DFM Rules for BGA’s

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

August 2012
DFM Guidelines for BGA’s and LGA’s

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

August 2012
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)
Area Array Packages

- 0.5mm u-SMD
- 0.8mm Flex CSP
- 0.75mm Flex CSP
- 1.0mm Laminate CSP
- 1.0mm Flex CSP
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
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
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
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
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 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
Component Issues - Decreasing Pitch

Potential Issues:

- Paste Volume Control
- Component/PCB Flatness
  - Internal Split Plane
  - NFP Removal Impacts
- Component/PCB Warpage
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
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
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
Micro-Via Design - Reliability Impacts

Different Via Hole Structures Impact Resistance To Assembly, 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
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.
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
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

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
PCB Finish Coatings

- **Hot Air Solder Level (HAL or HASL)**
  - Immerses 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**
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
PCB Finish Vs Solder Spread

- OSP
- Immersion Silver
- Immersion Tin
- ENIG

Amount Of Lead Free Solder Wicking Is Dependant Upon Finish
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
Copper Etch Process

- $H =$ Height of Copper Trace
- Etch Factor $= \frac{2H}{(W_{\text{max}} - W_{\text{min}})}$
- Etch Factor Should Be Larger Than 1.0
Etched Trace Design Considerations

- 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
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.
PCB Fabricator Variation

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.
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)
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.
Solder Paste Printing

- Low Aspect Area Ratio Printing

- High Aspect Area Ratio Printing
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
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
• Printing Process Positional Tolerances
  Windows Decrease With Decrease in CSP Pitch
Stencil To PCB Alignment

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

3 mil                     6 mil
Offset Paste - Normal Placement
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
Matched Offset - Paste & Part

Photo Courtesy of Juki Automation
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
“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.