BTC’s – Packages We Love To Hate

Dale Lee
Plexus Corporation
Dale.Lee@plexus.com
Early BTC’s

- PGA - Pin Grid Array
- BGA - Ball Grid Array
- BGA – Bump Grid Array
IPC-T-50 Definition

BTC = Bottom Terminated Component
BTC Advantages: Size and Cost

- Smaller, lighter and thinner than comparable leaded packages
  - Allows for greater functionality per volume

- Reduces cost
  - Component manufacturers: More ICs per frame
  - OEMs: Reduced board size

- Attempts to limit the footprint of lower I/O devices have previously been stymied for cost reasons
  - BGA materials and process too expensive

BTC Packaging

- BTC’s Cover A Wide Range Of Component Package Styles And Types Including Leadless Chip Carriers, DFN, QFN, LGA, Flat Lead Terminated Packages (i.e.-563), Etc.
BTC Packaging Issue

- Lack Of Package Standardization Increases Inconsistency In Assembly Processes And Stencil Tooling Designs
QFN/DFN Solder Fillet Formation

- QFN Wetting On Edge Leads With A Solder Bump Formed On The Pads In The Front Of The Vertical Lead Due To Non-Wetting.
  - Determined That The Bottom Pads Are Tin Plated Whereas The Sides (Vertical Surfaces) Are Raw Copper And Not Meant To Be Soldered.
Not All Pads Are Equal

- Packages That Are Singulated By Saw Or Punch Cut Are Not Solderable Per IPC-A-610 Section 8.3.13 Notes 2 And 5

Reference: QFN/DFN Inspection of Solder Joints, Linear Technology Application Note
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
Solder Mask Via Tenting Options

- Vias Tented from Top
- Vias Tented from Bottom
- Via Plugged from Bottom
- Via Encroached from Bottom

Atmel Application Note
Solder Mask Design

- Exposed Via In Pad
  - Ensure Solder Mask Is Not Applied On Opposite Side Of PCB.
  - If Applied, Ensure Via Hole Is Small Enough To Restrict Solder Mask Flow Through To Solder Attach Side Of PCB
  - If Top Side Via Plugging Is Required, Especially Is Under BTC Components, A Note Should Be Added To Fabrication And Assembly Drawing Referencing This Condition
LGA Pad Design

- Pad Geometry Influences Standoff Height - 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
Board Warpage

Board Warpage Within IPC Assembly Workmanship Standards (0.75% May Not Be Adequate For Some Component Packages To Obtain High Assembly Yields)
Removal Of Copper Plane And Unused Pads

- Removal Of Copper Planes And Unused Pads Concentrated In A Localized Area Can Result In Changes In PCB Thickness (Flatness). These Changes Under Components With Limited Coplanarity Compatibility Can Result In Open Solder Connections.
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 or Leadless (No Solder Ball/ Solder Bump) Component Packages
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
Silk Screen Design Low Standoff Components

- Silk Screen Should Not Be Placed Under Body Of Low Standoff Components
  - Increased Gap For Solder To Bridge During Solder Reflow Process
  - Potential Tilting Of Components (Open Connection)
  - Potential Latent Field Failure With Partial Solder Connection
Silk Screen Design Low Standoff Components

Low Component Stand-off Height
Excessive PCB to Component Standoff Height – Open Joint
Tilted Component
Open Joints (one edge or center standoff from PCB)
Misalignment

Component Types
Leadless
• QFN, DFN, LCC
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.
Exposed Via Impact

- It is recommended that the via diameter be 0.30 to 0.33 mm with 1.0 ounce copper via barrel plating. This is desirable to avoid any solder-wicking inside the via during the soldering process, which may result in voids in solder between the exposed pad and the thermal land.
Via Hole Size

Smaller Via Hole Size In Thermal Pad
Reduce Solder Flow Into Via Hole
Exposed Via Impact

Exposed Via Holes Provide Path For Flux Volatiles And Solder To Escape From Under Component Body

