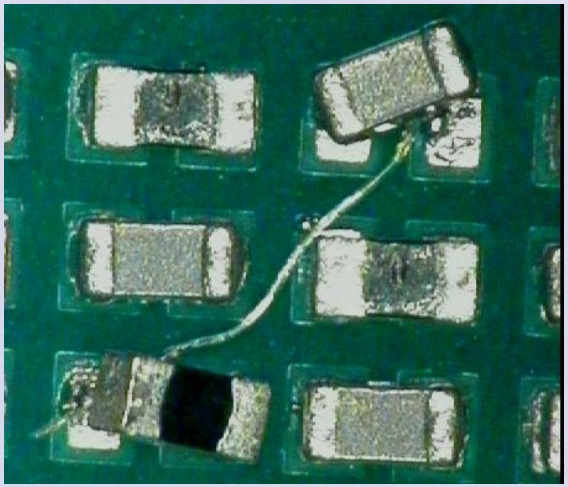
(The Thickness Of A Piece of Paper Can Be Difference Between 100% Rework or High Yields)

PLEXUS
The Product Realization Company

Effects of Mfg Environment

- Elements From Manufacturing Facility May Contribute Defects. These include:
  - Process Equipment Materials
  - ESD Control (Smock/Wrist Strap)
  - Paper Work
  - Component Packaging
  - Facility Environment
  - Clothing
  - Operators (Hair)



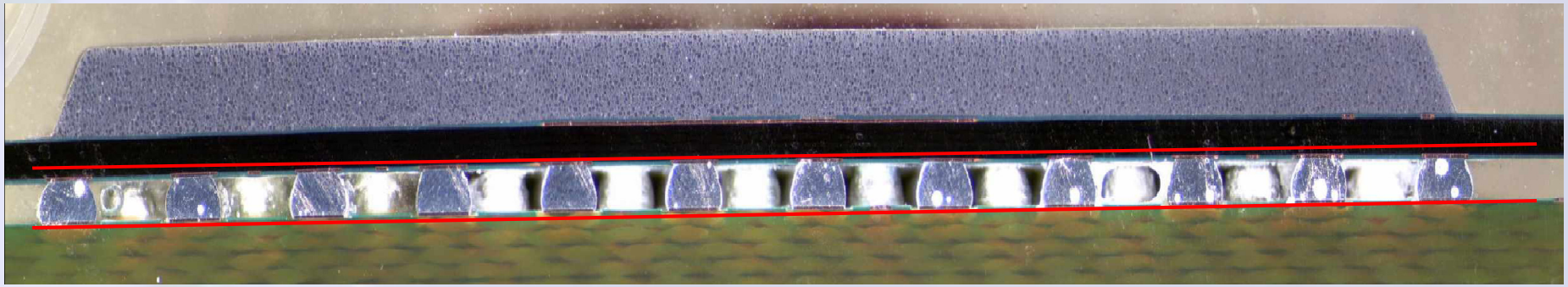


PLEXUS
The Product Realization Company

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.

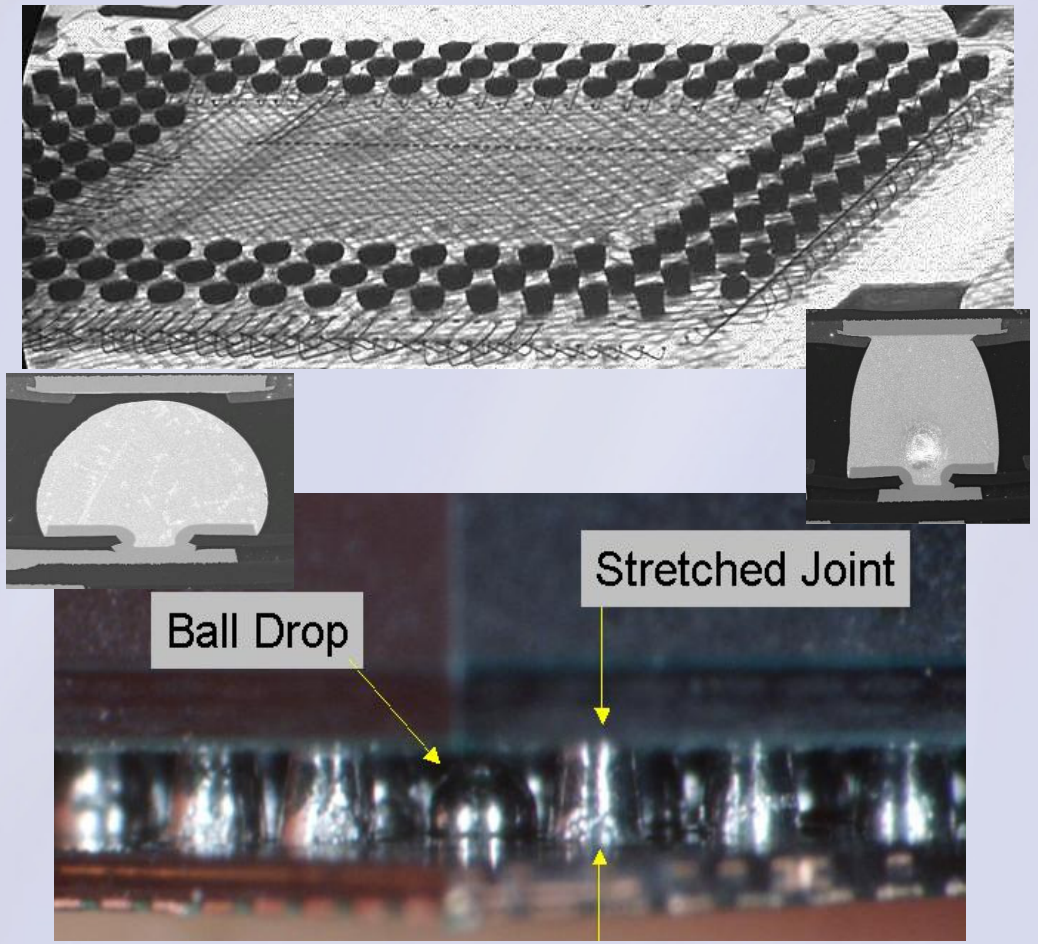


PLEXUS
The Product Realization Company

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

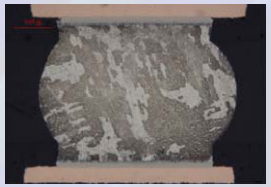


Some Photos Courtesy of Amkor

PLEXUS
The Product Realization Company

Component Issues - Decreasing Pitch

0.2 mm



0.4 mm

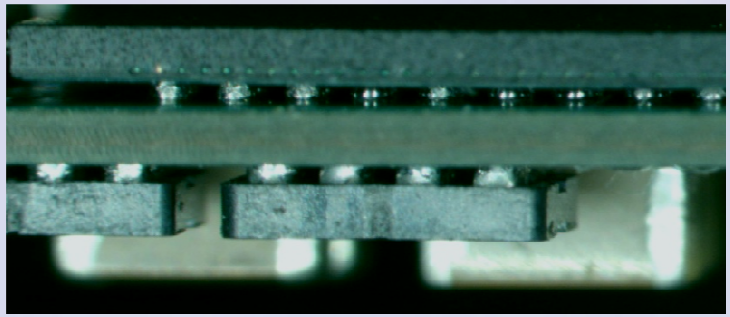
0.8 mm

1.0 mm

1.27 mm

Potential Issues:

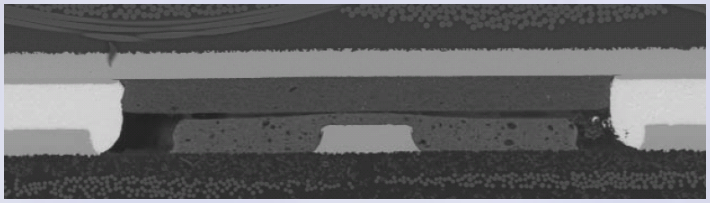
- Paste Volume Control
- Component/PCB Flatness
  - Internal Split Plane
  - NFP Removal Impacts
- Component/PCB Warpage



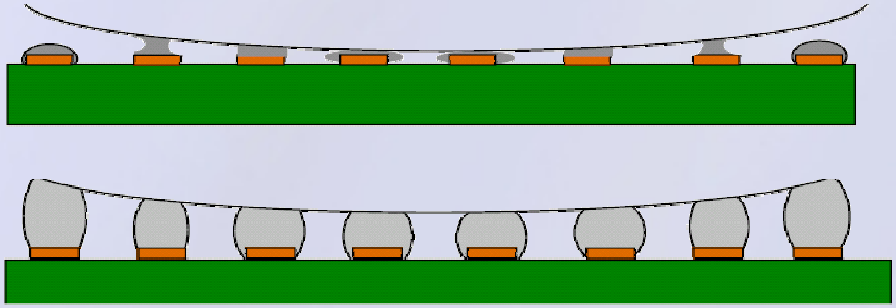
PLEXUS
The Product Realization Company

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)

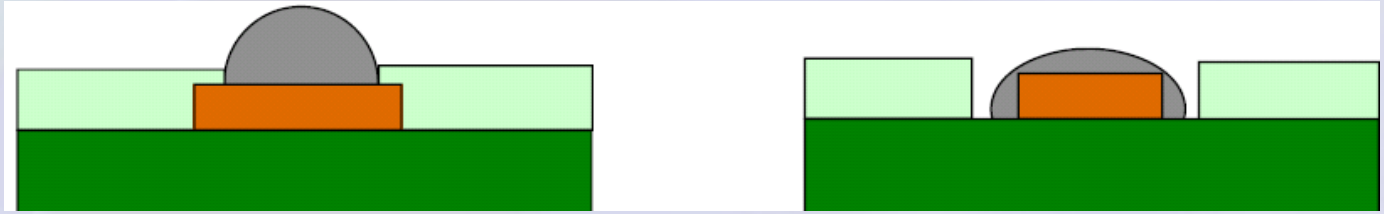


LGA Package

BGA Package

PLEXUS
The Product Realization Company

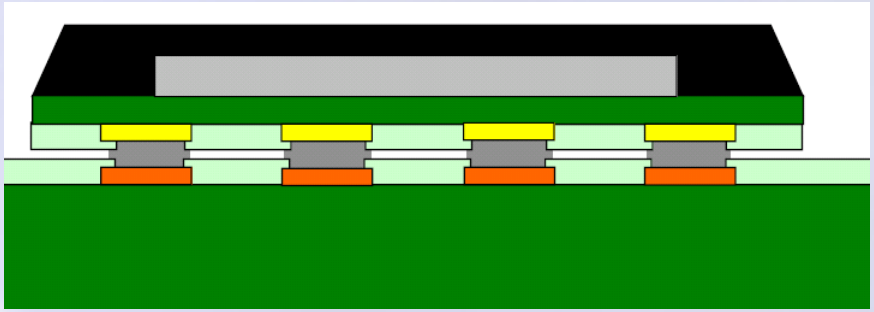
LGA Pad Design



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.

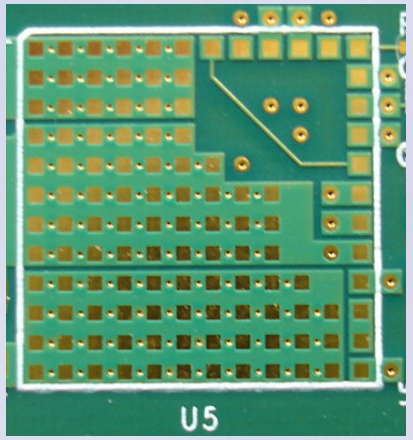


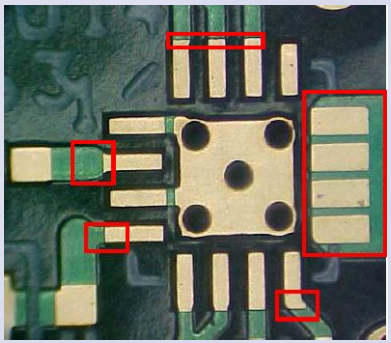
PLEXUS
The Product Realization Company

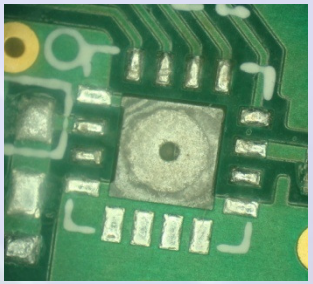
Component Issues - LGA & QFN

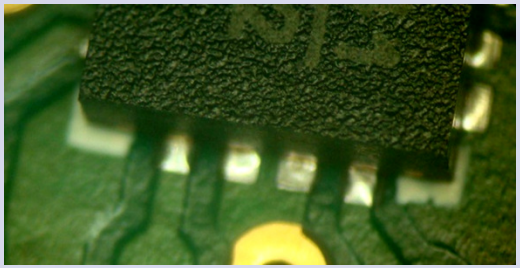
Potential Issues:

- 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



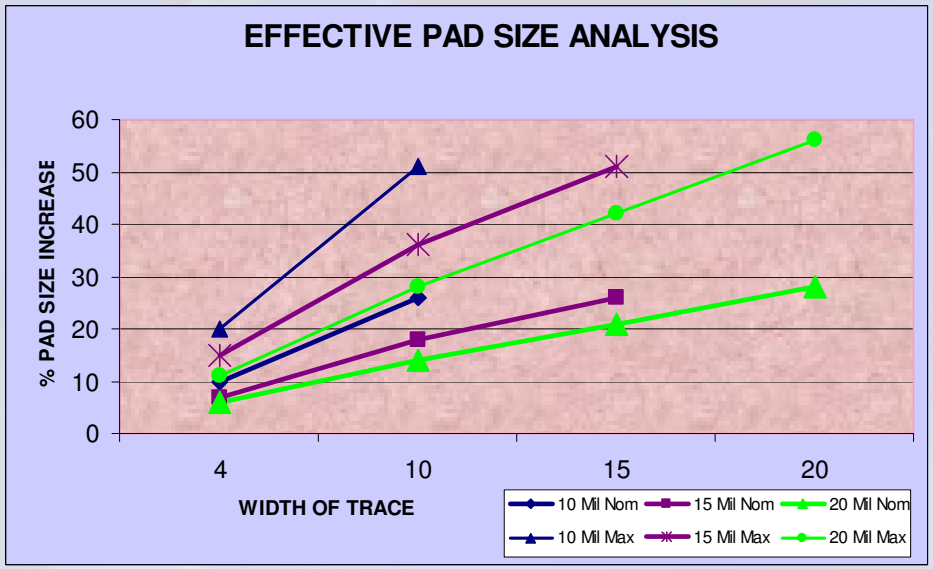


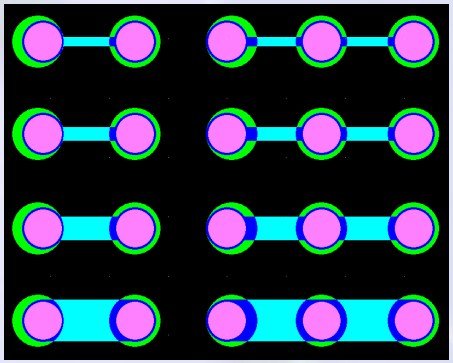


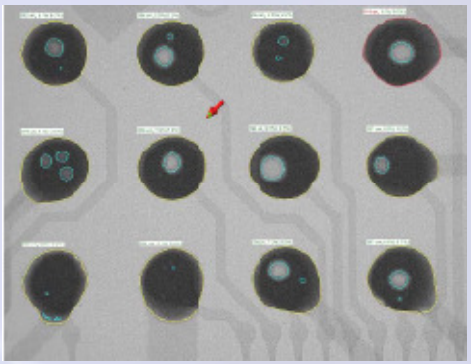


PLEXUS
The Product Realization Company

Trace Routing Impacts Solder Joint







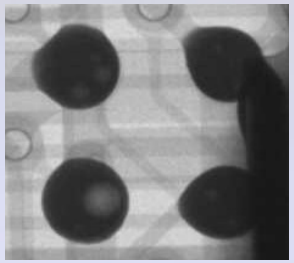
Increased Mounting Pad Size Affected By:

