DFM & DFA

Ankan Mitra, SMTA India Chapter
SMTA Luncheon Meeting 23rd November 2013
SMTA Kick-Off Meeting

Hello Bangalore, How do you do?

Hello Bengaluru कसे आहात?

Hello bengaluru कसे आहात?

Hello Bengaluru तमे केवी रीते जे?

హలో బెంగాలురు ఎలా ఉన్నావు?

Hello Bengaluru నీవు ఎంటాం?

Hello Bengaluru ఆప్ని కేమను ఆచేన?

ہیللو بنگلون  تم کیسے بو؟

Hello Bengaluru آپ کئے ہیں؟

Newprojs_ankan@yahoo.com
SMTA Kick-Off Meeting

Create a technology platform

Ensure SME growth to enhance Indian Supplier base

Bring in Advanced Electronics Manufacturing best practices

Newprojs_ankan@yahoo.com
Mode of Delivery

- Exchange of ideas
- Best Practices
- Next Practices

You are also encouraged to bring your defects for discussion and root cause analysis
**Agenda**

- What is DFM/DFA?
- Typical DFM/DFA Process Flow
- Major Yield Drivers
- Soldering terminology
  - Surface Tension / Wetting / Capillary Action / Wetting Angle
- Design Impacts
  - PCB
  - PCBA
  - Thermal Balance, Trace Routing, Equipment Limitation/Tolerance
- Manufacturing Tooling Design Impacts
  - Wave Solder
  - SMT Solder
  - Stencil Designing
- Manufacturing Process Design Impacts
  - Manufacturing Process Control / Process Window
  - Cleaning Impact
  - Reflow Process Warp-age
- Design for Assembly
  - Panel Designing
  - Next Higher Assembly
What all we have possibly heard?

- Design for Quality
- Design for Environment
- Design for Life Cycle Costs
- Design for Testability
- Design for Six Sigma
- Design for System Quality
- Design for Usability
- Design for Sustainability
What all we have possibly heard?

DFM is a for PCB Designers…

DFA only possible after Proto-build…

DFM need expensive licensed tool…

Same Rule Sets apply for all boards…

DFM / DFA is about Component Selection…

DFM / DFA is universal and not correlated to manufacturing process…

Like Quality – DFM/DFA is everyone’s business
DFM/DFA

Everything done in design is for manufacturing; hence DFM / DFA is a misnomer.
Introduction : DFM & DFA

Key Take-aways:

- DFM – different perspectives
- Basic understanding of Soldering
- Impact and Cost involved
- Key Concepts of DFM at various stages
- Review and Classification examples from various stages of manufacturing process
What is DFM and DFA?

Design for Manufacturability
Design for Assembly
Are about Yield & Reliability

Lack of design or manufacturing access → less knowledge on process/yield/reliability learning

“After-the-fact” FA on products is expensive, and most often unsuccessful

DFM & DFA with proper manufacturing information mitigates Risks introduced by:

• parametric variation in manufacturing
• greater sensitivity to variations
• reduce design specific yield/reliability risk

Newprojs_ankan@yahoo.com
What is DFM and DFA?

DFM

Yield

Reliability

Usually, yield and reliability are linked – but acceptable yield may not always guarantee acceptable reliability

Hence we need to understand:
  • Parameters which impact the yield
  • Basics of Manufacturability requirements
What is DFM and DFA?

4 Major Factors considered

**Impact per occurrence:** an infrequent issue can be significant if it has a large impact each time it occurs, e.g. scrapping an expensive component due to rework.

**Frequency per board:** even a small yield issue can become significant if a large number of components/features on each board are affected.

**Production volume:** even a small issue can have a significant impact in manufacturing for a product with very high production volume.

**NRE (Non-recursive Expenses):** for low volume production, the impact of additional NRE charges, e.g. tooling, line/process set-up time, can be significant.
What is DFM and DFA?

Categorization based on the Factors considered

**Show-stopper**: even a small issue can have a significant impact in manufacturing for a product with very high production volume.

