BASIC DESIGN RULES
LAYOUT ACCORDING TO THE WE PARAMETERS AND THE PCB FITS

Andreas Schilpp

WURTH ELEKTRONIK MORE THAN YOU EXPECT
WELCOME TO THE ANNIVERSARY WEBINAR

10 Years of Würth Elektronik PCB Webinars

- The first webinar of your Top 3, originally offered in May 2013, is more current than ever: Recently, the Würth Elektronik BASIC Design Rules have been updated and restructured.

- First webinar: 7 May 2013
- EN: 47 Attendees
AGENDA

BASIC Design Rules: Layout according to the WE parameters and the PCB fits

1. BASIC and standard – what is the difference?
2. Introduction of the new BASIC Design Rules
   ▪ Scope of application
   ▪ Contents
   ▪ What is different?
3. Background information on the changes
   ▪ Processes for copper structuring
   ▪ Specification of copper thickness according to IPC
   ▪ Aspect ratio for holes
4. Application of the BASIC Design Rules – an example

Andreas Schilpp
Technical Marketing
BASIC AND STANDARD – WHAT IS THE DIFFERENCE?

Definitions

**STANDARD**

- **a category or classification**
- available from all plants
- at a favorable standard price
- other categories are
  - Advanced
  - (Leading Edge / State-of-the-Art)
- Standard / Advanced are available in all technologies

**BASIC**

- **a technology. By BASIC technology we mean**
  - single-sided,
  - double-sided and
  - multilayer printed circuit boards.

Further example for standards

- **Standard Stackup**
  - Material in stock, processes standardized
  - Standard processes ensure high quality and favorable prices with short delivery times
**TIP
USE STANDARDS**

Use of standards:
- Standard material
- Material database of the PCB manufacturer (if available)
- Standard layer stackups also digital for import
- Standard design rules
- Standard delivery specification
- Standard packaging
- ...
The third generation in the history of PCBs is characterised by a multilayer structure, created by pressing copper foil, FR4 based copper-clad inner layer cores and pre-pregs.

The obvious additional effort and higher costs are offset by the many advantages:

- More layers to route complex components, such as BGA packages
- Possibility to design power supply systems to improve power integrity and EMC.
- Possibility to design transmission lines with defined impedance to improve signal integrity
# NEW BASIC DESIGN RULES

## Contents

<table>
<thead>
<tr>
<th>Chapter</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Scope of application</td>
<td>03</td>
</tr>
<tr>
<td>2</td>
<td>Basic notes</td>
<td>03</td>
</tr>
<tr>
<td>3</td>
<td>BASIC base material</td>
<td>03</td>
</tr>
<tr>
<td>4</td>
<td>BASIC Standard Stackups</td>
<td>03</td>
</tr>
<tr>
<td>5</td>
<td>Copper structures, spacings</td>
<td>04</td>
</tr>
<tr>
<td>6</td>
<td>Drills, drill pads, annular rings, clearances</td>
<td>06</td>
</tr>
<tr>
<td>7</td>
<td>Edge plating</td>
<td>07</td>
</tr>
<tr>
<td>8</td>
<td>Solder mask</td>
<td>07</td>
</tr>
<tr>
<td>9</td>
<td>Peelable masking</td>
<td>08</td>
</tr>
<tr>
<td>10</td>
<td>Markings</td>
<td>08</td>
</tr>
</tbody>
</table>
NEW BASIC DESIGN RULES

Scope of application

- The BASIC design parameters (copper structures or spacing, holes, solder mask, markings and solder surfaces) are basically valid for all PCB technologies, beyond BASIC for example also for
  - HDI Microvia,
  - Embedding technology or
  - Flex solutions.
NEW BASIC DESIGN RULES

Flyer

BASIC DESIGN RULES

WEBINAR | 08.02.2023

Contents

1 Scope of Application
These design rules apply in compliance to B2/B3 technology, v. 5.0. For stacked or multilayer designs, the PCB manufacturer must ensure that the design parameters are met. The design parameters are checked by the manufacturer's own quality assurance system (QA) and verified by the customer's QA department.

2 Basic Notes
- Please refer to general standards IEC 61215, IEC 627-1 and IEC 627-2 for all aspects of the solar cell.
- The design must be in compliance with the manufacturer's specifications.
- The design must meet the requirements for operating conditions as specified in the manufacturer's documentation.
- The design must meet the requirements for environmental conditions as specified in the manufacturer's documentation.
- The design must meet the requirements for mechanical conditions as specified in the manufacturer's documentation.

3 Basic Base Material

4 Basic Standard Stackups

WEBINAR | 08.02.2023
NEW BASIC DESIGN RULES

Flyer

4 COPPER STRUCTURES, SPACINGS
a. The conductor spacing, more generally the "copper spacing," is crucial for the performance of the copper conductor. The effects of interference on these transistor-shaped images must be minimized. All copper conductors must be oriented perpendicular to each other.

b. The minimum spacing between copper conductors is determined by the conductors' orientation and the conductors' dimensions. This spacing ensures that the conductors do not interfere with each other.

c. The minimum spacing between copper conductors is determined by the conductors' orientation and the conductors' dimensions. This spacing ensures that the conductors do not interfere with each other.

d. Minimum conductor widths:
   - The minimum allowed conductor width is 6 microns. For more defined conductors, such as those used in registers, the conductor widths are 4 to 6 microns.
   - A 3 micron copper conductor width is possible according to the rules for conductor widths. The minimum allowed conductor width is 1 micron, and 1 micron is a positive value for use in conductors.