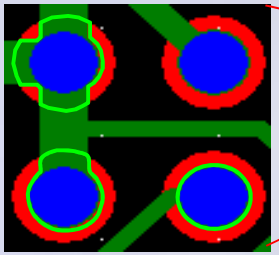
- Number Of Trace Connections To Each Pad
- Width Of Trace Connections To Each Pad
- Size of Pad
  - Small Pads Have Less Margin
- Uniformity Of Trace Egress Direction
  - Some Package Types Are More Sensitive Than Others
- Uniformity Of Trace Sizes

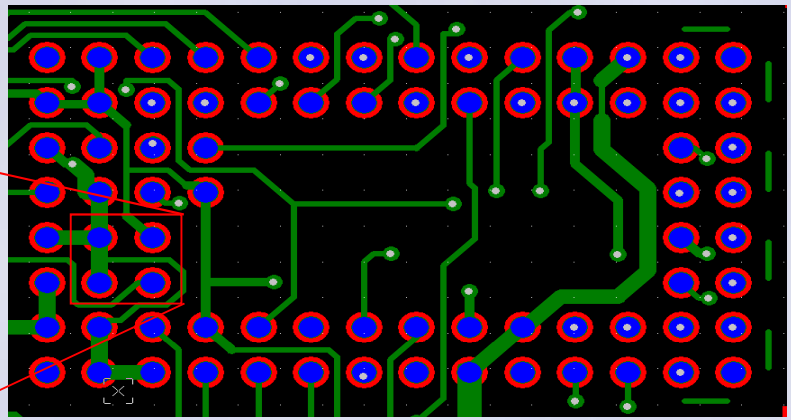
PLEXUS
The Product Realization Company

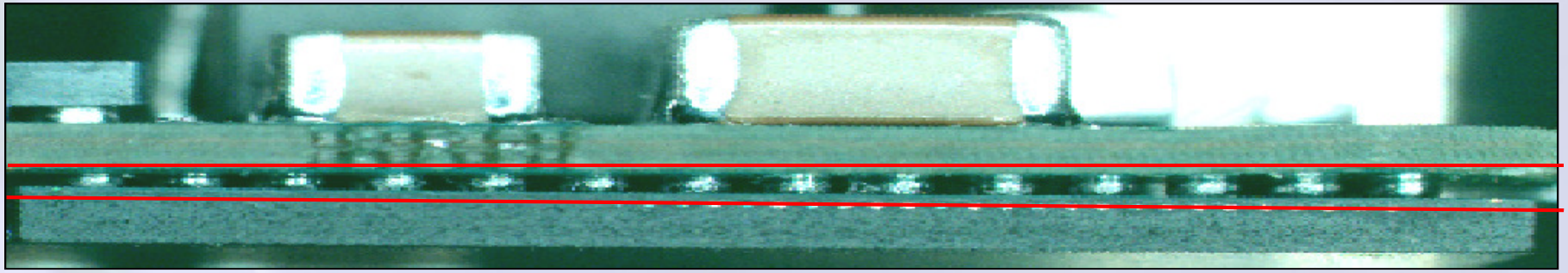
Trace Routing Impacts Solder Joint

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





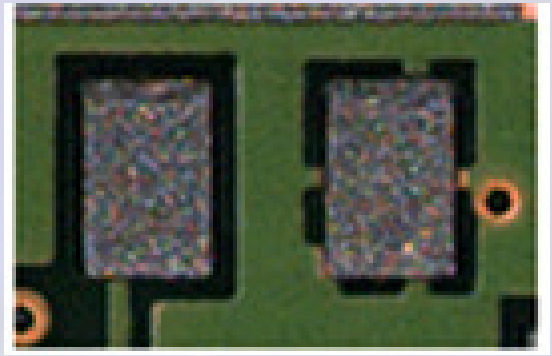


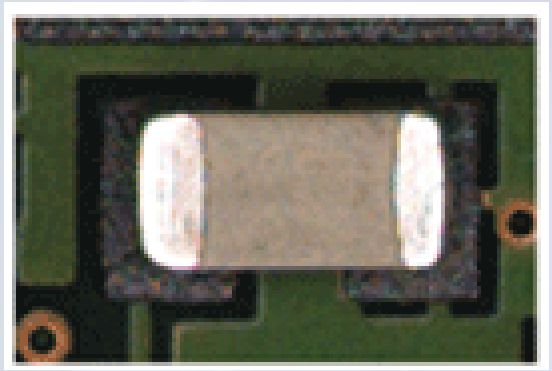


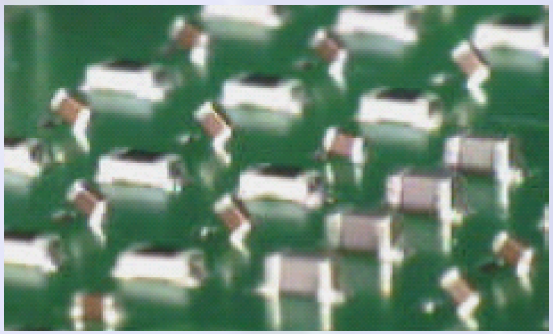
Concentrations Of Design Variability Can Create:
Solder Bridge, Open Connection, Insufficient Solder, Tilted Components

PLEXUS
The Product Realization Company

Component Pad - Thermal Imbalance







- Multiple Trace Connections
  - Number Of Trace Connections Per Pad
  - Uniformity Across All Pads On Single Component
- Solder Mask Defined Pad
- Increased Soldering Defects
  - Delayed Reflow Across SMT Components
    - Tombstone Components
    - “Ball in Socket” Area Array Component

PLEXUS
The Product Realization Company

Component Pad - Thermal Imbalance

- Small Passive Component Pad Design
  - Mixed Solder Mask Defined
    - Multiple/Large Trace
    - Exposed Plane
  - Smaller Component Package, The Greater The Impact

PLEXUS
The Product Realization Company

Thick Copper Design

A CTE Mismatch Between Copper And FR-4 Exists

- Copper = 18ppm
- FR-4 = 15-16ppm
- As Copper Gets Thicker, It Has Greater And Greater Influence Over Post Lamination Dimensions Of The Resulting Substrate

Rolled , Annealed Copper Reacts Differently Than ED Copper

- Design Influences On Net Shrinkage Become Far More Significant Standard Scaling Factors (That Compensate Net Shrinkage Effects) Do Not Apply!

More Resin Is Required To “Fill” Etched Out Areas

- “In-plane” CTE Increases With % Resin Content
- “Another Complication To Net Shrinkage Prediction”
- Resin Rich Package = Higher Z-axis CTE = Greater Sensitivity To Thermal Excursions, Assembly In Particular.
- Heavy Copper Substrates Generally Require More Heat Input To Assemble
- Reliability!

PLEXUS
The Product Realization Company

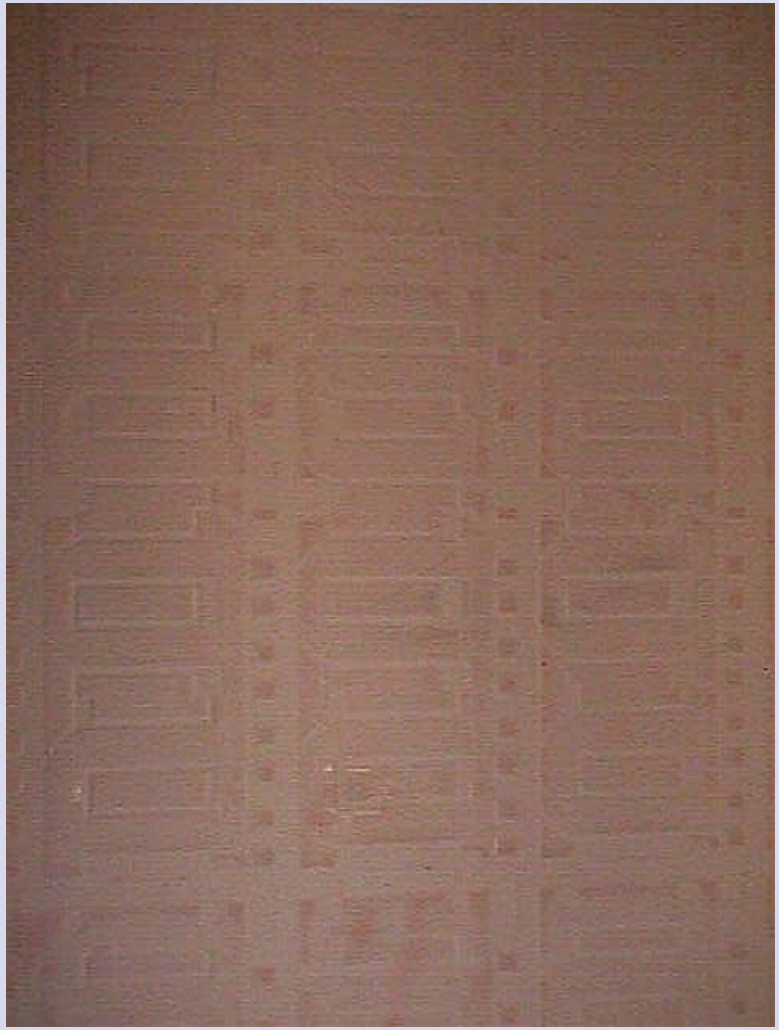
Thick Copper Design

- Heavy Copper, Embedded In Glass Epoxy, Can Result In Nonplanar Conditions On The Surface Of The Substrate
- The Heavier The Copper, The Worse The Effect
- In Multi-layer Applications, Local Areas With “Stacked” Copper, Beside Areas That Are Etched Out, Can Result In Extreme Non-planarities
- PCB Fabrication Industry Typically Refers To This As “Image Transfer”

PLEXUS
The Product Realization Company

Thick Copper Design

Lack Of Planarity
- Reduces The Effective Resolution Of The Fabrication Process, Limiting Feature Size

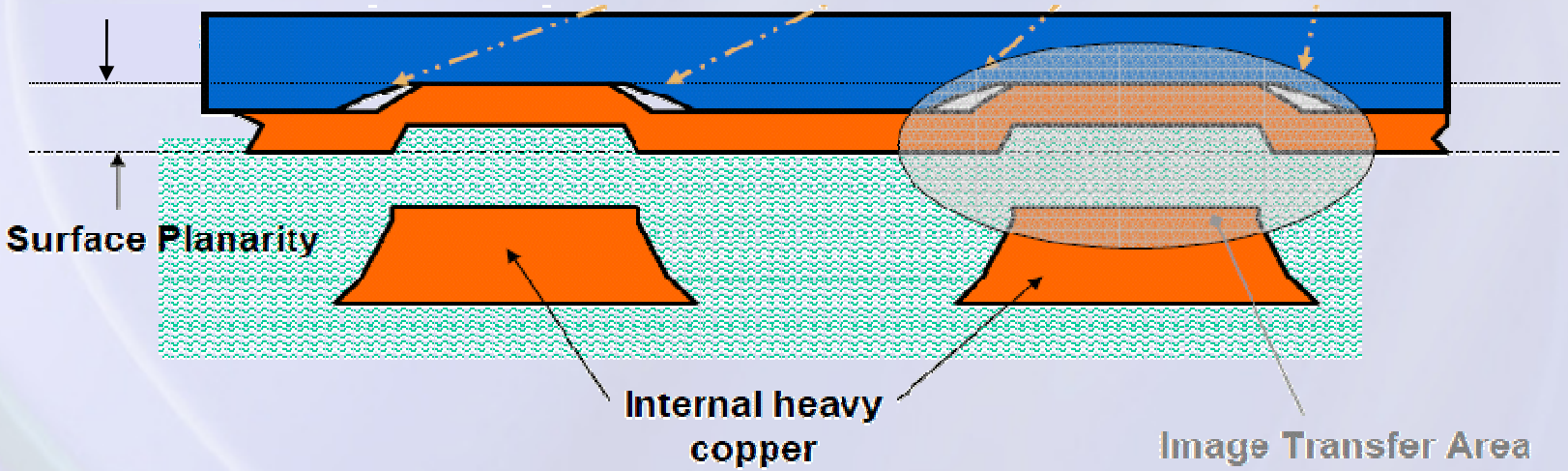


PLEXUS
The Product Realization Company

Heavy Copper Lamination

- Imaging Resist Thickness Is 1.3 - 2mils. At Some Point, Image Transfer Interferes With The Imaging Resists Ability To “Conform” To The Lack Of Planarity On The Surface Of The Substrate.