**Better to change / Costs $$$ is retained**: even a small yield issue can become significant if a large number of components/features on each board are affected.

**Good to change**: an infrequent issue can be significant if it has a large impact each time it occurs, e.g. scrapping an expensive component due to rework.

Examples!!!
What is DFM and DFA? – How will you categorize these examples.

**Silkscreen Under Zero Standoff Components**

**PROBLEM:** the combined thickness of the solder mask and silkscreen lifted the body of the component, causing open solder joints.

**Show-stopper**

**IMPACT:** $0.01 / component due to increased defect level.

On this board, there were 300 components with this problem and therefore total cost adder was $3 per board.
What is DFM and DFA? – How will you categorize these examples.

**Silkscreen Clearance**

**PROBLEM:** Silkscreen inside BGA land pattern, too close to lands.

**Better to change**

**IMPACT:** Silkscreen may overlap onto lands, causing poor solder joints.

Newprojs_ankan@yahoo.com
What is DFM and DFA? – How will you categorize these examples.

Master Panel Utilization

When adding handling edges, ensure that the number of images per Master Panel is not reduced, which increases the cost of each PCB.

Handling edge

Working border

Good to change

IMPACT: Images #5 and #6 no longer fit within the working area of the Master Panel. PCB cost increases by 50% due to slight increase in board size from the addition of handling edge.

Newprojs_anikan@yahoo.com
Typical DFM/DFA Process Flow

System Architecture
- Start Specifications
- Major Component Selection

System Design
- Finalize Specifications
- Secondary Comp. Selection

Physical Design
- System Box/Rack
- Sub-Components

MDA Information
- Component Symbols
- Existing sub assemblies

DFx Info
- Design Guidelines
- asm, fab, test

Technical Specs.
- Proven components

Market Analysis
- Price/Volume Break
- Committed Delivery
- Allocation Info.

Quality & Reliability
- Proven performance
- Component Level
- Assembly Level

What CEM's can provide?

eg, proven footprints, spacings, processes, component price/ availability

Newprojs_ankan@yahoo.com
Typical DFM/DFA Process Flow

- Positive Cash Flow
- Negative Cash Flow
- Design Collaboration
- Rapid NPI
- Volume BTO
- Faster Time to Profitability (Global)
- Cost Reduction
- Increased Profitability
- EOL Transition
- Time to Market
  - 25% Faster Profitability
  - +6-8% Margin

Newprojs_ankan@yahoo.com
Typical DFM/DFA Process Flow

System Architecture
- System Design
- Physical Design
- Proto and Pilot
- Volume Production

Traditional Design Error Detection and Feedback

Future Participative Design Review and Feedback

Want early, real time, interactive collaboration vs release, correct, redo

Newprojs_ankan@yahoo.com
Major Yield Drivers

- All Processes are set still Manufacturing Yield Issues???
Major Yield Drivers

- 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)
Major Yield Drivers

- Defects related to Lead Free Design & Manufacturing Processes
  - Component termination to PCB Surface Finish mis-match
  - Component termination to Manufacturing chemistry mis-match
  - Same PTH dia. Maintained for Leaded and Lead-free board
  - PCB Laminate material not verified for qualification in Lead-free process
Soldering terminology

- Surface Tension
- Wetting
- Capillary Action
Soldering terminology

- Wetting Angles

A: <90°
B: 90°
C: >90°
D: >90°

Ref.: IPC-A-610
Soldering terminology

- **Relevance:**
  - Enough volume of solder paste is required on the pad for a Surface Mount Device
  - Enough board thickness should be present to ensure reliable connection between barrel and termination for a Through hole device
  - Pad or Annular Ring should be sufficient to ensure wetting angle requirements are met
  - Special consideration when mixed processes like Intrusive soldering is used
Soldering terminology

- A Visual Comparison:

- Preferred Wetting
- Small Amount of Dewetting
- Complete Dewetting
- Nonwetting