5 DRILLS, DRILL PADS, ANNULAR RINGS, CLEARANCES
a. Drill diameter:
   - The PCB design specification for the drill diameter is defined by the drill size and the size of the pads. The drill size is determined by the trace width and the pad size. The drill diameter is calculated using the following formula:

   \[
   \text{Drill diameter} = \sqrt{\text{pad size} \times \text{trace width}}
   \]

b. Spacing between holes:
   - Minimum diameter between holes (based on final diameter) is calculated as a percentage of the pad size. The percentage is determined by the trace width and the pad size. The minimum allowed diameter is calculated by dividing the pad size by the trace width and multiplying the result by 100.

6 WEBINAR | 08.02.2023
NEW BASIC DESIGN RULES

Flyer

8 SOLDER MASK

- Laquer residues in via holes
- Follow-up error ENIG Surface
What has the greatest influence on the minimum copper spacing (subtractive etching technique on inner layers)?

- Number of layers
- Thickness of the copper foil
- Thickness of the electroplated metallization
- Total thickness of copper foil and electroplated metallization
- Prepreg thickness
NEW BASIC DESIGN RULES

Flyer: What is new?

a. The conductor spacing, more generally the “copper spacing”, is crucial for the production of the copper structures. This affects all features such as trace-trace / trace-shape / shape-trace / shape-shape / trace-all pin pads / trace-all Via pads / trace-all non signal geometries / etc..

b. For the design of the conductor width, criteria such as current carrying capacity, permissible tolerances and other specifications apply.

c. In principle, the conductor width can also be larger or smaller than the conductor spacing.

Example: conductor width/conductor spacing (line/space) 80 μm / 120 μm is easier to manufacture than 100 μm / 100 μm.

d. Minimum conductor widths:

i. The minimum allowed conductor width is 60 μm, for impedance defined structures smaller than 75 μm please consult us.

ii. UL marking for conductor widths <4 mil is partly only possible according to UL-94. For conductor widths ≥ 4mil, marking according to UL-94 and UL-796 is possible. We ask for your consultation.
BASICS: COPPER STRUCTURING

Inner layers: Imaging positive + etching subtractive

CUSTOMER DATA
RESIST
BASE COPPER
BASE MATERIAL
ETCHING PROCESS
NEW BASIC DESIGN RULES

Flyer: What is new?

- Table for inner layers

<table>
<thead>
<tr>
<th>Starting foil thickness</th>
<th>Minimum copper thickness$^3$</th>
<th>Minimum conductor spacing Standard</th>
<th>Minimum conductor spacing Advanced</th>
<th>Minimum possible line width</th>
</tr>
</thead>
<tbody>
<tr>
<td>17,1 µm [1/2 oz.]</td>
<td>11,4 µm</td>
<td>100 µm</td>
<td>75 µm</td>
<td>60 µm</td>
</tr>
<tr>
<td>34,3 µm [1 oz.]</td>
<td>24,9 µm</td>
<td>120 µm</td>
<td>100 µm</td>
<td>60 µm</td>
</tr>
<tr>
<td>68,6 µm [2 oz.]</td>
<td>55,7 µm</td>
<td>180 µm</td>
<td>150 µm</td>
<td>125 µm</td>
</tr>
<tr>
<td>102,9 µm [3 oz.]</td>
<td>86,6 µm</td>
<td>250 µm</td>
<td>225 µm</td>
<td>175 µm</td>
</tr>
</tbody>
</table>

$^3$ IPC-6012E-EN Table 3-14: Internal Layer Foil Thickness after Processing
BASICS: COPPER STRUCTURING

Outer layers: Imaging negative + copper deposition selective + etching base copper
NEW BASIC DESIGN RULES

Flyer: What is new?

- Table for outer layers

<table>
<thead>
<tr>
<th>Starting foil thickness</th>
<th>Minimum copper thickness$^1$</th>
<th>Minimum conductor spacing Standard</th>
<th>Minimum conductor spacing Advanced</th>
<th>Minimum possible line width</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>IPC-class 1, 2</td>
<td>IPC-class 3</td>
<td></td>
<td></td>
</tr>
<tr>
<td>8,5 μm [1/4 oz.$^2$]</td>
<td>26,2 μm</td>
<td>31,2 μm</td>
<td>100 μm</td>
<td>75 μm</td>
</tr>
<tr>
<td>12 μm [3/8 oz.$^2$]</td>
<td>29,3 μm</td>
<td>34,3 μm</td>
<td>100 μm</td>
<td>80 μm</td>
</tr>
<tr>
<td>17,1 μm [1/2 oz.]</td>
<td>33,4 μm</td>
<td>38,4 μm</td>
<td>120 μm</td>
<td>100 μm</td>
</tr>
<tr>
<td>34,3 μm [1 oz.]</td>
<td>47,9 μm</td>
<td>52,9 μm</td>
<td>180 μm</td>
<td>160 μm</td>
</tr>
<tr>
<td>68,6 μm [2 oz.]</td>
<td>78,7 μm</td>
<td>83,7 μm</td>
<td>275 μm</td>
<td>225 μm</td>
</tr>
<tr>
<td>102,9 μm [3 oz.]</td>
<td>108,6 μm</td>
<td>113,6 μm</td>
<td>390 μm</td>
<td>320 μm</td>
</tr>
</tbody>
</table>

