TOLERANCE FORGOTTEN: IMPACTS OF TODAY’S COMPONENT PACKAGING AND COPPER ROUTING ON ELECTRONIC

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

April 2013
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)
SUPPLIER DATA DIMENSIONS

Identify Controlling Dimension
Identify Alternate Dimension
Verify Accuracy Of Units
EQUIVALENT UNITS

Single Step Measurement
Multiple Step Measurement (Accumulation Of Tolerance Differences)
Higher Number Of Steps – Greater Impact Of Differences

0.025” * 18 = 0.450” (11.43mm) vs 0.64 * 18 = 11.52mm (0.4535”)
Shift Of 0.0035” or 0.09mm
WHAT IS WRONG HERE

Mechanical Drawing  dimensions in mm(inch)

![Mechanical Drawing Image]

Figure 17  Open Frame Converter

Notes

1) All dimensions in mm (inches)  
   Tolerances: \( x \pm 5 \) \( \times x \pm 0.02 \)  
   \( xx \pm 25 \) \( \times xx \pm 0.010 \)

2) Input and function pins are 1.02mm (0.040") dia.  
   with 1.68mm (0.065") vented dia. standoff shoulders.

3) Output pins are 1.57 mm (0.052") dia. With 0.097"  
   vented shoulder.

4) All pins are coated with 90%/10% solder finish,  
   Gold, or Matte Tin over Nickel underplating

5) Weight: 45 g open frame converter  
   85 g baseplated converter

6) Workmanship: Meet or exceeds  
   IPC-A-610 Class II

PLEXUS  
The Product Realization Company
NOT ALL SUPPLIERS PACKAGES ARE EQUAL

Pin To Pad Analysis

Incorrect Package Width
COMPONENT ISSUES – DECREASING PITCH

Potential Issues:

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

Different Via Hole Structures Impact Resistance To Assembly, Higher Level Assembly / Disassembly, Rework or Handling Damage
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
MECHANICAL STRAIN IMPACTS

- PCB Design
  - MicroVia/Buried Design Impacts
  - Mechanical Strain Concentration
  - Test Point Concentration

- Pad Cratering Issue
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
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
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 or Leadless (No Solder Ball/ Solder Bump) Component Packages
Stencil Alignment of Solder Paste To Pad Tolerance May Be Critical To Good Manufacturing Yields (Dependant Upon PCB Surface Finish)

Example – OSP Finish
PCB FINISH VS SOLDER SPREAD

- OSP
- Immersion Silver
- Immersion Tin
- ENIG

Amount Of Lead Free Solder Wicking Is Dependant Upon Finish
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 Free  
Tin-Lead
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.
Pad Size Comparison

- Same design data may not yield same PCB pads sizes.
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.
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)
Tolerances between suppliers can impact assembly and soldering processes. Industry tolerances may not meet requirements for today’s assembly.

<table>
<thead>
<tr>
<th>Layer</th>
<th>Result (PCB A),mil</th>
<th>Result (PCB B),mil</th>
<th>Different,%</th>
</tr>
</thead>
<tbody>
<tr>
<td>L2</td>
<td>1.1</td>
<td>1.4</td>
<td>27.2</td>
</tr>
<tr>
<td>L3</td>
<td>0.5</td>
<td>0.6</td>
<td>20.0</td>
</tr>
<tr>
<td>L4</td>
<td>0.5</td>
<td>0.6</td>
<td>20.0</td>
</tr>
<tr>
<td>L5</td>
<td>0.5</td>
<td>0.6</td>
<td>20.0</td>
</tr>
<tr>
<td>L6</td>
<td>0.5</td>
<td>0.6</td>
<td>20.0</td>
</tr>
<tr>
<td></td>
<td>1.2</td>
<td>1.4</td>
<td>16.7</td>
</tr>
</tbody>
</table>
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
**LEAD TO HOLE CLEARANCE**

**Lead Free Soldering**
- Lead Clearance Minimum Increased
- Increasing Board Thickness May Further Increase Lead To Hole Clearance (Aspect Ratio)
- Larger Holes Create Less Voids

**Smaller Hole To Lead 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
SOLDER PASTE PRINTING

Low Aspect Area Ratio Printing

High Aspect Area Ratio Printing
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
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
Smaller Components Decrease Total PCB & Assembly Process Tolerance
Minor Misalignment Can Impact Process Yields

3 mil

6 mil
OFFSET PASTE – NORMAL PLACEMENT

Photo Courtesy of Juki Automation
Normal Manufacturing Process Variability May Exceed Allowable Assembly Process Tolerance For High Yield, Reliable Assembly
Matched Tooling (Stencils) To Materials (PCB) May Be Required
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.
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
THERMAL SHOCK

Smaller Passive Components Increase Sensitivity To Localized Soldering Process Temperatures
Impacts of PCB Design on Ability to Remove Soldering Process Residues
- Solder Mask
- Component Size
- Cleaning Exposure Time
High component density on PCB assembly impacts ability to locate test points on deliverable PCBA.

Implementation of “Off Board” test points has increased to provide test accessibility.

Depanelization exposes test point traces and increases potential for other failure modes if not addressed in design (i.e. - Z-axis spacing)
“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.
QUESTIONS