Root Cause Failure Analysis of Printed Circuit Board Assemblies Through Analysis of Product and Tooling Design

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

April 2013
What do you mean 0% first pass yield?

My process is in control, so what happened?

Photo Courtesy of G. Toren
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
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)
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
  - Small Pads Have Less Margin
- Uniformity Of Trace Egress Direction
  - Some Package Types Are More Sensitive Than Others
- Uniformity Of Trace Sizes
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
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
Component Issues - Decreasing Pitch

Potential Issues:
- Paste Volume Control
- Component/PCB Flatness
  - Internal Split Plane
  - NFP Removal Impacts
- Component/PCB Warpage
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
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
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
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
Stencil To PCB Alignment

- Smaller Components Decrease Total PCB & Assembly Process Tolerance
- Minor Misalignment Can Impact Process Yields
Offset Paste - Normal Placement

Photo Courtesy of Juki Automation
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

“Using Stencil: Design to Reduce SMT Defects”, SMT, April 2006
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
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
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.
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
What To Do?

• Increase Solder Temperature?

• **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
Wave Solder & Rework Issues

Limit Effects Of Copper Dissolution

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

- 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
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

Alpha Metals, SMT Mag Webcast, Jan 2006
PCB Finish Vs Solder Spread

- OSP
- Immersion Silver
- Immersion Tin
- ENIG

Amount Of Lead Free Solder Wicking Is Dependant Upon Finish
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.
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
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
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
SMT Thieving Pads

**Purpose Is:**

To Wick Excess Solder Away From Pins

Provide Solder Wave Surface Tension Breaking Points

Direct Flow of Solder to Pins and Around Features

• Photos Courtesy of Cookson - Alpha Metals
Careful - Clean May Not Be Reliable

Solder Paste Bridge Between Pads on BGA After Reflow and Cleaning
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
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.
Thank You!

Questions

Dale Lee
Dale.Lee@Plexus.Com