PCB PRODUCTION, PART 4
HIGH DENSITY INTERCONNECT PCBs

Michael Kress

WURTH ELEKTRONIK MORE THAN YOU EXPECT
AGENDA

1. High Density Interconnect (HDI) so far: MICROVIA.hdi
   - Production processes MICROVIA.hdi
   - Standard Stackups and Design Rules
   - Application examples
   - The limits of MICROVIA.hdi

2. HDI next generation: SLIM.hdi
   - Production processes SLIM.hdi
   - Standard Stackups and Design Rules
   - Application examples
   - Advantages and challenges of SLIM.hdi

3. Summary and outlook
Poll: Multiple choice with only one correct answer

What is the smallest BGA pitch you are currently using?

- BGA pitch greater than or equal to 0.8 mm
- BGA pitch greater than or equal to 0.5 mm
- BGA pitch greater than or equal to 0.4 mm
- BGA pitch greater than or equal to 0.3 mm
- BGA pitch less than 0.3 mm
Complexity / number of processing steps

Costs

100 %

1. 1 + 6 + 1

2 + 4 + 2

3. 2 + 4 + 2

Laser drilling from 1 to 3

115 %

Inside Microvias (staggered)

120 %

Buried Vias mechanically drilled

142 %

Additional inside microvias

150 %

2 + 4b + 2

2 + 4(6b) + 2

200 %

single lamination

double lamination

triple lamination

MICROVIA.HDI

Production processes / cost relations

MICHAEL KRESS | 07.12.2022
Find many other standard stackups on our Website
MICROVIA.HDI

Design Rules

- Microvias (for impedance controlled PCBs)
- Stacked microvias
- Standard microvias
- Outer layer layout

Microwa aspect ratio = 1:0.8 (diameter / depth)

Download here
The limits of MICROVIA.hdi - BGA-Pitch 0.50 mm

<table>
<thead>
<tr>
<th></th>
<th>Var. 1</th>
<th>Var. 2</th>
<th>Var. 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>BGA solder pad</td>
<td>300 – 330 μm</td>
<td>240 μm</td>
<td>275 μm</td>
</tr>
<tr>
<td>Solder mask clearance</td>
<td>50 μm</td>
<td>40 μm</td>
<td>35 μm</td>
</tr>
<tr>
<td>Microvia pad outer layers</td>
<td>≥ 300 μm</td>
<td>275 μm</td>
<td>275 μm</td>
</tr>
<tr>
<td>Microvia pad inner layers</td>
<td>275 μm</td>
<td>275 μm</td>
<td>275 μm</td>
</tr>
<tr>
<td>Track width / spacing outer layers</td>
<td>≥ 100 μm</td>
<td>80 / 90 μm</td>
<td>75 μm</td>
</tr>
<tr>
<td>Track width / spacing inner layers</td>
<td>75 μm</td>
<td>75 μm</td>
<td>75 μm</td>
</tr>
</tbody>
</table>
MICROVIA.HDI

The limits of MICROVIA.hdi - BGA-Pitch 0.40 mm
**MICROVIA.HDI**

The limits of MICROVIA.hdi

- Limit soldermask - WHY?
  - min. web width = 70 µm
  - min. distance solder mask web to pad edge = 35 µm

In total: Pad edge to pad edge min. 140 µm, see sketch on the right side.

In our design example for 0.40 mm BGA pitch this means:
- maximum possible solder mask web at the narrow point:

  400 µm (pitch) - 275 µm (pad) - 2x (35 µm (solder mask clearance))

  = 50 µm solder mask web
PCB PRODUCTION, PART 4: HDI

Poll: Multiple choice with only one correct answer

How much would reducing the pad Ø from 0.275 mm to 0.225 mm help in the layout?

- Helps very strongly (enables completely new solutions)
- Helps strongly
- Helps less
- Does not help at all
  (then we ask for additional explanation in the question field)
HDI NEXT GENERATION: SLIM.HDI

Profile SLIM.hdi?

- Anylayer-Microvia-Technology
- Very thin, rigid FR-4.1 materials (Tg150 °C, low halogen, filled) ultra-thin stackup
- Laser drilled microvias Ø 85 µm in pad Ø 200 µm
- Very thin copper layer thicknesses on all layers
- Optimal for routing of finest BGA components
- 75 µm structures
- Options
  - Impedance defined design
  - Stiffener
  - Solder carrier
Production processes Anylayer Microvia Technology SLIM.hdi 1-2b-1

- Inner layer production of core with laser drilled microvias L2 - L3 + copperfilling
- Inner layer etching up to max. 25 µm copper thickness

- Pressing to 4-layer multilayer
- Laser drilling microvias Top - L2 and Bot - L3 with subsequent copperfilling
- Etching of outer layers up to max. 35 µm copper thickness (nominal 25 µm)
- Outer layer fabrication with solder mask and final surface finish
**FR-4.1 Materials**
(Tg150 °C, low halogen, filled)

- Core material selection:
  - 0.06 mm, starting copper 12 µm
  - 0.10 mm, starting copper 12 µm