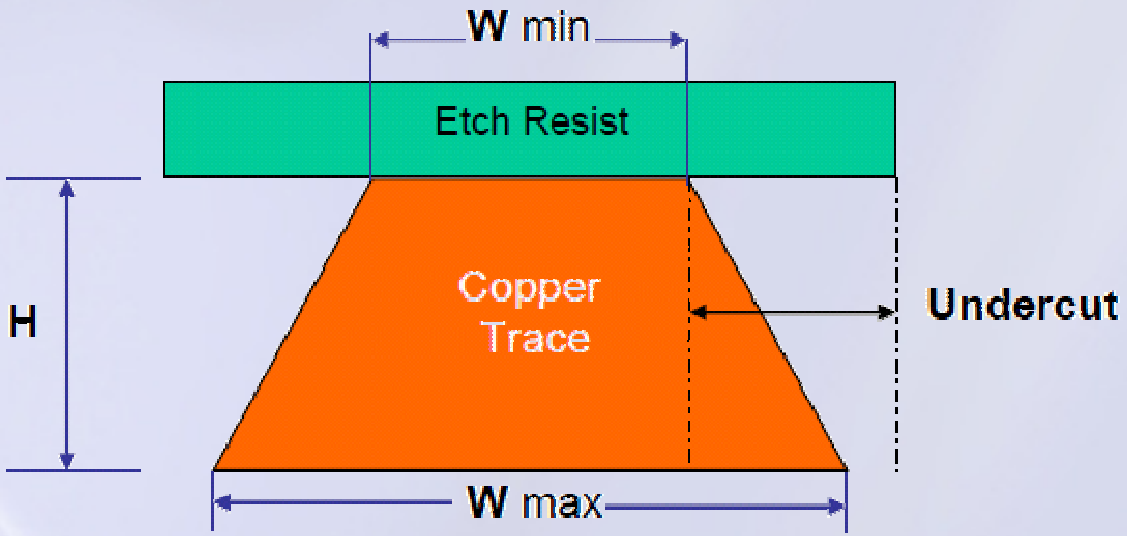
- This Lack Of Planarity Also Reduces The Effective Resolution Of The System, Limiting Feature



PLEXUS
The Product Realization Company

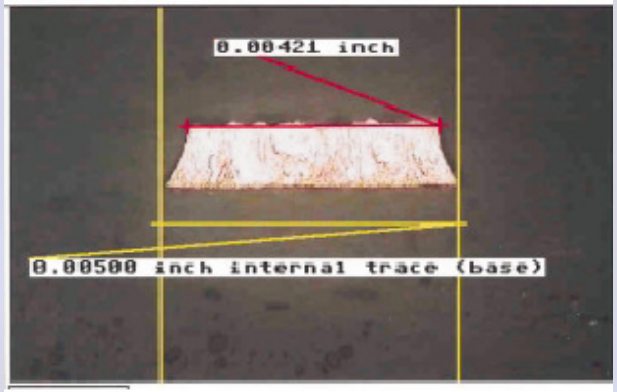
Copper Etch Process

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

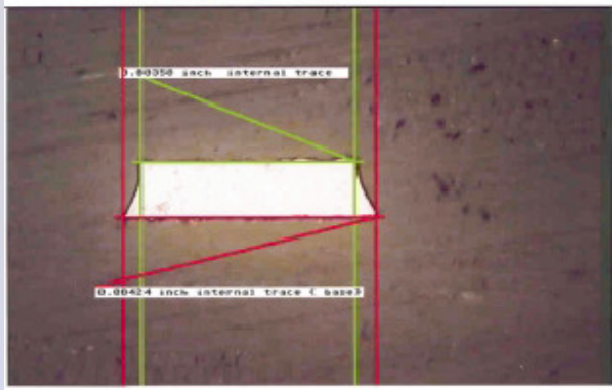


PLEXUS
The Product Realization Company

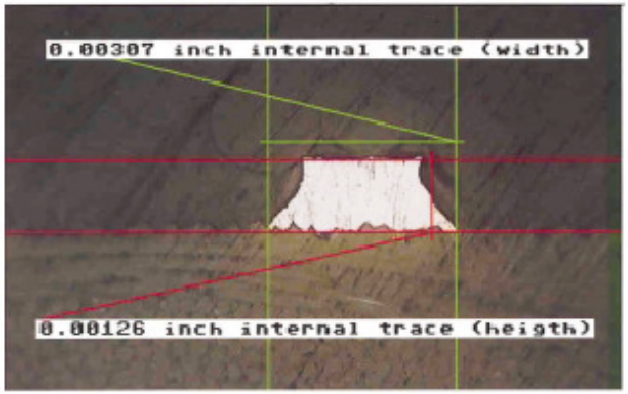
Etched Trace Design Considerations



0.005” Trace 1 oz copper



0.004” Trace 1 oz copper



0.003” Trace 1 oz copper

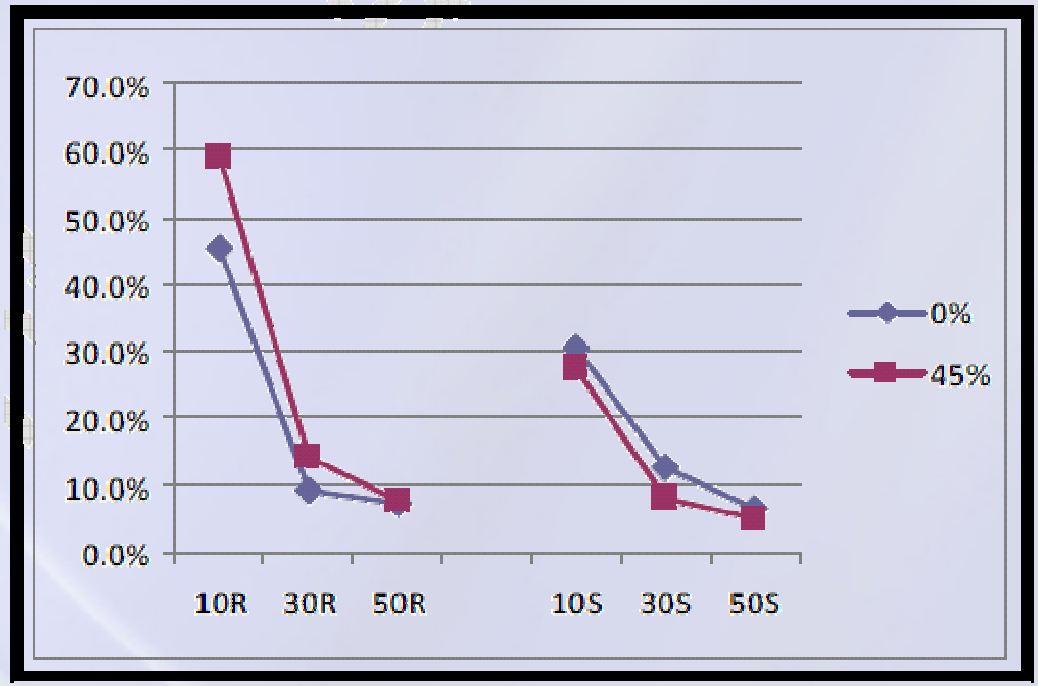
- Thinner Copper Produces More Precise Geometries on Lines Less Than 0.005” in Width
- Actual Conductor Shape is Close to a Trapazoid
- Copper Thickness is Slightly Reduced After PCB Processing

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.



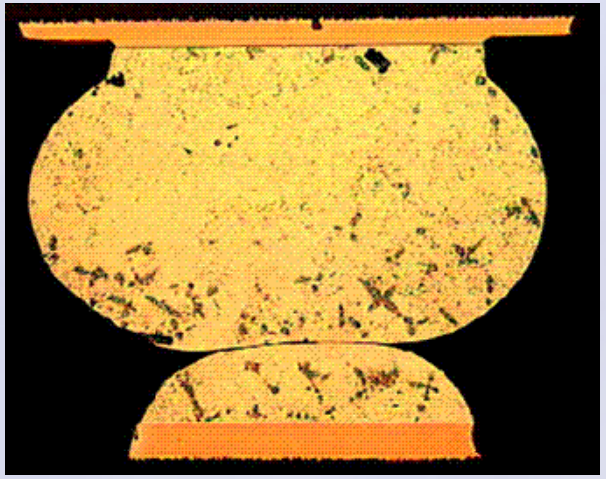
Percentage Reduction by Pad Size

PLEXUS
The Product Realization Company

Warpage & Thermal Profile Issues

May Require Change In Production Process

- Tooling To Bridge Warpage Gap.
  (Increased Solder Paste Volume Application, etc. Reflow Pallets For Board Support)
- Reflow Profile To Bridge Component Warpage Gap.
  (Decreased Thermal Change Rate And Delta T Vertically In Component Package – Reduce Surface To Cooler Location Temperature Delta - TCE Induced Warpage)
- 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)

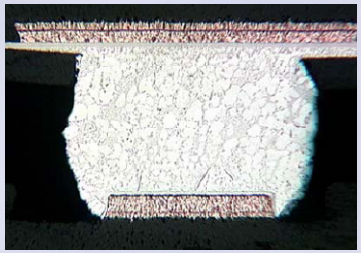


Large ΔT across Board

PLEXUS
The Product Realization Company

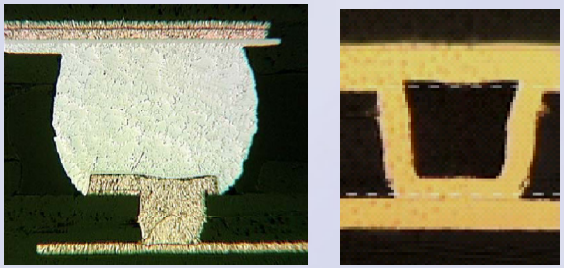
Micro-Via in Pad

- 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



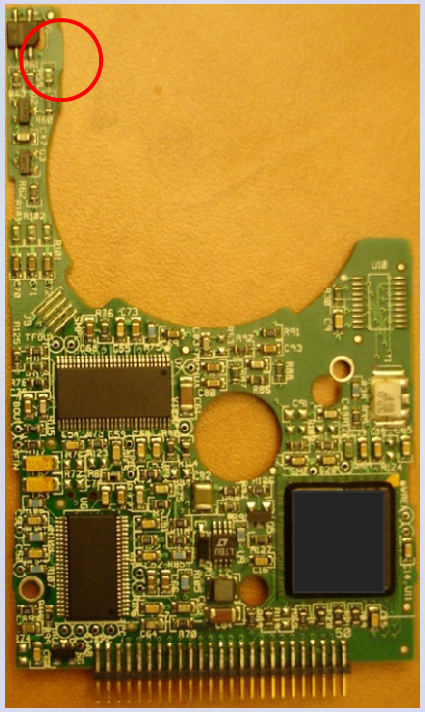
Plated Closed & Filled Via

PLEXUS
The Product Realization Company

PCB Tolerances

Artwork Feature Positional Tolerances Increase

- Inner/Outer Layer Shrinkage (Technology / Material Dependant)
- Some Materials Do Not Shrink Uniformly
- PCB Size – TCE Impact On Dimensional Measurements
- Fabrication & Depanelization Material Movement / PCB Fabrication Tolerances Artwork Registration
  - As Fabricated
  - Pre-routing of Array
  - After Depanelization

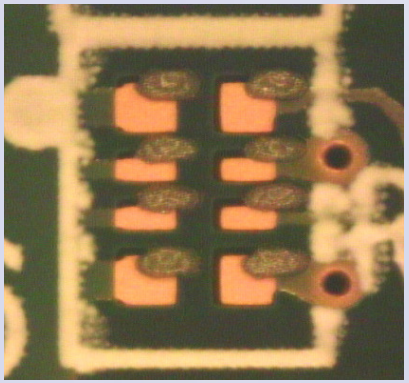


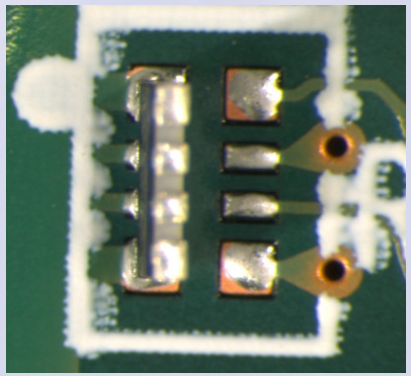
PLEXUS
The Product Realization Company

Stencil Tolerances

Artwork Feature Positional Tolerances Increase

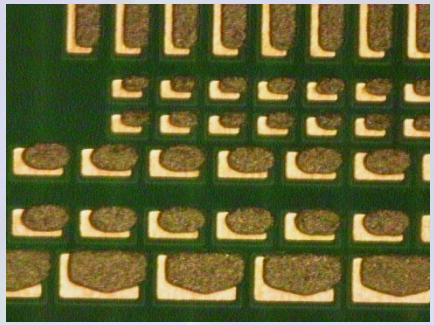
- Fabrication Tolerances Artwork Registration
  - Etched Feature Position
  - Etched Feature Size
  - Etched Feature Quality
  - Etched Feature Directional Etch
- Stencil Print Directional Compensation Orientation



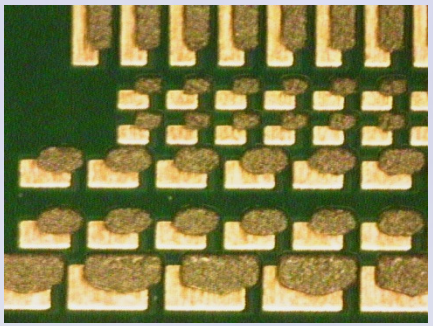


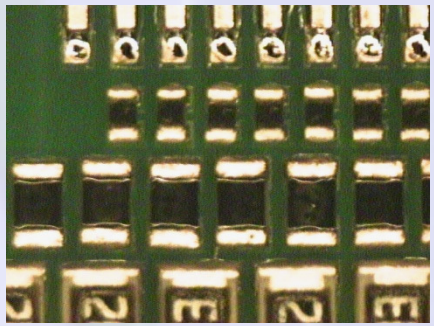
PLEXUS
The Product Realization Company

Stencil To PCB Alignment

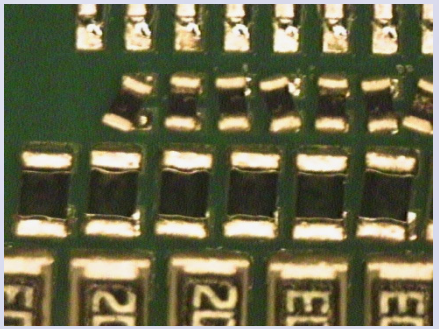


- Smaller Components Decrease Total PCB & Assembly Process Tolerance
- Minor Misalignment Can Impact Process Yields