Ref.: IPC-T-650
Soldering terminology
Design Impacts

- PCB

- PCB Design Consideration
  - Board Material Properties
  - Surface Finish of PCB
  - Test Point Requirement
  - PCB Assembly Requirement (Component Interference with Mechanics)
  - Panel Design for PCBs
Design Impacts

- PCBA

  PCB Assembly Design Consideration

  - Component Compatibility to Manufacturing & application
  - Component Orientation to ease manufacturing
  - Proximity of similar component packages
  - Tall Profile Components (Component Interference with Mechanicals)
  - Consideration for Thermal shadow and Thermal sinks on board
  - Component clearance for special operations (Under-fill / cleaning)

Newprojs_ankan@yahoo.com
Design Impacts

- Thermal Balance

Large copper areas on your circuit board should always be a grid if possible - gridded area is called “hatch”

May not be possible if high frequency conductor traces present

**Advantage:** The “grid” helps a great deal to avoid “Twist” and “Bow” effects, especially for circuit boards with only one layer.
Design Impacts

- **Thermal Balance**

Large copper areas should be counterbalanced with “copper filling” in the opposite layer.

Conductor trace distribution as uniformly as possible across the board.

For MLBs symmetrically opposite layer with “copper filling”
Design Impacts

- Thermal Balance

The thicknesses of the copper foil in the layer buildup of your circuit board should always be symmetrically distributed.

An asymmetric layer build-up can be manufactured, but has high possibility of deformations.
Design Impacts

- **Trace Routing**
  - Traces running at the edge of the board needs:
    - Additional thickness – to absorb mechanical stress (handling / depanellization)
    - Stitching of trace with additional copper spread at intervals and bends (non-possible for signal traces)
  - Spacing violation between fiducial and trace
    - Impact: this fiducial can’t be used
Design Impacts

- **Limitation/Tolerance**
  - *(During PCB Fabrication) Silk-screen clearance*
  - Adequate silkscreen clearance reduces the risk of silkscreen mis-registration onto the lands (typical tolerance is +/- 0.3mm (0.012”)) and improves fine pitch paste printing quality.

- *(During PCB Assembly) Equipment Limitation vs. PCB Tolerance*
  - Thicker PCBs may require special handling, e.g. fixtures, if they are too thick to fit in all of the assembly equipment
    - All assembly & inspection equipment, including conveyors, must be considered
  - Maximum nominal thickness 3.0mm (0.120”), to account for typical PCB thickness tolerance of +/-10% (allows maximum PCB thickness of 3.3mm / 0.130”)

Newprojs_ankan@yahoo.com
Design Impacts

- **Process Tooling Design / Tooling Holes**
  - Tooling Holes are used:
    - By PCB Fabricator for manufacturing processes
    - PCB Assembly – for clamping requirement to fixtures
    - PCB Assembly – for special processes like Selective Soldering
    - PCBA Test – clamping on fixtures (eg. ICT)
  - The PCB fabricator requires a working border on the Master Panel for tooling, test coupons, etc. Gaps for routing must also be considered in the Master Panel layout.

Newprojs_ankan@yahoo.com
The PCB fabricator requires a working border on the Master Panel for tooling, test coupons, etc. Gaps for routing must also be considered in the Master Panel layout.
Design Impacts

- **Process Tooling Design / Tooling Holes**
  - Tooling holes are used in ICT to physically align the Assembly Panel for accurate probing of the test points. Some assembly equipment also requires tooling holes for registration.

- 3 NPTH per Assembly Panel, shared for SMT and ICT
  - 4.0mm (0.157”) diameter preferred, 3.18mm (0.125”) accepted
  - Centres for SMT tooling holes located at 5mm (0.197”) from edge of Assembly Panel
Design Impacts

- Process Tooling Design / Tooling Holes
  - Selective Wave Fixture Tooling Holes
    - Provide 1 hole for every 100 square inches of board, e.g. for a 10” x 10” board (254mm x 254mm), 1 hole is recommended, for 18” x 20” (457mm x 508mm) board, minimum 3 - 4 holes are recommended
    - Locate 1 NPTH as close to the centre of the board as possible, and spread the remaining holes along the card X and Y centrelines
    - Locate the tooling holes near to areas that will have openings in the selective wave fixture, to aid in maintaining a seal between the fixture & the PCB

