Signal Integrity: Impedance matching in combination with BGA fan-out
Agenda

- Introduction – fine pitch BGAs and impedance

- Examination of different BGAs according to their impedance requirements:
  - 1.0 mm pitch BGA
  - 0.8 mm pitch BGA
  - 0.65 mm pitch BGA
  - 0.5 mm pitch BGA
  - 0.4 mm pitch BGA

- advantages µVias

Philipp Reeb
Produktmanagement
BGAs and their impedance requirements

- Impedance defined tracks, single ended / differential in an BGA need to be fanned out
- Typical requirements, 50ohm (coaxial wires, GPS) 90ohm (USB) 100ohm (LVDS, Ethernet, DDR3, SATA)
- Bandwidth on a PCB normally appr. 45 – 120ohm (CAN Bus)
- Ideal: critical signals on the inner layers thinner and more constant copper thickness (higher precision ), good EMC shielding
- Type and number of the connections( number of rows) leading to the number of layers of the PCB as via pad size is restricted Design Rules
BGAs: what needs to be considered according to impedance

- BGAs 1.27mm, 1.00mm, 0.8mm are possible without µVias. Possible track width is determined by the grid spacing and the pad sizes, the number of layers and the PCB thickness also depend on this.
  
  ![Basic Design Guide](image)

- BGA Pitch 0.65mm only possible with µVias
  
  ![HDI design guide](image)

- BGA Pitch 0.5mm and 0.4mm only with µVias, layer distance 60-70µm
  
  e.g. 50 and 100ohm signals should be routed on the core layers below the µVia layers, or fine line structures (< 100µm)
To consider:

- In most PCBs with less than 12 layers even 35µm copper on the cores are possible.
- For high layer PCBs with more than appr. 12 layers it could be needed to work with 17µm copper, especially with defined thickness.

Smaller structures are possible
Smaller layer distances are possible

Important:

- Consideration of pad sizes and aspect ratio rules according to design rules (basic- and HDI design guide) to have a high reliability during thermal cycles in the via.

more stable copper in the barrel, higher reliability
BGA 1.00 mm

- 1.00 mm pitch BGA: with PTHs
  - Due to restrictions of the via pad size the number of the rows that need to be fanned out leads to the number of signal layers needed
  - Number of layers determine which layers distances are needed
  - Layer distances define the track widths and gaps
    - Depending on the layer count there are different possibilities to dimension the track widths and gaps between the tracks
    - There are more possibilities to dimension the tracks and gaps if less layers are needed
  - Possibly two tracks can be routed in between the pads (100µm track and space), Tracks need to be correctly dimensioned outside the BGA (Impedance requirements)
## BGA 1.0mm outer layer examples

<table>
<thead>
<tr>
<th>S1</th>
<th>GND</th>
<th>Folie</th>
<th>t</th>
<th>(µm)</th>
<th>Zo 50 Ohm @ 153 µm track width</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>17.8 µm</td>
<td>1</td>
<td>55</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>1 x 2113</td>
<td>3.6</td>
<td>92</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>35 µm</td>
<td>33</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>S1</th>
<th>GND</th>
<th>Folie</th>
<th>t</th>
<th>(µm)</th>
<th>Zdiff 90 Ohm @ 125 / 130 / 125 µm</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>17.8 µm</td>
<td>1</td>
<td>108</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>35 µm</td>
<td>33</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>S1</th>
<th>GND</th>
<th>Folie</th>
<th>t</th>
<th>(µm)</th>
<th>Zdiff 100 Ohm @ 100 / 142 / 100 µm</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>17.8 µm</td>
<td>1</td>
<td>130</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>35 µm</td>
<td>33</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>S1</th>
<th>GND</th>
<th>Folie</th>
<th>t</th>
<th>(µm)</th>
<th>Zdiff 120 Ohm @ 100 / 200 / 100 µm</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>17.8 µm</td>
<td>1</td>
<td>130</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>35 µm</td>
<td>33</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
BGA 1.0 mm inner layer examples

- Zo 50 Ohm @ 110 µm track width
- Zdiff 90 Ohm @ 100 / 122 / 100 µm
- Zdiff 100 Ohm @ 100 / 230 / 100 µm