3 mil



6 mil

PLEXUS
The Product Realization Company

Offset Paste - Normal Placement

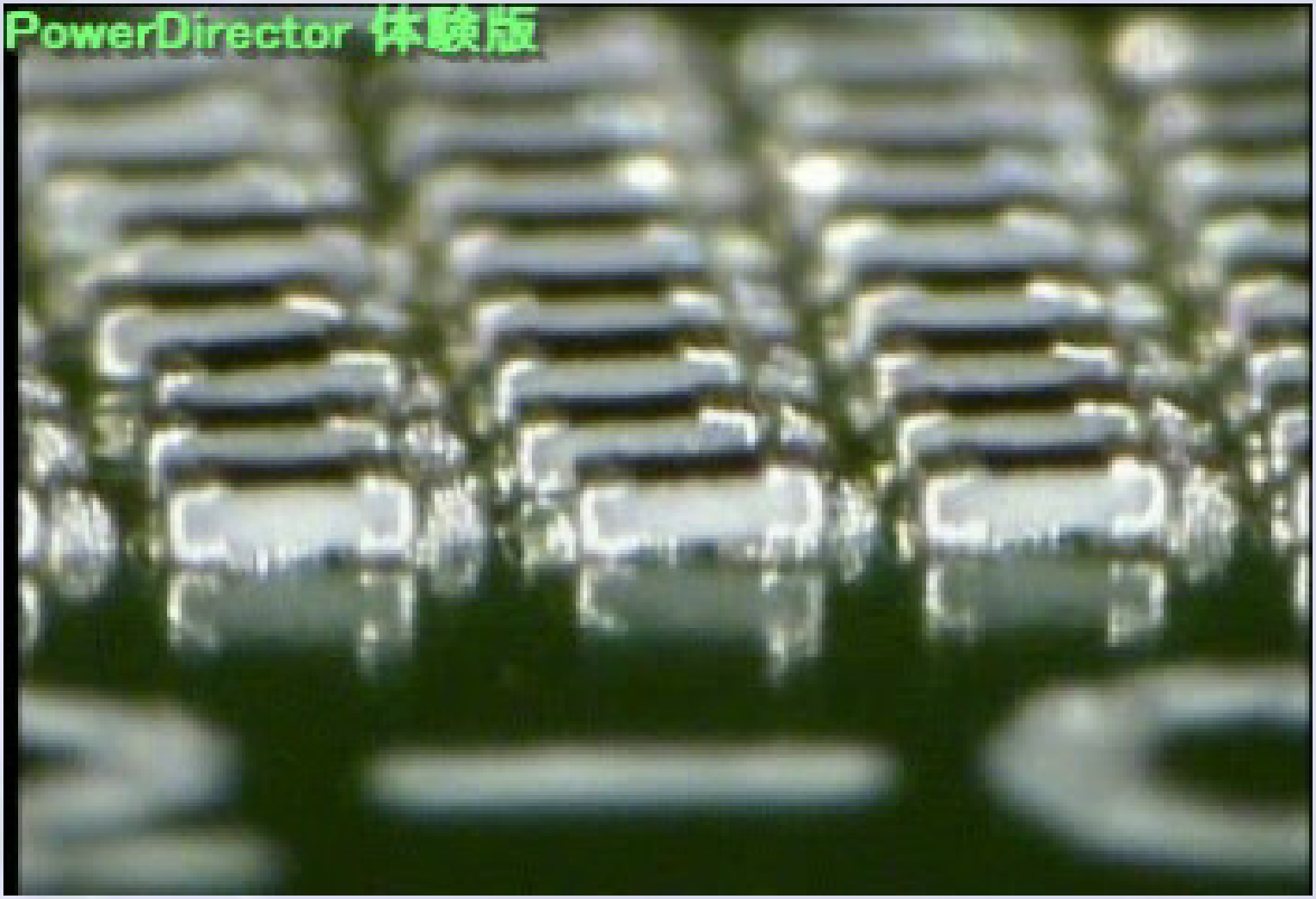
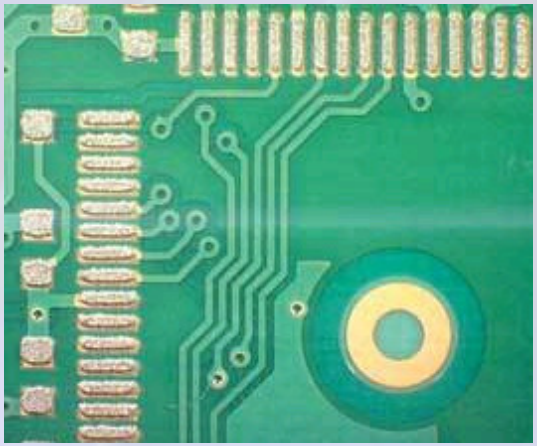


Photo Courtesy of Juki Automation

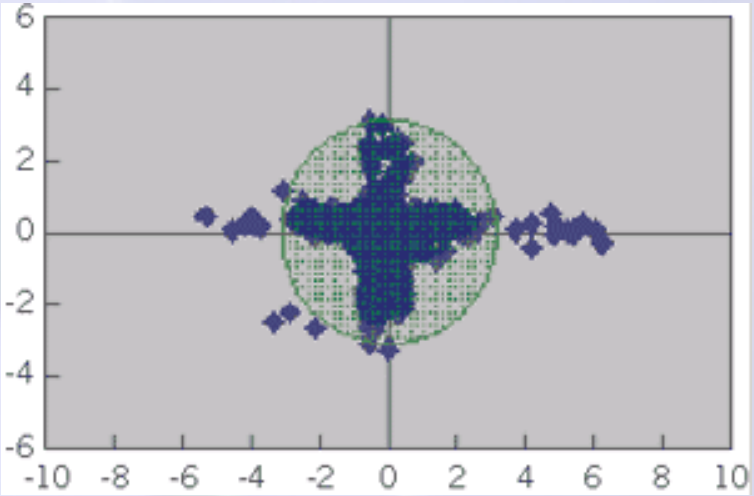
PLEXUS
The Product Realization Company

Match Tooling Design To Parts

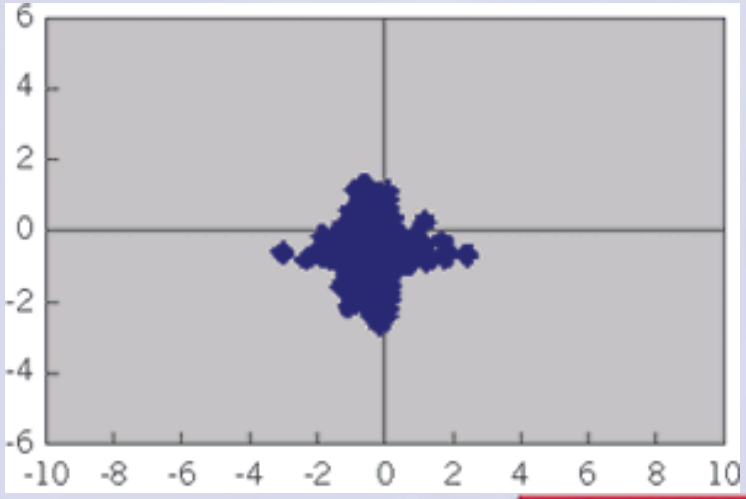


- Normal Manufacturing Process Variability May Exceed Allowable Assembly Process Tolerance For High Yield, Reliable Assembly
- Matched Tooling (Stencils) To Materials (PCB) May Be Required

PCB to Stencil Pad Positional Deviation Measurements No Adjustments



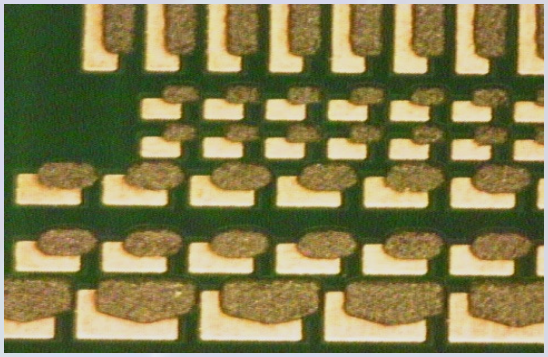
PCB to Stencil Pad Positional Deviation Measurements With Scaling Adjustments

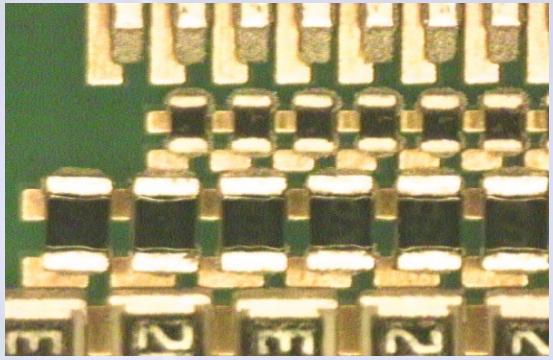


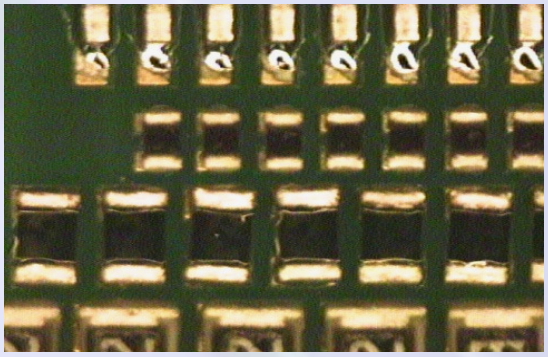
"Using Stencil: Design to Reduce SMT Defects", SMT, April 2006

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

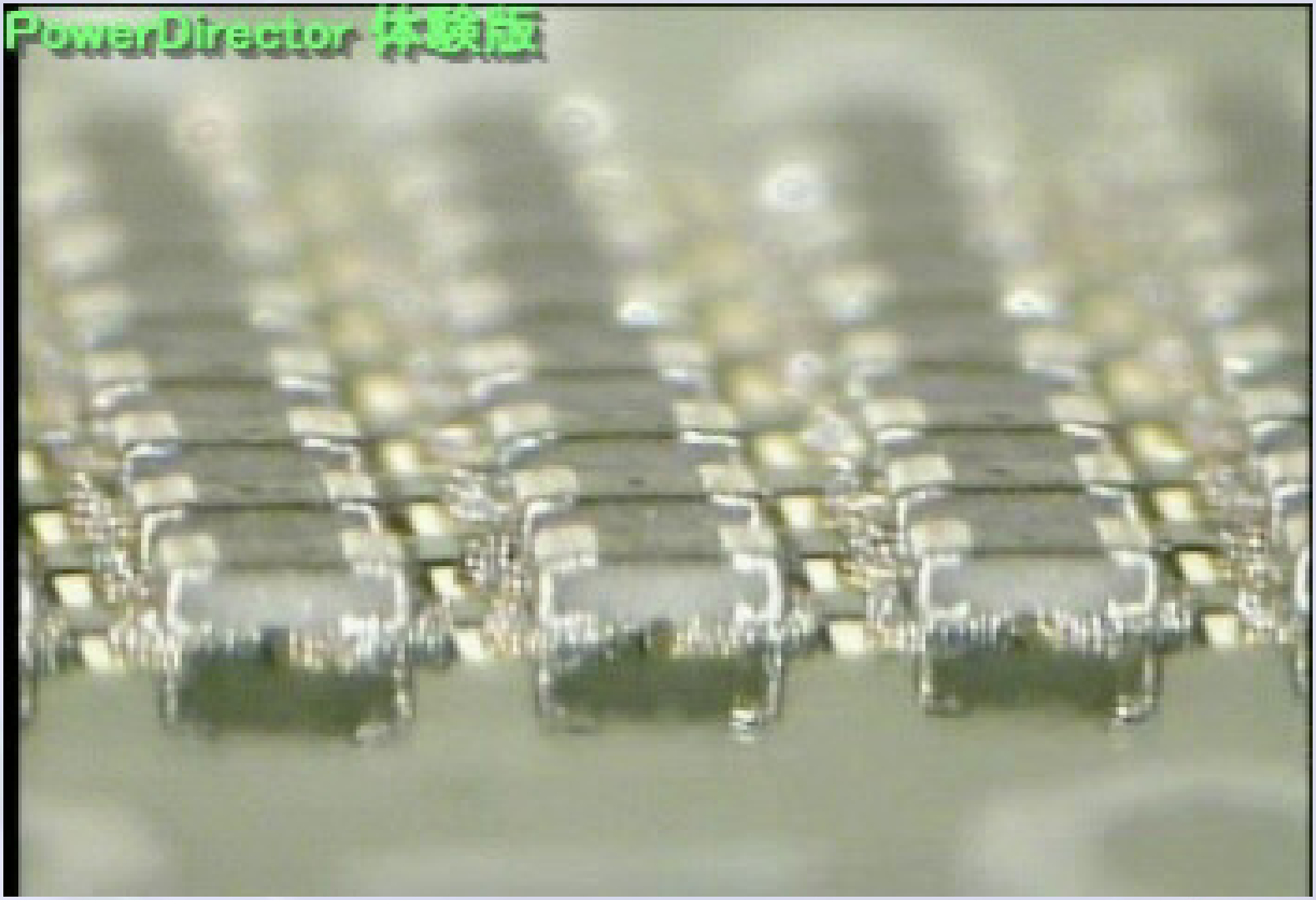
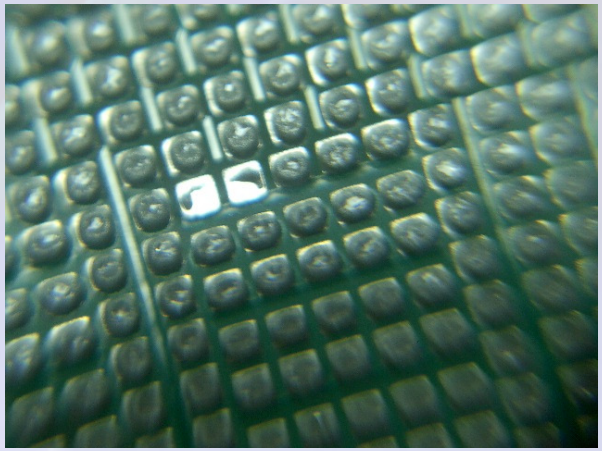