- Final thickness by number of layers

<table>
<thead>
<tr>
<th>layer count</th>
<th>total thickness</th>
</tr>
</thead>
<tbody>
<tr>
<td>4 layers 1-2b-1</td>
<td>≤ 0.35 mm</td>
</tr>
<tr>
<td>6 layers 2-2b-2</td>
<td>≤ 0.45 mm</td>
</tr>
<tr>
<td>8 layers 3-2b-3</td>
<td>≤ 0.60 mm</td>
</tr>
</tbody>
</table>
SLIM.HDI

Design Rules

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Technical Standard</th>
<th>Advanced requirements</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Line widths and spacing</td>
<td>75µm/75µm only Microvias</td>
<td>75µm/100µm with PTH</td>
</tr>
<tr>
<td>A</td>
<td>Minimum pad diameter for microvia (for PTH)</td>
<td>225µm (-)</td>
<td>200µm (400µm)</td>
</tr>
<tr>
<td>B</td>
<td>Finished hole diameter of lasered microvia</td>
<td>85µm</td>
<td>85µm</td>
</tr>
<tr>
<td></td>
<td>Distance copper to outline</td>
<td>≥300µm</td>
<td>≥225µm</td>
</tr>
<tr>
<td></td>
<td>Number of copper layers in total</td>
<td>4 bis 8</td>
<td></td>
</tr>
<tr>
<td>C</td>
<td>Thickness of core (FR4.1 - TG150, halogenfree, filled)</td>
<td>100µm</td>
<td>60µm</td>
</tr>
<tr>
<td></td>
<td>Thickness of cold-bonded stiffener made of FR-4.0 material</td>
<td>0.8 mm</td>
<td>1,00mm – 1,55mm</td>
</tr>
<tr>
<td></td>
<td>Thickness of cold-bonded solder carrier made of FR-4.0</td>
<td>0,8mm</td>
<td>0,8mm</td>
</tr>
<tr>
<td></td>
<td>Thickness of glue for stiffener or solder carrier</td>
<td>50µm</td>
<td></td>
</tr>
<tr>
<td>W</td>
<td>Minimum bridge width photosensitive solder mask</td>
<td>70µm</td>
<td>50µm</td>
</tr>
<tr>
<td>Cl</td>
<td>Minimum clearance of copper pad with solder mask, circumferential</td>
<td>40µm</td>
<td>35µm</td>
</tr>
</tbody>
</table>

Further specifications available on request, please contact us: slim.hdi@we-online.com

- Land Pad acc. IPC

→ Microvia-in Pad possible

Download Design Rules here
**SLIM.HDI**

**Design Parameters BGA 0.35 mm with solder mask**

Comparison Design Rules SLIM.flex / SLIM.hdi:

- BGA pad diameter Ø: 210 µm / 225 µm
- Solder mask web: 70 µm / 55 µm
- Solder mask clearance: 35 µm / 35 µm
- Laser drilled Microvia Ø: 85 µm
- Lines / spaces: 75 µm

Only microvia-in-pad technology including copperfilling possible!
SLIM.HDI

Design Parameters BGA 0.30 mm pitch with solder mask, Technology SLIM.flex

Design Rules only applicable for SLIM.flex:

- BGA pad diameter Ø: 180 µm
- Solder mask web: 50 µm
- Solder mask clearance: 35 µm
- Laser drilled Microvia Ø: 60 µm
- Lines / spaces: 75 µm

Only microvia-in-pad technology including copperfilling possible!
PCB PRODUCTION, PART 4: HDI

Poll: Multiple choice with only one correct answer

Have you recently processed PCBs with a total thickness of less than 0.50 mm?

- Yes, happens more often lately
- Yes, but they are still isolated cases
- No, has not been an issue lately
CHALLENGES IN THE ASSEMBLY OF SLIM.HDI

Very thin stackups → handling in the assembly process difficult

Solution 1
- FR4 solder carrier 0.80 mm
  - This results in single-sided assembly!
  - What to do if 2-sided assembly becomes necessary?

Solution 2
- Opening of the FR4 solder carrier
  Solder paste printing via step stencil / dispenser technology.
  - Further option:
    FR4 reinforcement 0.1 - 0.15 mm only in the delivery panel
SOLDER CARRIER

Short Film about the System Solution PCB on FR4 Solder Carrier
SUMMARY

HDI = MICROVIA.hdi & SLIM.hdi

- Limits for standard MICROVIA.hdi design: BGA component pitch 0.40 mm
- From BGA component pitch 0.35 mm: SLIM.hdi offers finer design parameters
- From BGA component pitch 0.30 mm: SLIM.flex technology
- Challenges in the assembly process with PCB thicknesses ≤ 0.50 mm
  - Solder carriers for the assembly process, talk to us

Outlook:

SLIM.hdi (with rigid materials) for BGA component pitch 0.30 mm in development
THANKS FOR YOUR ATTENTION

Basics of printed circuit board production
High Density Interconnect (HDI)