- Zo 50 Ohm @ 172 µm track width
- Zdiff 90 Ohm @ 125 / 125 / 125 µm
- Zdiff 100 Ohm @ 100 / 140 / 100 µm
- Zdiff 120 Ohm @ 100 / 350 / 100 µm
BGA 0.8 mm

- 0.80 mm pitch BGA: with PTHs and/or μVias
  - layer stack like 1.0mm pitch BGA
  - Depending on BGA size maybe μVias could be useful
  - more space for fan out due to smaller pads
  - with PTHs max. 1 track can be routed in between the pads on inner layers

PTHs

μVias
Advantages of µVias at BGA pitch 0.8 mm

BGA – Pitch 0.80 mm  20 x 20 rows  Design study

How many signal layers are needed?
0.8mm outer layer examples

<table>
<thead>
<tr>
<th>S1</th>
<th>Foil</th>
<th>17.5 µm</th>
<th>[µm]</th>
<th>[µm]</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1 x 1080</td>
<td>1080</td>
<td>16</td>
<td></td>
</tr>
<tr>
<td></td>
<td>1 x 2116</td>
<td>2116</td>
<td>33</td>
<td></td>
</tr>
<tr>
<td></td>
<td>2 x 1080</td>
<td>2160</td>
<td>33</td>
<td></td>
</tr>
</tbody>
</table>

Zo  50 Ohm @ 110 µm track width
Zdiff 90 Ohm @ 100 / 130 / 130 µm
Zdiff 100 Ohm @ 100 / 270 / 100 µm

Zo  50 Ohm @ 178 µm track width
Zdiff 90 Ohm @ 150 / 150 / 150 µm
Zdiff 100 Ohm @ 100 / 127 / 100 µm

Zo  50 Ohm @ 237 µm track width
Zdiff 90 Ohm @ 165 / 125 / 165 µm
Zdiff 100 Ohm @ 125 / 130 / 125 µm
Zdiff 120 Ohm @ 100 / 200 / 100 µm
BGA 0.8 mm inner layers examples

Adaption on short lengths less critical, at long length adaption is needed!
BGA 0.65mm
BGA 0.65 mm

- A 0.65mm pitch BGA definitely needs µVias for the fan out
- µVias possible with 2113 or 2116 prepreg (90-110μm)
  - 50 and 100ohm are possible on µVia layers
  - 100 μm track width needed (standard)
- Critical signals should be routed on core layers below the µVia layers
  - Thinner and more constant copper thickness (higher precision), good EMC shielding
  - More possibilities to control layer distances
- Fan-out with µVias in combination with buried vias
- Min. µVia pad size 325 μm
Examples of imp. controlled tracks for BGA pitch 0.65 mm

- **S1 Zo**  50 Ohm @ 154 µm track width
- **S1 Zdiff**  90 Ohm @ 125 / 125 / 125 µm
- **S1 Zdiff**  100 Ohm @ 100 / 137 / 100 µm
- **S2 Zo**  50 Ohm @ 91 µm track width
- **S2 Zdiff**  90 Ohm @ 100 / 160 / 100 µm
- **S2 Zdiff**  100 Ohm @ 90 / 300 / 90 µm
- **S2 Zo**  50 Ohm @ 110 µm track width
- **S2 Zdiff**  90 Ohm @ 100 / 122 / 100 µm
- **S2 Zdiff**  100 Ohm @ 100 / 230 / 100 µm
- **S3 Zo**  50 Ohm @ 128 µm track width
- **S3 Zdiff**  90 Ohm @ 125 / 144 / 125 µm
- **S3 Zdiff**  100 Ohm @ 100 / 155 / 100 µm
BGA 0.5 mm

- Fan-out only possible with µVias
- Small layers distances on the µVia layers (only 1080 Prepreg min. pads size 275 µm)
- Depending on the layer order very fine line structures are needed to route the most common impedance defined structures

**better:**
- Critical signals on the core inner layers below the µVia layers (see other BGAs)
- Better EMC shielding against external sources
- Fan-out with µVias in combination with buried vias ideal