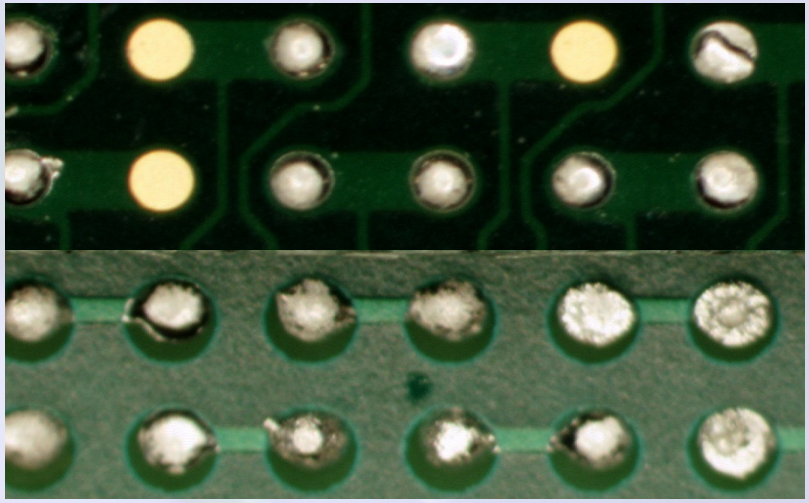
Photo Courtesy of Juki Automation

PLEXUS
The Product Realization Company

Solder Paste Printing Volume



Air Bubble In Solder Paste Bead



Open/Unwetted LGA Connection

- 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

PLEXUS
The Product Realization Company

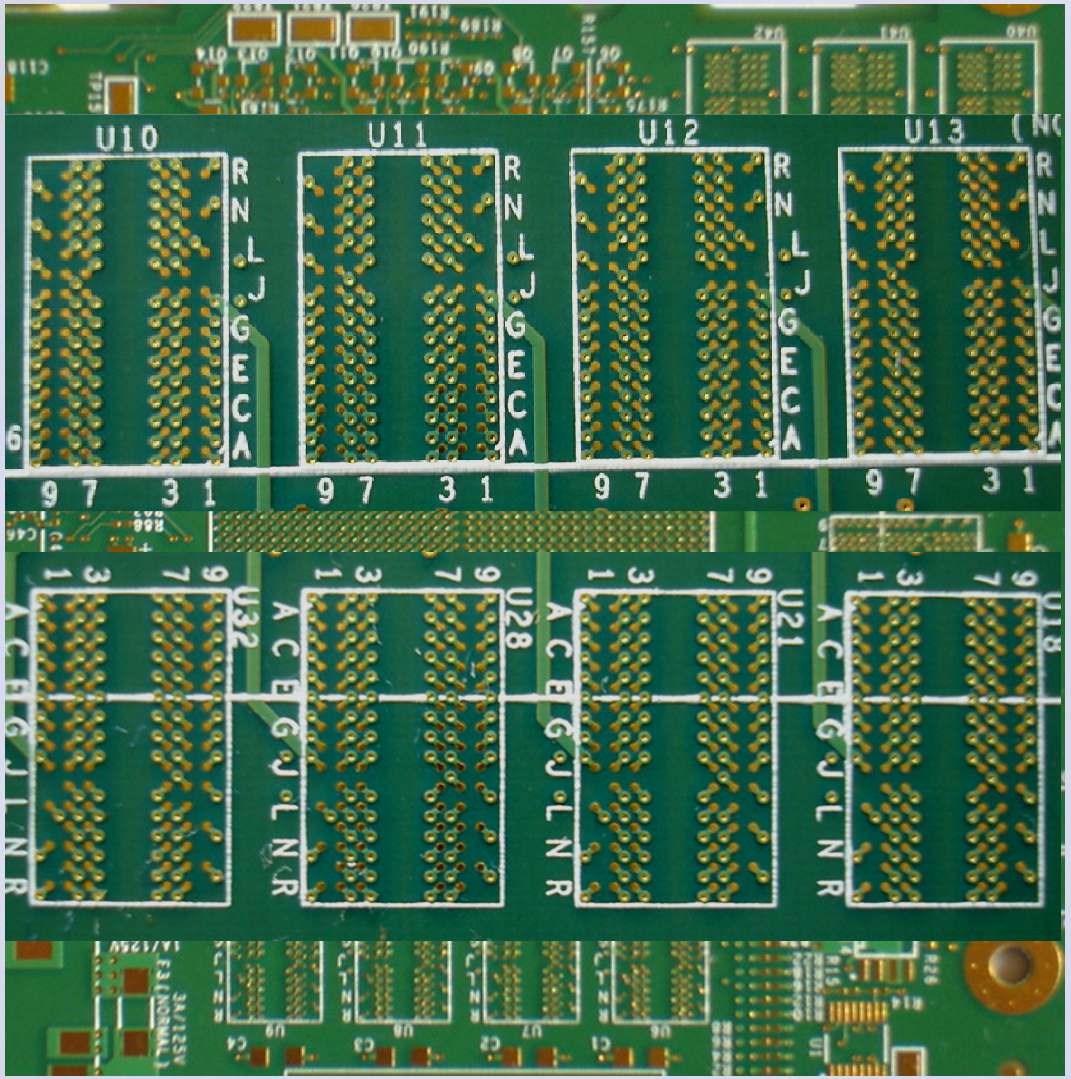
Silk Screen Design

Low Component Stand-off Height

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

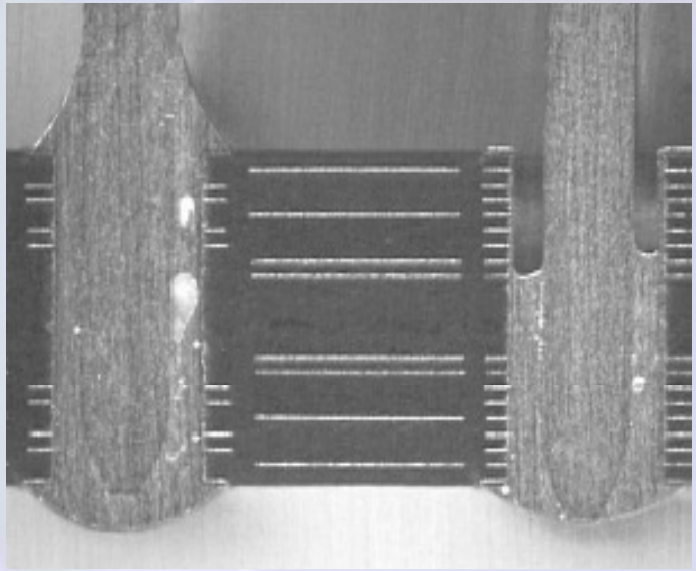
Component Types

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



PLEXUS
The Product Realization Company

Internal PCB Impacts



Number Of Layer Connections to Plated Through Hole

- Increased Number Of Layer Connections Increases Thermal Mass Of Plated Through Hole
- Increased Number Of Plane Layer Connections Greatly Increases Thermal Mass Of Plated Through Hole
- Increase Thermal Pad Isolation To Improve Solder Flow To Topside

Issues Include:

- PTH Hole Fill

PLEXUS
The Product Realization Company

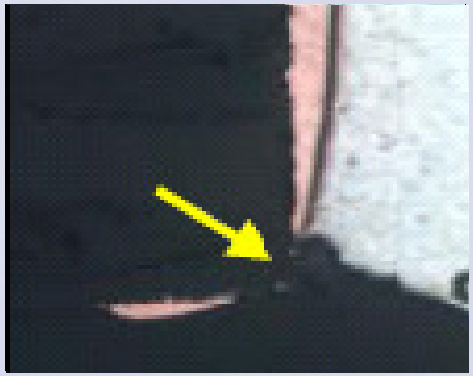
What To Do?

- Increase Solder Temperature?

250°C

Lead Free Solder Issue

Higher Solder Temperatures Or Increased Solder Dwell Times Create Problems With Pads On Solder Side

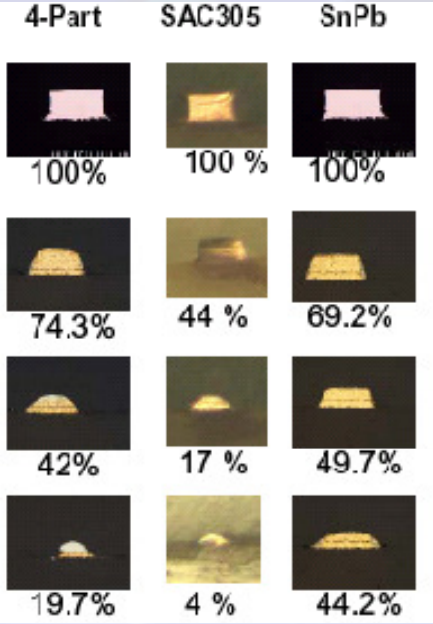


* Dr. S. Zweiger, Solectron GMBH, Productronica Green Day, November 2005

PLEXUS
The Product Realization Company

Wave Solder & Rework Issues

Limit Effects Of Copper Dissolution



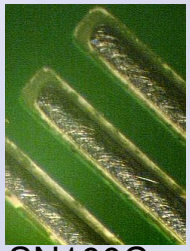
0.005” Trace Dissolution in 30 Seconds

- Use Lower Dissolution Rate Solder Alloy
  - Modified SAC Alloy
    (Sb, Ni, Zn, Ge, In, Etc)
  - Non-SAC Alloy
    (Sn/Cu/Ni, Etc)


Original

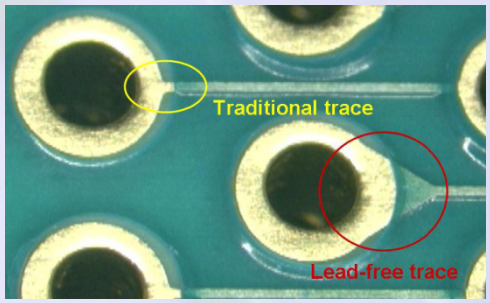

SnPb


SAC 305


SN100C

After wave soldering, 265°C solder temperature, 12 seconds contact time

- Pad Trace Connection
  - Tear Drop
  - Snow Man Connection
  - Wide Trace
  - Greater Than 0.010”

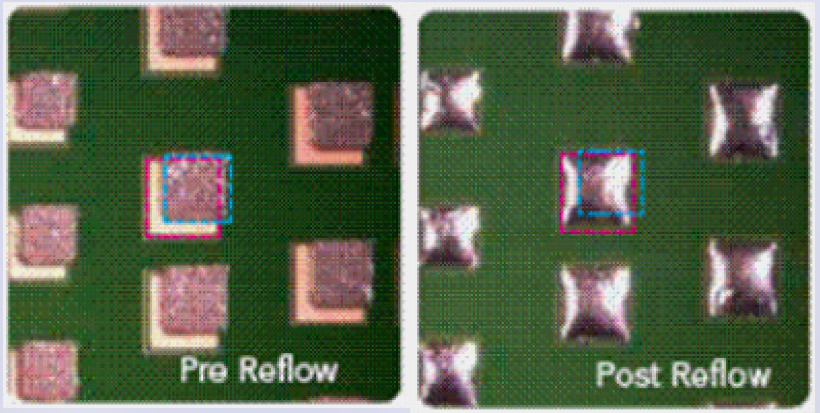


- PCB Photo Courtesy of Cookson – Alpha Metals
- Byle, Jean & Lee, “Copper Dissolution Rate in Pb-Free Soldering Fountain Systems”, SMTA-I 2006

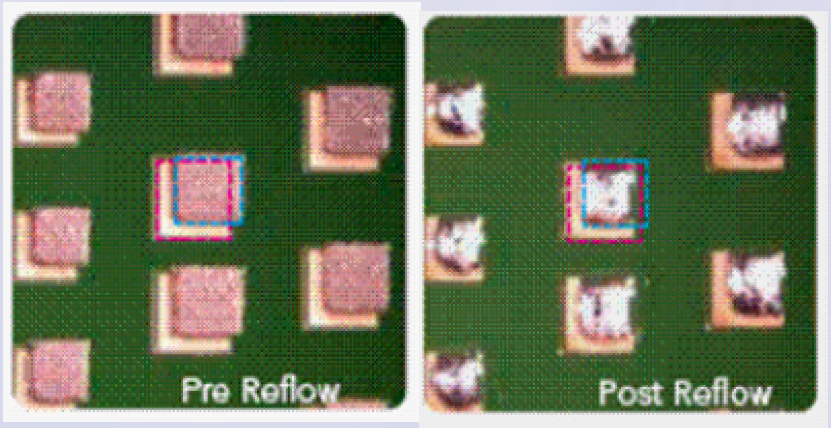
PLEXUS
The Product Realization Company

Lead Free Solder Spread

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



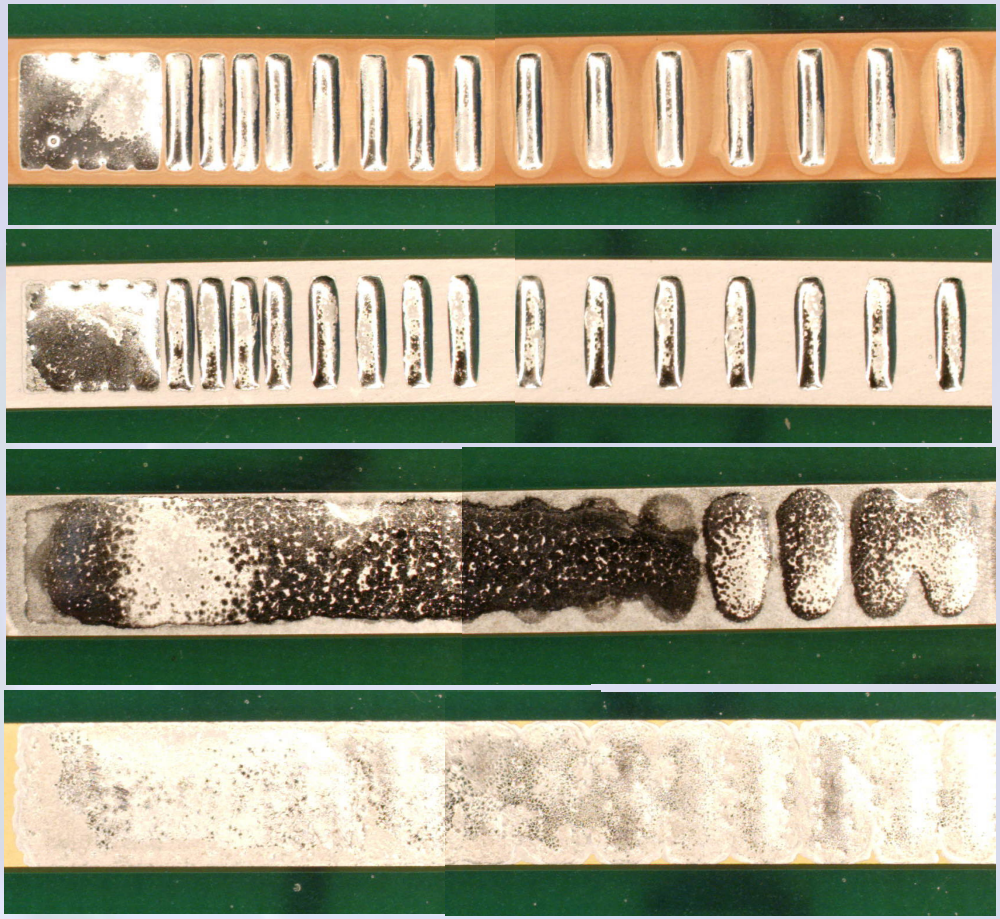
Tin-Lead Paste



Lead Free Paste

PLEXUS
The Product Realization Company

PCB Finish Vs Solder Spread



– OSP

– Immersion Silver

– Immersion Tin

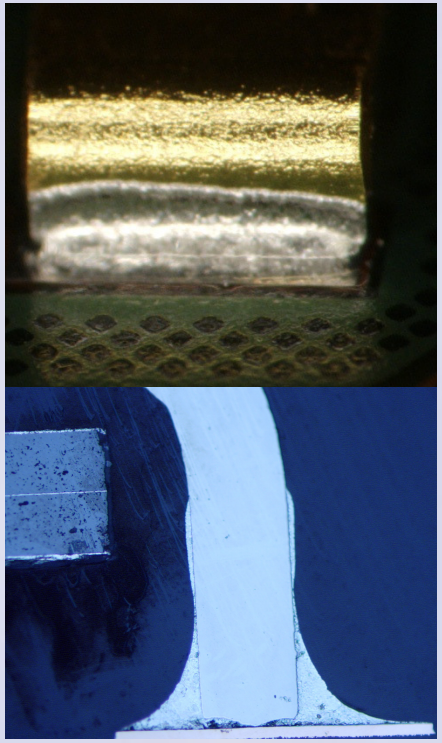
– ENIG

Amount Of Lead Free Solder Wicking Is Dependant Upon Finish

PLEXUS
The Product Realization Company

Tin-Lead Vs Lead Free Wicking

Depending Upon The Pairing Of PCB Surface Finish And Component Lead Finish, The Amount Of Solder Wicking / Spread Can Induce or Reduce Solder Defect Formation.