Note: Solder In Via Not Continuous.
Thermal Pad Void Formation

Voids In Thermal Pad

Void Size and Quantity Decrease With Increased Number Of Exposed Via Holes In Thermal Pad
5 primary options recommended by IPC 7093 and component supplier guides

- **Option 1**: Open Copper Current Common Practice
- **Option 2**: Via Tenting
- **Option 3**: Encroached Vias
- **Option 4**: VIPPO
- **Option 5**: Floating Mask
- **New Option 6**: SMD Windows
New Design Point: SMD Windows

29 Design Rule Parameters Controlled

- Incorporates solder mask defined windows within the thermal pad area and I/O leads
  - Utilize low cost open through-hole via structures
  - Eliminate solder wicking down thermal vias
  - Ensure proper via counts to manage heat/power
  - Maximize thermal pad % coverage with solder
  - Reduce stand-off variability; improving reliability
  - Provide proper ground return paths, ensuring long term electrically stable system operation
  - Enable safe, repeatable rework process windows

Kelly, Matt et al, “Via-In-Pad Design Considerations for Bottom Terminated Components on Printed Circuit Board Assemblies”, SMTAI 2013, Rosemont, IL.
Solder Preforms – Thermal Pad

- Offers Predictable Solder Volume
- Same Metallic Properties As Solder Paste
- Flux Coating Compatible With Solder Paste
- Size Engineered To Component
- Increased Standoff Height

Homer, Seth et al, “Minimizing Voiding In QFN Packages Using Solder Preforms”, APEX Proceedings 2012, Las Vegas, NV.
Design For Cleaning

Removal Of Solder Mask Improves Ability To Clean Flux Residues From Under Component Body – Increase Stand-off
QFN/MLF Components

- Flux Residue Is Heavy For All Solder Masks
- Gap Height Is Better For
  - No Solder Mask Pads
  - Strategies For Increasing Gap Height Needed
- To Clean
  - No Solder Mask Is Best Strategy
- Time In Wash Is A Critical Parameter
BGA Components

- Flux Residue Is Lower For
  - SMD & NSMD Pads
- On No Solder Mask Pads
  - Flux Flows Away From Solder Ball
  - Penetrates Into Mask
  - More Challenging To Clean

- To Clean
  - SMD Best
  - NSMD 2\textsuperscript{nd} Best
  - No Solder Mask Worst
QFN Gap Height

Main Effects Plot for Gap Height ($\mu$m)

Data Means

<table>
<thead>
<tr>
<th>Component</th>
<th>Ground Plane Pattern</th>
</tr>
</thead>
<tbody>
<tr>
<td>MLF124 Double Row</td>
<td></td>
</tr>
<tr>
<td>MLF88 Single Row</td>
<td></td>
</tr>
<tr>
<td>HEXAGON</td>
<td>0</td>
</tr>
<tr>
<td>SLOT</td>
<td>6</td>
</tr>
<tr>
<td>SQUARE</td>
<td>25</td>
</tr>
<tr>
<td>STANDARD</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Solder Mask Definition</th>
<th>Via Holes</th>
</tr>
</thead>
<tbody>
<tr>
<td>NoSM</td>
<td>80</td>
</tr>
<tr>
<td>NSMD</td>
<td>90</td>
</tr>
</tbody>
</table>

Graph showing the effect of different components and ground plane patterns on gap height.
Exposed Via Hole Impact

- Increase Number Of Via Holes In Thermal Pad Results In:
  - Fewer Voids In Thermal Pad
  - Less Flux Residue Under Component Body
  - Decreased Component Stand-off Height
Summary

- Lack Of Standard Definition For This Component
- Variability In Package Design Increases Complexity In Assembly Process Requirements
- Manufacturing Tolerances For High Yield Tighter Than Current Industry Material Standards
- Increased Number Of Via Holes In Thermal Pad Decreases Voiding
- Increased Number Of Via Holes In Thermal Pad Decreases Stand-off Height For Same Solder Paste Coverage Percentage
- Cleaning Improves With Increase Stand-off Height And Removal Of Solder Mask On QFN’s And Solder Mask Defined Pads on BGA’s
Thank You!

Questions