Less reflections than PTHs
BGA 0.5 mm

- Fan-out only possible with µVias
- Small layers distances on the µVia layers (only 1080 Prepreg min. pads size 275 µm)
- Depending on the layer order very fine line structures are needed to route the most common impedance defined structures

**better:**
- Critical signals on the core inner layers below the µVia layers (see other BGAs)
- Better EMC shielding against external sources
- Fan-out with µVias in combination with buried vias ideal

Less reflections than PTHs
Discontinuities in possible wiring schemes

Version 1: PTH

Version 2: Microvia / Buried Via

Blind and Buried Vias result in reduced discontinuities!
BGA 0.5 mm outer layers and inner layers µVia

- 75 µm structures in between possible with via in pad technology

<table>
<thead>
<tr>
<th>Layer</th>
<th>Foil Thickness</th>
<th>Λ1</th>
<th>Λ2</th>
</tr>
</thead>
<tbody>
<tr>
<td>S1</td>
<td>9 µm</td>
<td>1</td>
<td>25</td>
</tr>
<tr>
<td></td>
<td>12 µm</td>
<td>3.5</td>
<td>68</td>
</tr>
<tr>
<td>S2</td>
<td>9 µm</td>
<td>3.5</td>
<td>62</td>
</tr>
<tr>
<td></td>
<td>2 x 1080</td>
<td>25</td>
<td></td>
</tr>
<tr>
<td></td>
<td>17.6 µm</td>
<td>140</td>
<td>16</td>
</tr>
</tbody>
</table>

Possible on outer layers (S1) with a reduced copper thickness (appr. 25 µm)

- S1 Zo 50 Ohm @ 123 µm track width
- S1 Zdiff 100 Ohm @ 100 / 168 / 100 µm

On inner layers (S2) strongly restricted

- S2 Zo 50 Ohm @ 75 µm track width
- S2 Zdiff 99 Ohm @ 75 / 225 / 75 µm

Therefore better:
Critical signals on the core inner layers below the µVia layers (see other BGA's)
BGA 0.4 mm

- No tracks possible in between the pads
- Enough solder mask in between the solder pads
- Impedance:
  - WE suggestion like BGA 0.5mm pitch
- Critical signals on the core inner layers
- Fan-out with µVias in combination with buried vias or with PTHs outside the BGA area
BGA 0.4 mm

- No tracks possible in between the pads
- Enough solder mask in between the solder pads
- Impedance:
  - WE suggestion like BGA 0.5mm pitch
- Critical signals on the core inner layers
- Fan-out with µVias in combination with burried vias or with PTHs outside the BGA area
BGA 0.5 mm & 0.4 mm outer layers

0.5 mm BGA

S2 Zo 50 Ohm @ 89 µm track width
S2 Zdiff 90 Ohm @ 100 / 142 / 100 µm
S2 Zdiff 100 Ohm @ 75 / 125 / 75 µm

0.5 mm BGA tracks and gaps need to be dimensioned correctly outside the BGA area

better:

S2 Zo 50 Ohm @ 122 µm track width
S2 Zdiff 100 Ohm @ 100 / 150 / 100 µm
Summary

- In the fan-out of BGAs impedance requirements of the components need to be considered.
- In general, critical signals should be routed on the inner layers.
- BGA pitch 1.0 mm and 0.8 mm and larger can be routed with the help of PTHs; therefore, less restrictions in the layer stack up.
- BGA pitch 0.65 mm, 0.5 mm, and 0.4 mm need μVias for the fan-out; therefore, more restrictions in the layer stack up.
- Correct pad sizes and aspect ratio rules should be considered for each BGA fan-out so that reliability is given.
- Würth Elektronik is always ready to support you with an individual stack up according to the needed BGA.
Understanding the connections is the secret to success!

Many thanks for your interest!

Philipp Reeb
WÜRTH ELEKTRONIK GmbH & Co. KG
Produkt Management
Signalintegrität
Circuit Board Technology
T.: +49 7622 397 277

E. philipp.reeb@we-online.de
W. www.we-online.de