Tin-Lead

PLEXUS
The Product Realization Company

Defects Caused By Improper Thermal Balance

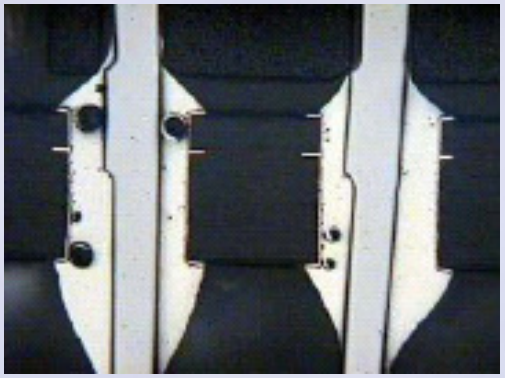
Wave Solder Process
Thermal Shock
Damage

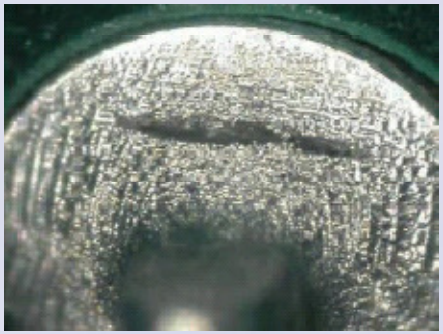
- Increase thermal pad isolation
- Balance number connections for component per each plated through hole
- Uniformity of trace size connection to component plated through hole

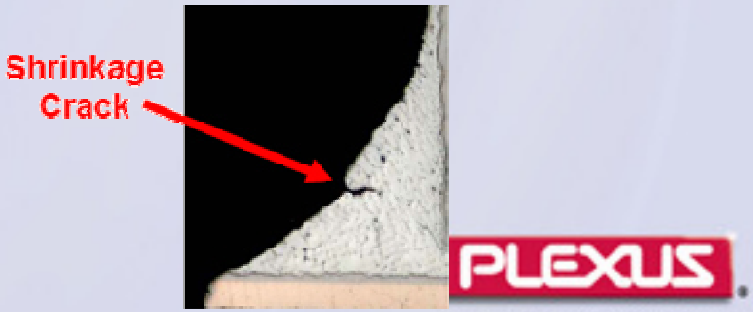
PLEXUS
The Product Realization Company

Lead To Hole Clearance

- **Lead Free Soldering**
  - Lead Clearance Minimum May Increase
  - Increasing Board Thickness May Further Increase Lead To Hole Clearance (Aspect Ratio)
  - Larger Holes Create Less Voids
- **Smaller Lead To Hole Clearance Decreases Shrinkage Holes / Hot Tear Joints**





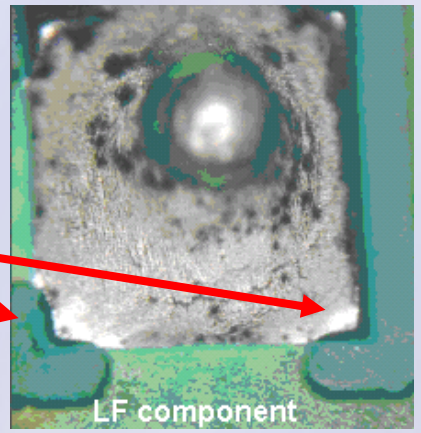


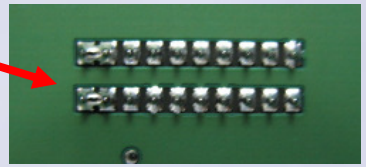
IPC-A-610D, Fig. 5.67

PLEXUS
The Product Realization Company

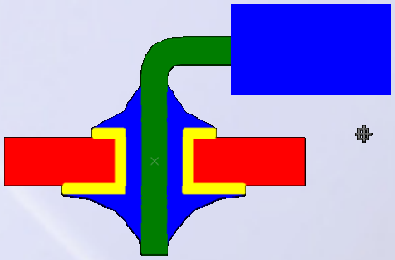
Through Hole Pad Design

- Square Pads Should Not Be Used On Solder Side
  - Increased Pad Lifting*
  - Increased Solder Defect – Bridge/Flag/Web





- Decrease Component / Top Side Pad Size**
  - Reduced Fillet Lifting



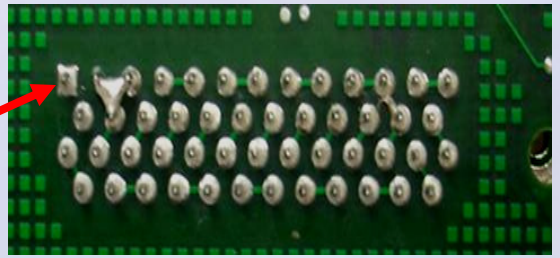
* Dr. S. Zweiger, Solectron GMBH, Productronica Green Day, November 2005

** K Puttlitz, K Stalter, "Handbook of Lead-Free Solder Technology For Microelectronic Assembly", pp 628, Fig 48

PLEXUS
The Product Realization Company

Through Hole Pad Design

- All Pads Should Be Same Shape
  - Oval or Round
  - Pin 1 square pad should not be used on solder side

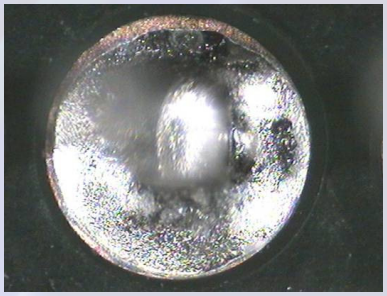


  - In some situation legend ink in between pads can help to minimize solder bridges

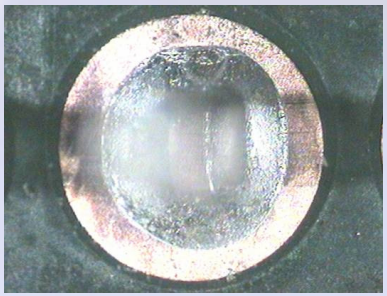
- High Density Components (< 2mm Pitch)
  - Pads Should Be Oval In Shape
  - Staggered Pad Designs Should Be Used To Enhance Solder Joint Formation On Exit Side Of Component

PLEXUS
The Product Realization Company

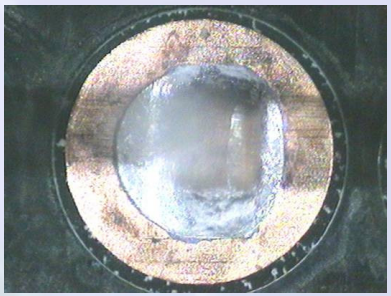
Effect of Board Thickness On Topside Fillet Formation



0.062”



0.100”



0.150”

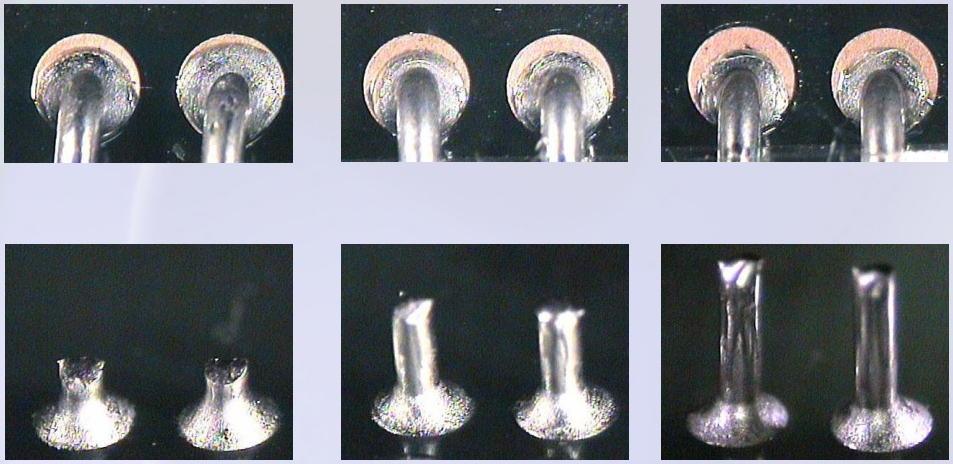
Board Thickness

- Board thickness increase thermal mass of plated through-hole
- Increase thermal pad isolation to improve solder flow to the top side
- Increase lead hole clearance (aspect ratio) to improve solder flow to top side

PLEXUS
The Product Realization Company

Effect of Lead Length on Topside Fillet Formation

1.0mm Φ hole, 0.45mm Φ lead



Lead Length: 0.040”/1mm

Lead Length: 0.080”/2mm

Lead Length: 0.120”/3mm

- Lead Length protrusion too long on solder side:
  - Only a small quantity of solder retained in soldering area, solder flow downwards along component lead
- Lead Length protrusion too short on solder side:
  - Only a small quantity of solder can transferred, and the heat transfer maybe insufficient

PLEXUS
The Product Realization Company

Solder Mask Selection

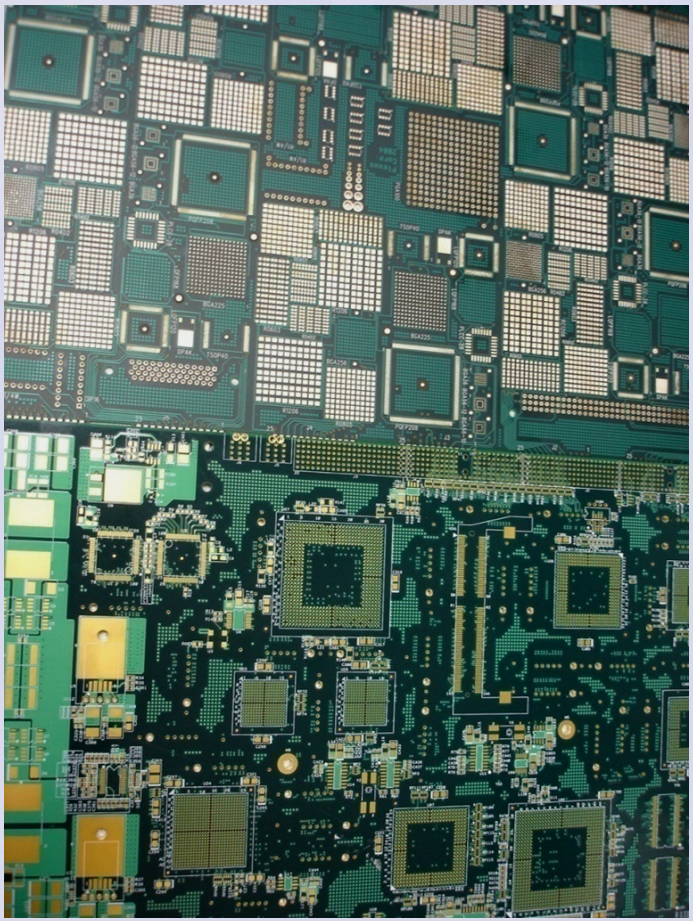
Matte Finish Solder Mask Has Increase Surface Energy To Hold Flux To The Board (More Flux For Solder Joint Formation)

Dull/Low Reflection Appearance

Glossy Finish Solder Mask Has Low Surface Energy to Flux To The Board (Less Flux For Solder Joint Formation – Solder Balls, Icicles, Flags, etc.)

Highly Reflective Appearance

Preferred

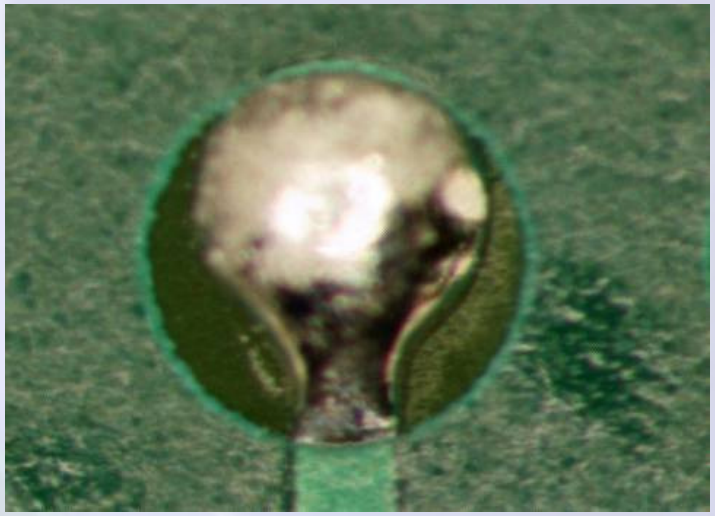


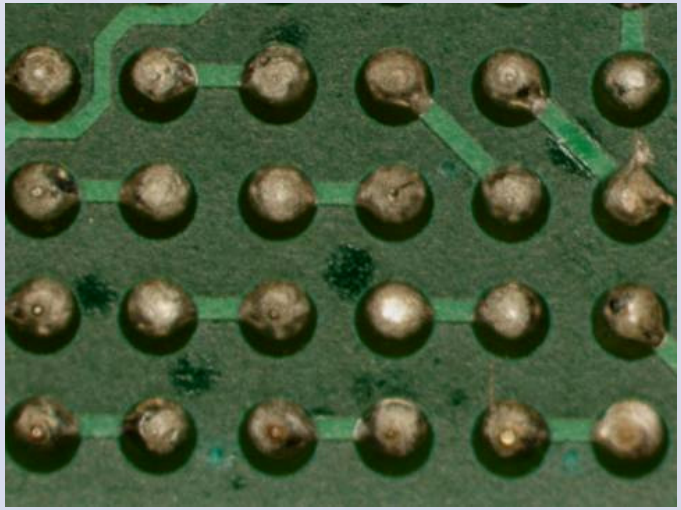
Not Preferred

PLEXUS
The Product Realization Company

Cleaning

"Remember it is not the cleanliness of the entire board that causes the electrical failure but the amount of residue between two pads on a critical circuit the define the cleanliness of the assembly."