1) IPC-6012E-EN Table 3-15: External Conductor Thickness after Plating
2) Extra cost: No standard copper foil
BASICS: PTHS AND VIAS

Why are annular rings needed around via holes?

- Relationship between via pad size, registration and drill diameter
  - Annular ring specifications of IPC classes (see IPC-2221)
  - Influence of the drill tool: deflection
  - Influence of the machine
  - Layout-dependent shrinkage/expansion values: Offset of layers relative to each other
- Ideal versus real

- Important for the drill diameter: Aspect ratio - next slide
BASICS: PTHS AND VIAS

Aspect Ratio

- Ratio drill-diameter $b$ – drill depth $a$
- PTH & BV: max. 1:8

- Important for a complete and even copper build-up in the barrel.

Example PTH:
- $b = ?$
- $a = 1.6\text{mm}$
- $b = a / 8 = 0.20\text{mm}$
  - Smallest drill-$\text{Ø} = 0.20\text{mm}$
  - Smallest pad-$\text{Ø} = 0.45\text{mm}$
## BASICS: PTHS AND VIAS

### Table

<table>
<thead>
<tr>
<th>Pad size</th>
<th>Remark</th>
<th>Aspect Ratio[1]</th>
<th>Drill tool diameter</th>
<th>Finished hole diameter</th>
<th>Tolerance (Standard)</th>
<th>Copper clearance plane on inner layer without Pad</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.60 mm</td>
<td>Standard</td>
<td>8:1</td>
<td>0.35 mm</td>
<td>0.25 mm</td>
<td></td>
<td>≥ 0.80 mm</td>
</tr>
<tr>
<td>0.55 mm</td>
<td></td>
<td></td>
<td>0.30 mm</td>
<td>0.20 mm</td>
<td>+0.1/-0.05 mm</td>
<td>≥ 0.75 mm</td>
</tr>
<tr>
<td>0.50 mm (Cu max. 35 μm)</td>
<td>Max. ca. 12 layers Max. ca. 1.80 mm PCB thickness</td>
<td></td>
<td>0.25 mm</td>
<td>0.15 mm</td>
<td></td>
<td>≥ 0.70 mm</td>
</tr>
<tr>
<td>0.45 mm (Cu max. 35 μm)</td>
<td>For less complex Layer stackups</td>
<td></td>
<td>0.25 mm (0.20 mm)</td>
<td>0.15 mm</td>
<td></td>
<td>≥ 0.70 mm</td>
</tr>
</tbody>
</table>

1) Aspect Ratio[1] for drill holes: Ratio of drill hole length or depth to drill hole tool diameter. For further information, see technical delivery specification chapter 3.7.1.
BASICS: PTHS AND VIAS

Why do you need copper releases in ground inner layers without a pad?

- Non-used / non-functional pads:
  - Pad removal does not add space on inner layers because tolerances, offset and deflection are real and permissible.
  - A design rule check must be performed before pad removal!
  - The total clearance is pad size plus minimum copper clearance!

- Annular ring

- ATTENTION with RIGID.flex: Do NOT remove non-used / non-functional via pads on flex layers for reliability reasons!
BASIC DESIGN RULES: LAYOUT ACCORDING TO THE WE PARAMETERS

Usage of table for UFBGA 7X7X0.6 169L P 0.5mm

**TOP**
- Layer 2
- Layer 3
- Layer 4
BASIC DESIGN RULES: LAYOUT ACCORDING TO THE WE PARAMETERS

Poll: Multiple choice with only one correct answer

Please rate the BASIC Design Rules: How many points on a scale of 1 to 10 do you give the document?

- 1 to 2 points (unusable)
- 3 to 4 points (incomplete and incorrect)
- 5 to 6 points (ok, but incomplete)
- 7 to 8 points (good, helpful, understandable)
- 9 to 10 points (almost perfect, I use it often)

NOTES

- Participants with a rating of less than 5 points I will contact in writing after the webinar (this is not a threat, but a partnership offer!)
- Feel free to write me errors, additions and optimizations in the question field
SUMMARY

New BASIC Design Rules

- BASIC is a technology
- Standard is a category or classification
- Conductor spacing is important for PCB manufacture
- Conductor width and conductor spacing should be considered independently of each other. Different values should be considered at least locally.
- Annular rings around via holes are still necessary
- Non-used pad removal does not save space

- Würth Elektronik Webinars – more than you expect!
MANY THANKS FOR YOUR ATTENTION!