Newprojs_ankan@yahoo.com
Manufacturing Tooling Design Impacts

- Wave Solder
  - Special considerations for Thick PCBs for following combinations:
    - OSP surface finish
    - No-clean wave solder flux
    - Multiple board reflows prior to wave solder
    - Missing thermal reliefs
    - Multiple power/ground planes
  - There may be problems with wave solder hole fill and PTH repair on boards that have 3 or more ground planes
    - If possible, connect each soldered PTH barrel to a maximum of 2 planes. If the planes must be tied together, they should be connected using separate vias which do not need to be soldered.
Manufacturing Tooling Design Impacts

- **SMT Solder**
  - For special shape boards which require additional fixtures, ensure the overall size (including the fixture) can be held through the SMT Machines

- Rework and Repair: Space between components, thermal relief and consideration for back to back components to ensure that rework is possible without disturbing peripheral components

Newprojs_ankan@yahoo.com
Manufacturing Tooling Design Impacts

- SMT Solder
  - Ensure adequate spacing between SMT components by providing adequate keep-out area
  - Typical Component Spacing Table extract

<table>
<thead>
<tr>
<th>Area Array and Leadless</th>
<th>J-Lead</th>
</tr>
</thead>
<tbody>
<tr>
<td>W/N</td>
<td>W</td>
</tr>
<tr>
<td></td>
<td>N</td>
</tr>
<tr>
<td>W</td>
<td>.150&quot;</td>
</tr>
<tr>
<td>N</td>
<td>.100&quot;</td>
</tr>
</tbody>
</table>

‘W’ & ‘N’ refer to the ‘With Leads’ or ‘No Leads’ edge of the package

- SMT components between 0.080” (2mm) and 0.197” (5mm) of board edges
Stencil Designing

- Special requirements for paste-in-hole
  - Standoffs should not come in contact with the wet solder paste
  - Minimum standoff height = 0.003” (0.076mm) + (stencil thickness x 1.8)
  - Preferred: 0.035” (0.89mm)
  - Acceptable: 0.020” (0.5mm)
  - Minimum: 0.015” (0.38mm)

“Dog ear” peaks can be up to 1.8X the height of the paste “brick”.

![Diagram showing standoffs and paste-in-hole with nominal and peak heights labeled as $h_{nom}$ and $h_{peak}$ respectively.](image-url)
Manufacturing Process Design Impacts

- Cleaning Impact
  - Ensure process steps are sequenced properly to avoid rework
    - Adhesive or heat-shrink glue for attachment may get affected if a cleaning process is introduced
Manufacturing Process Design Impacts

- Reflow Process Warp-age
  - 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
Design for Assembly

Panel Designing

- Includes requirements for all assembly & test equipment: size and location of tooling holes, fiducials, corner chamfers, board edge component keepouts.

---

SMT and Paste-in-Hole component
keepout area: 5.0mm (0.197") sides A and B on conveyed edges.

- Fiducials:
  - At least 3 required, located in corners
  - Minimum spacing of fiducial centres: 5mm (0.197") from panel edges

- Tooling Holes:
  - 3 non-plated holes required - 4mm (0.157") preferred, off
  - 3.175mm (0.125") acceptable
  - Tolerance: +0.075mm (0.003") to -0.00mm (0.00")
  - Holes to be in 3 corners as shown, centred 5mm (0.197") from each edge (minimum)

- Direction of Travel

Note: Leading and trailing edges must be straight as shown. Corner chamfers are allowed and recommended. Cutout areas are not recommended.

---

Newprojs_ankan@yahoo.com
Design for Assembly

- Next Higher Assembly
  - Interference Considerations for Next Higher level assembly
- Ease of Assembly
- Rework & Repair capability post assembly
Wrap-up

Newprojs_ankan@yahoo.com
Wrap-up

Q&A

Newprojs_ankan@yahoo.com