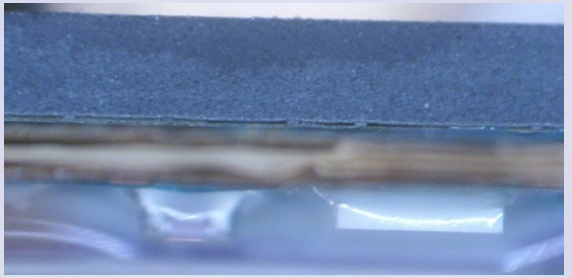


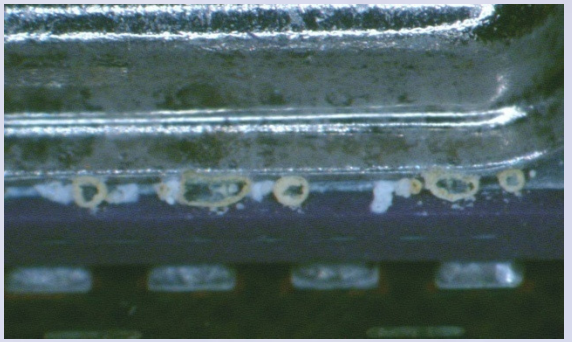
Terry Munson, President/Senior Technical Consultant, Foresite
Ask the Experts, Circuitnet.com, 7 Jan. 2008

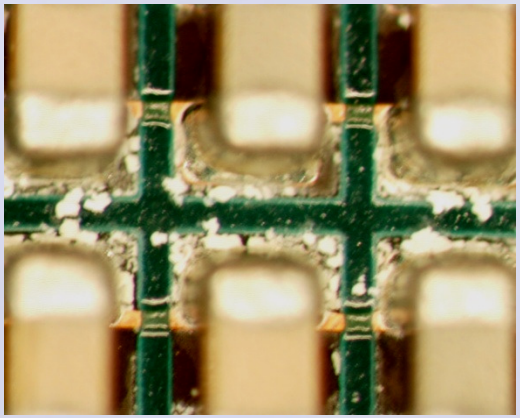
PLEXUS
The Product Realization Company

Cleaning Chemistry Compatibility

- Cleaning Chemistries Must Work With Tight Component Spacing
  - Component To Component Spacing
  - Component Stand-off Height
- Compatible With Greater Diversity Of Materials
  - PCB's
    - Substrate Materials
    - Solder Mask
    - Silk Screen Ink
    - Pad Finish
  - Component Materials
    - Package
    - Seals
    - Marking Inks
  - Solder



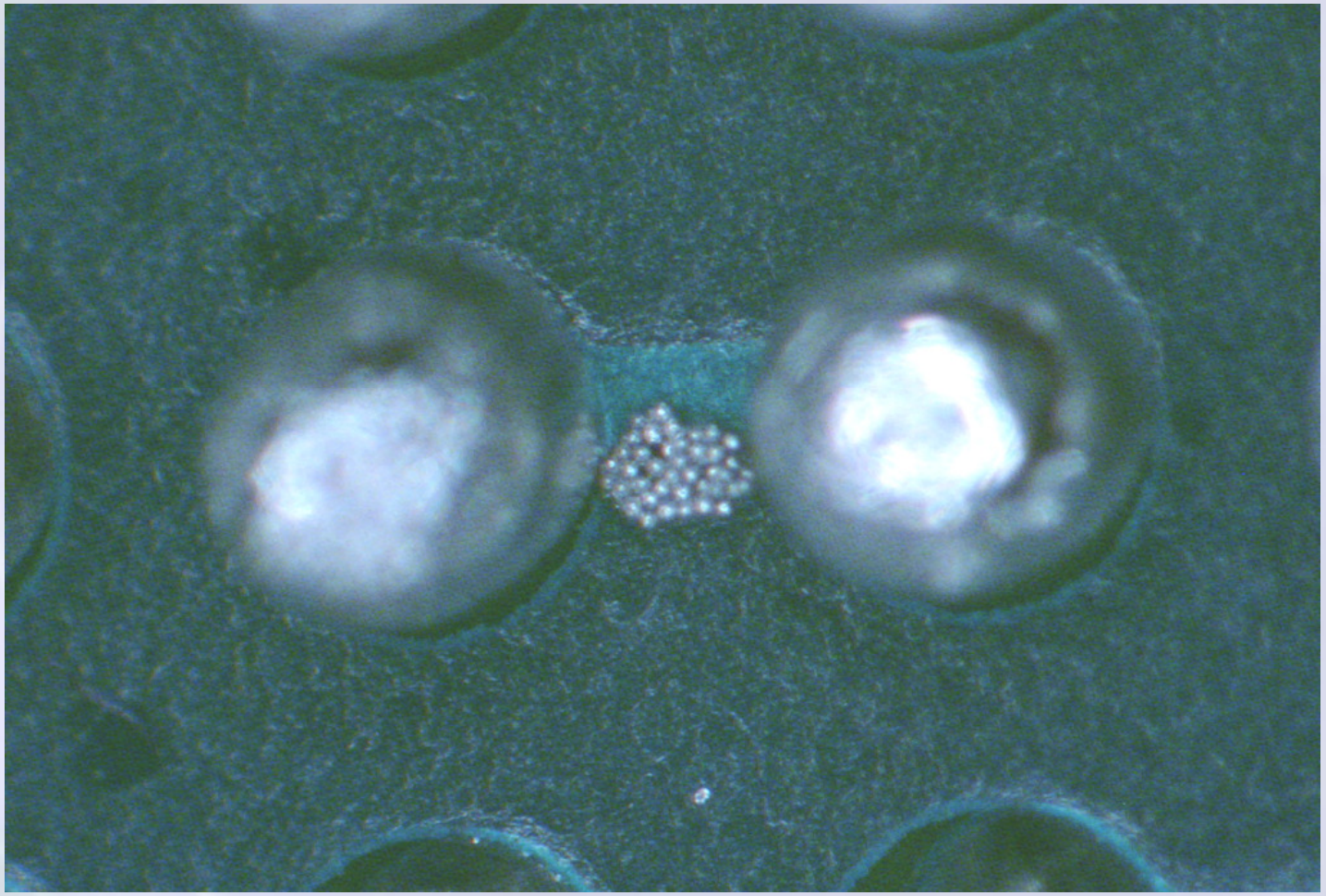




PLEXUS
The Product Realization Company

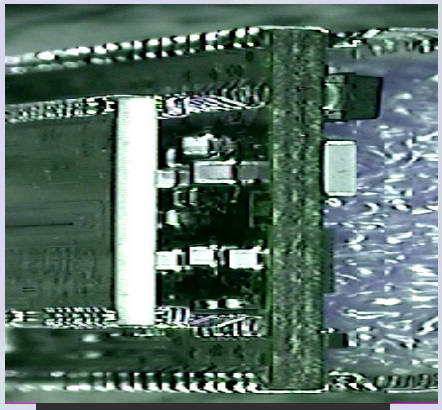
Careful – Clean May Not Be Reliable

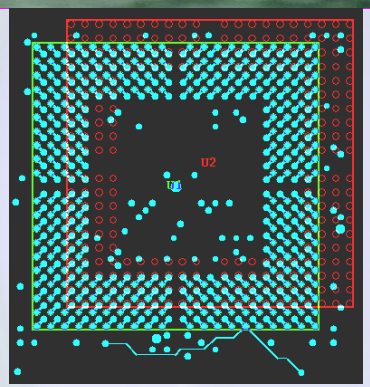
Solder Paste Bridge Between Pads on BGA After Reflow and Cleaning

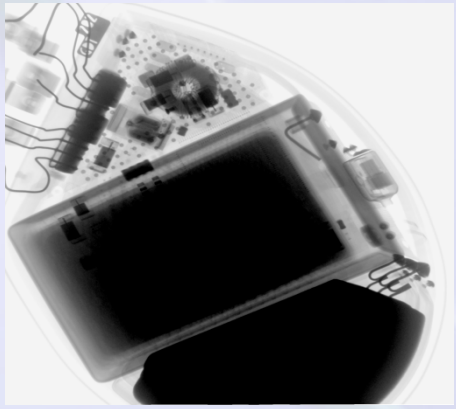


PLEXUS
The Product Realization Company

Inspectability - AOI, AXI, HVI







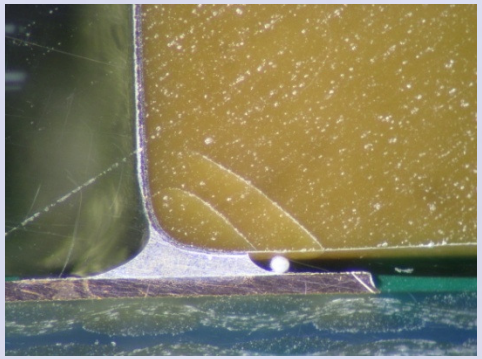
- Accessibility to Solder Joints –AOI, HVI, AXI
  - Component Spacing – Line of Sight
  - Solder Fillet Length
- Masking of Solder Joints
  - Solder Connections Under Component Body
  - Insufficient Separation Between Solder Joints
    - Stacked Component Vertical Separation
    - PCB Thickness
  - Material Density; X-ray Permeability
  - Side to Side Shadowing
- Insufficient Background Contrast
  - Component Package to PCB
  - Solder Joint to PCB

PLEXUS
The Product Realization Company

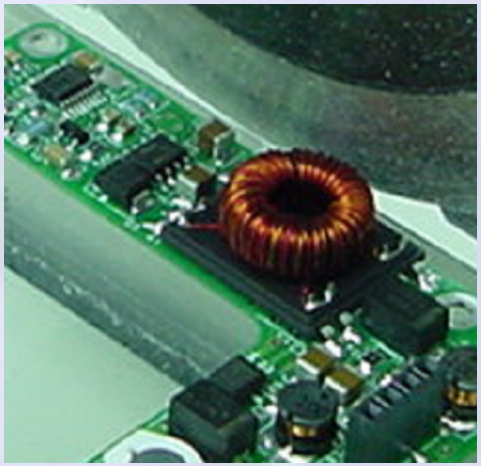
Cracked Cap Issue

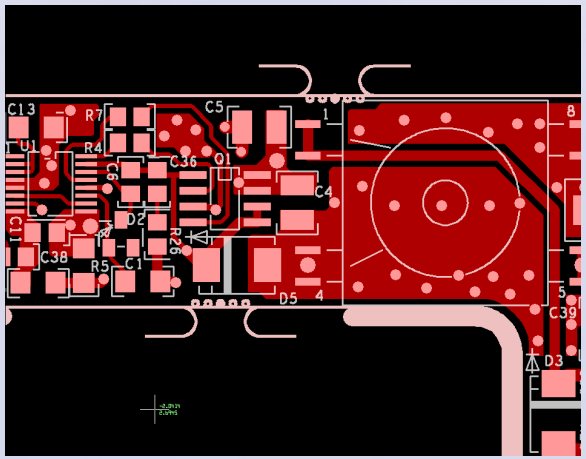
Field Return Issue

- Capacitor Functional Failure
  - Cracked Component



- What is root cause?

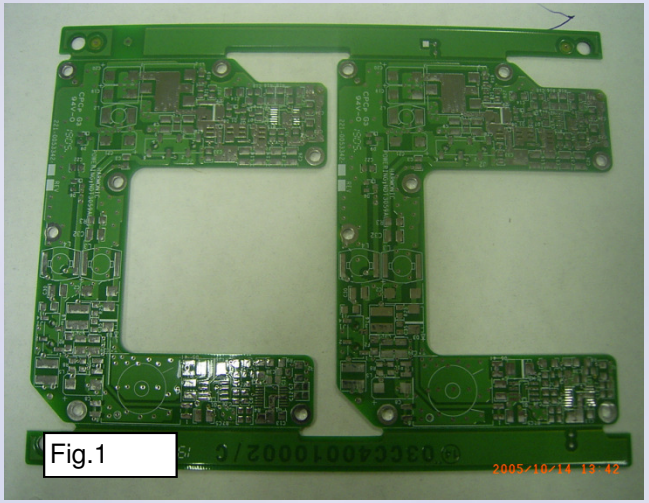




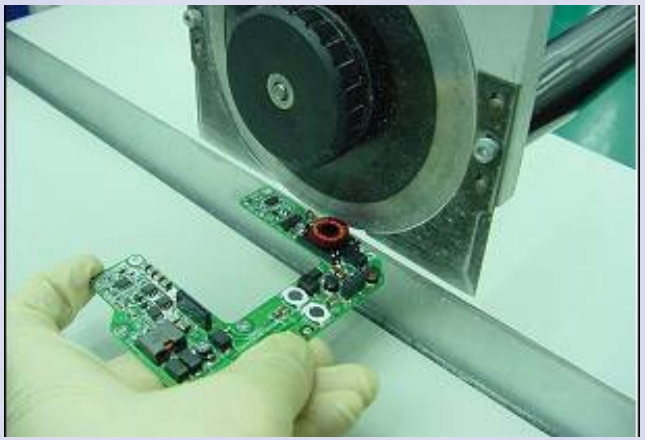
PLEXUS
The Product Realization Company

Cracked Cap Issue

- PCA Is Assembled In Panelized Form
- PCA Is Depanelized Using A V-Score Slicing System

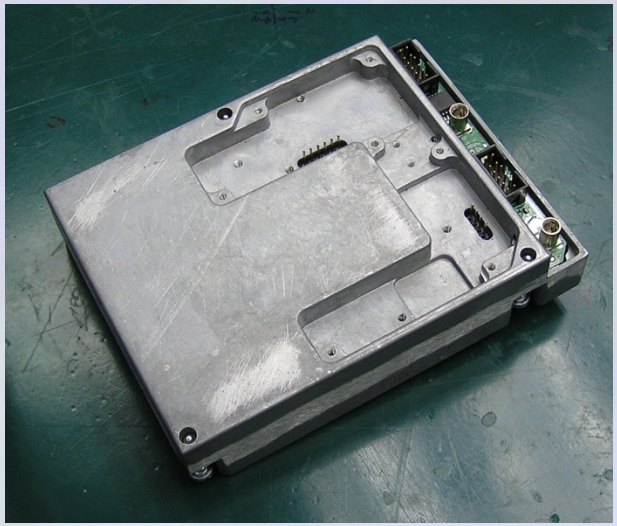


- No Indication Of Issue At ICT & Functional Test



PLEXUS
The Product Realization Company

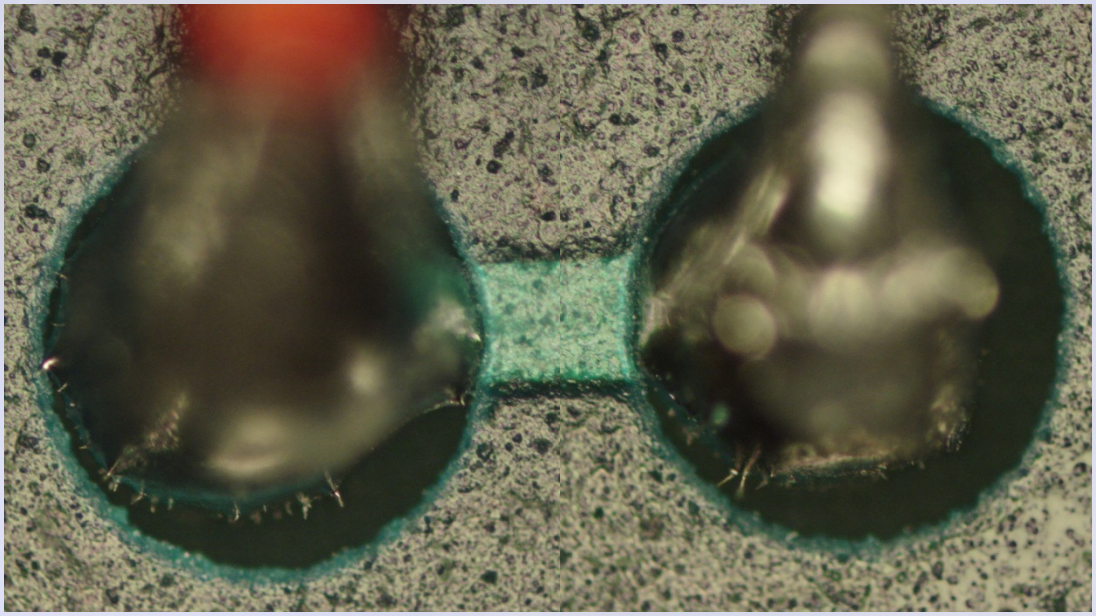
Cracked Cap Issue



- Functional Test Required Mounting Assembly In RF Test Unit Fixture
- Fixture Did Not Have PCA Eject Mechanism In Fixture – Bending Of PCA To Extract From Test Fixture Cracked Capacitor.

PLEXUS
The Product Realization Company

Source Of Contamination

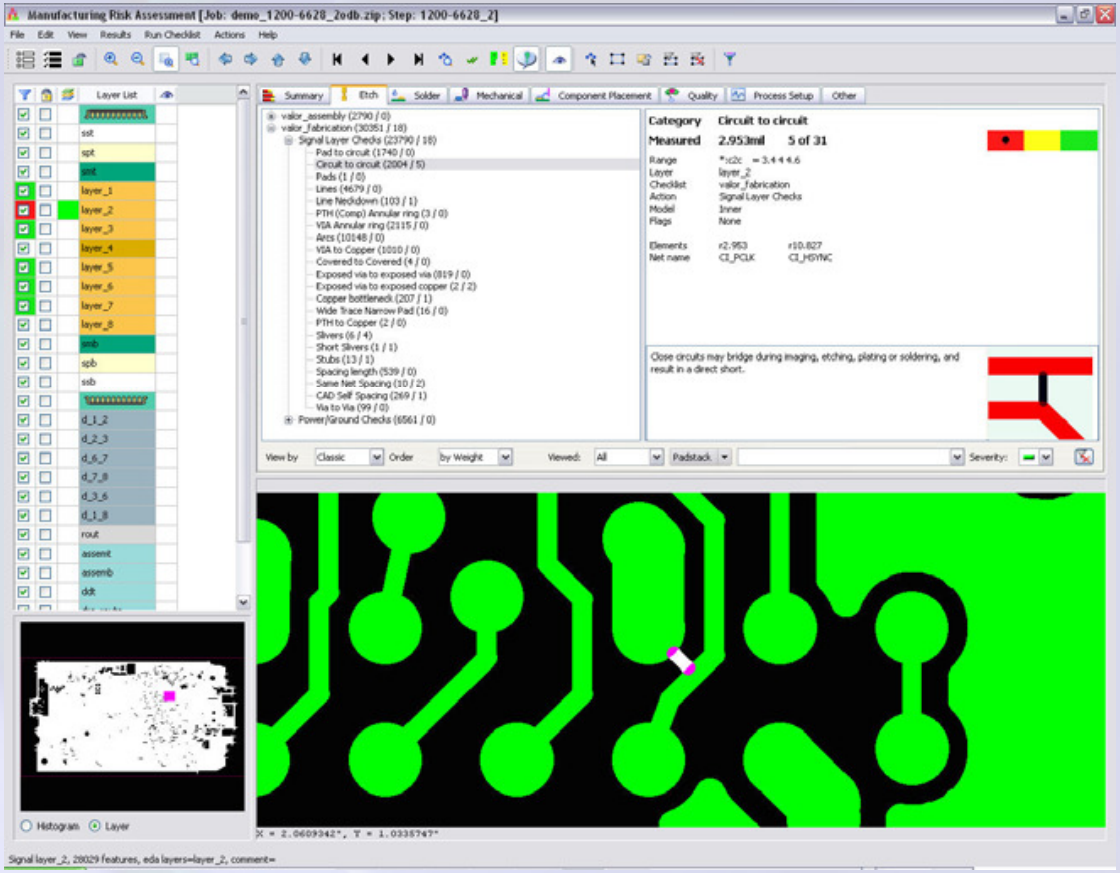


What is Source of Whiskers in Solder Joints at PCB Interface Surface?

1. **Solder Paste**
2. **Stencil Printer**
3. **Reflow**
4. **PCB Finish**
5. **Thermal Aging**

PLEXUS
The Product Realization Company

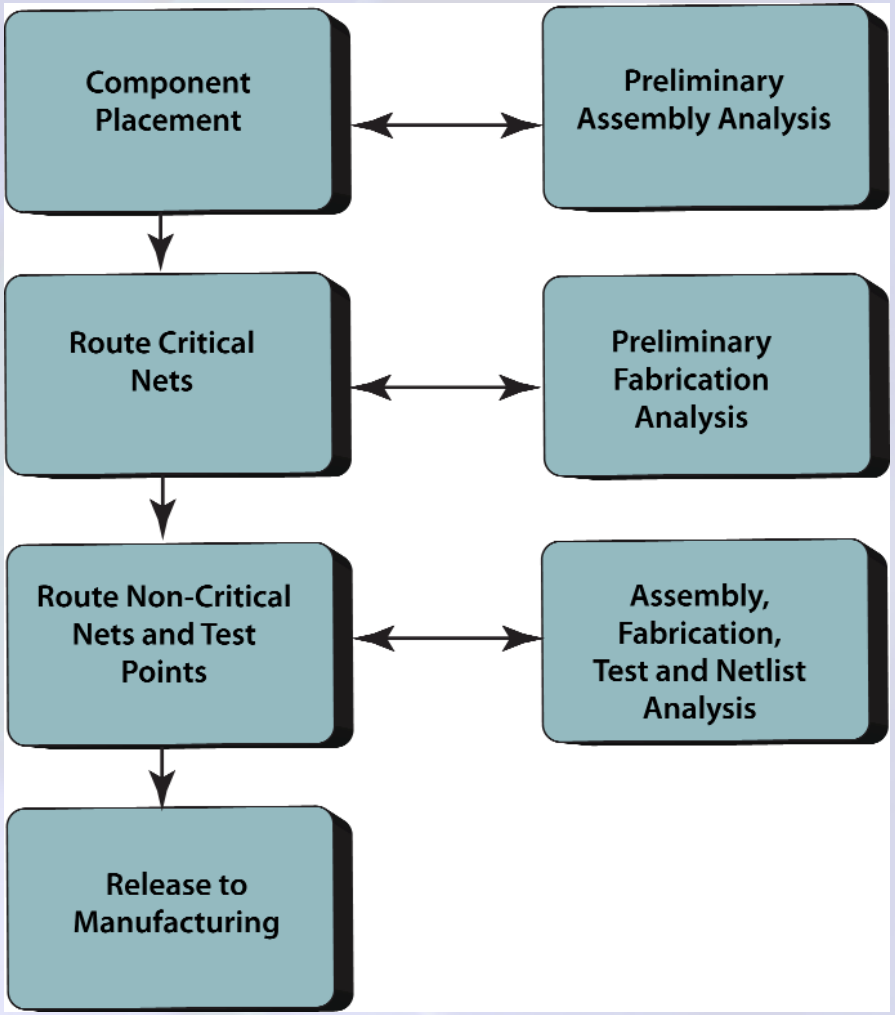
DFX Analysis Software



- Minimize the number of revision spins to bring a new design to production release
- Lower the cost to manufacture
- Improve the reliability of a design
- Reduce the amount of time spent on supplier “call-backs”
- Work with all major PCB layout tools

PLEXUS
The Product Realization Company

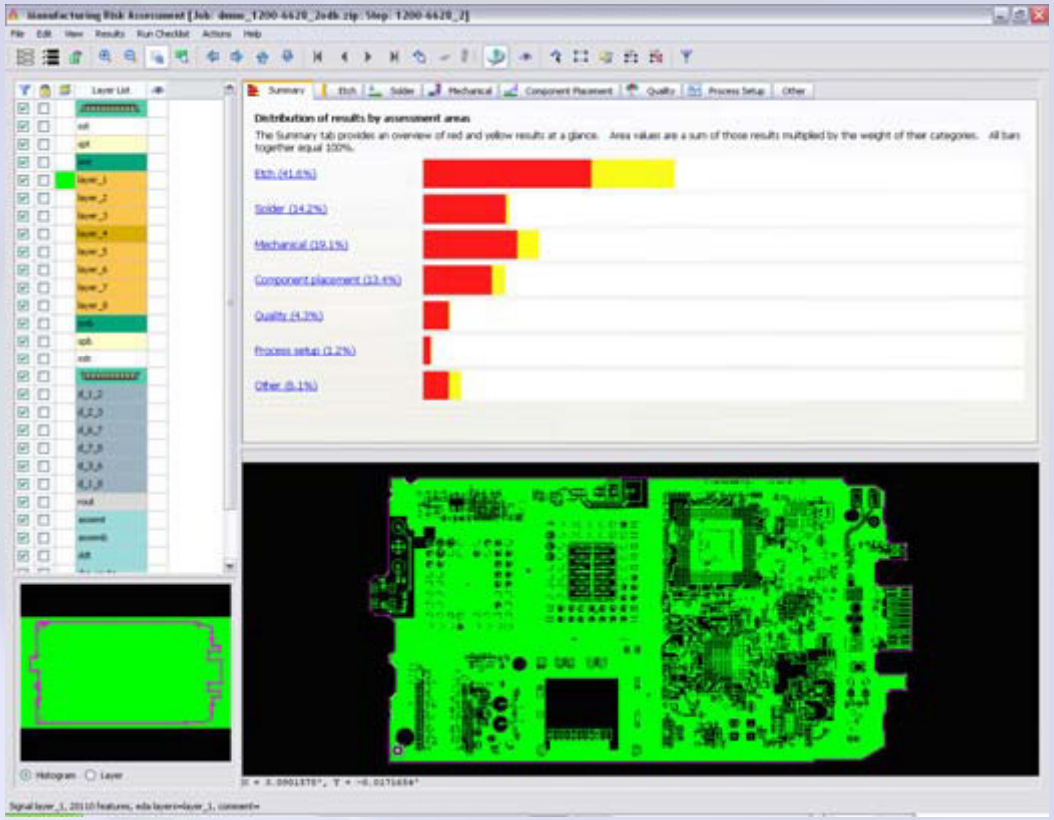
DFX Software Flexibility



- Concurrent DFX Resolves Issues At The Lowest Possible Cost
- Software Flexibility Should Allow For Multiple Points Of Access Into The Design Process

PLEXUS
The Product Realization Company

DFX Analysis Results



- Analysis Results Should Be Easy To Review And Intuitive To Any Reviewer
- Issues Presented With Severity Level Ranking

PLEXUS
The Product Realization Company

DFX PCA-PCB Software Systems

- Mentor -Valor Computerized Systems
  (Enterprise 3000, Trilogy 5000)
- Mentor – Router Solutions
  (BOM Explorer, CAMCAD)
- Siemens UGS - Technomatix
  (Assembly Expert)
- Aegis Software
  (CircuitCAM – Fusion)
- SMS
  (Linecontrol)
- Zuken
  (CR-5000 Board Modeler)
- Downstream Technologies
  (CAM350)
- ADIVA
  (DRC – Design Analysis)
- Graphicode
  (GC-CAM, GC-Place, GC-ICT)
- Wise Software
  (Gerbtool)
- Lavenir
  (Viewmate)

PLEXUS
The Product Realization Company

DFX PCA Test Software Systems

- Mentor -Valor Computerized Systems
  (Enterprise 3000, Trilogy 5000)
- Mentor – Router Solutions
  (CAMCAD)
- Siemens UGS - Technomatix
  (Test Expert)
- Teradyne
  (D2B Alchemist, D2B Strategist)
- Digitaltest
  (C-Link)
- Agilent
  (Access Consultant)
- Aster Technology
  (Testway)

PLEXUS
The Product Realization Company

DFX Mechanical Software Systems

Analysis Software

- Boothroyd and Dewhurst
  (DFMA)
- Geometric Ltd.
  (DFMPro)
- Galorath
  (SEER)

Design Software

- PTC
  (Pro/Engineer)
- Autodesk
  (AutoCAD)
- Dessault
  (Solidworks , Catia, Delmia)
- Siemens
  (NX - SDRC)

PLEXUS
The Product Realization Company

Data Format Flexibility

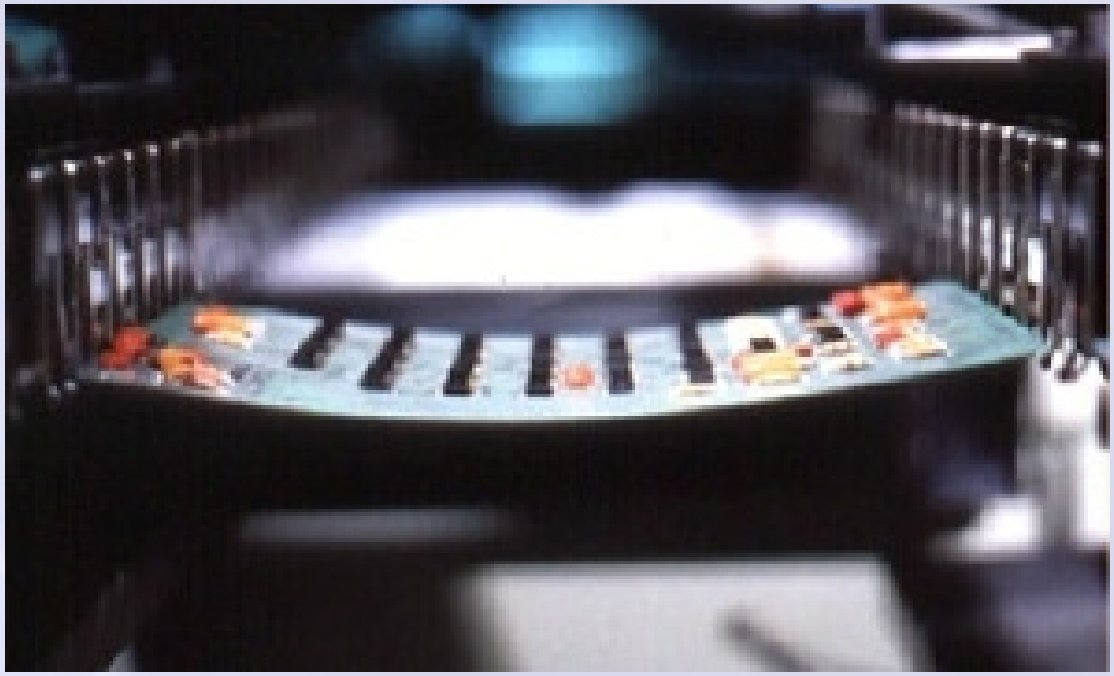
- Fabrication & Electrical Design Information
  - Board Fabrication Data - Gerber (274D, 274X), Drill Files (Excellon), etc.
  - CAM Tools - ODB++, GenCAD, GenCAM, FATF, etc.
  - CAD Tools - Cadence, Mentor, Zuken, Altium, etc.
  - Net Lists - IPC-D-356
- Assembly Information
  - BOM/AVL/Component Placement - ASCII, Excel, Word, etc.
  - Drawings - DXF, HPGL, Gerber, PDF, Postscript, HTML, etc.
- Mechanical Designs
  - Design Data –STEP, IGES, DXF/DWG, VRML, etc.

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

Thank You!

Questions

Dale Lee
Dale.Lee@Plexus.Com

PLEXUS
The Product Realization Company