HOW DOES YOUR PCB LAYOUT INFLUENCE THE COSTS IN PCB MANUFACTURING?

Jürgen Wolf
Co-Authors: Andreas Dreher – Holger Krause – Andreas Nies – Jens Töbeck

WURTH ELEKTRONIK MORE THAN YOU EXPECT
AGENDA

How does your PCB layout influence the costs in PCB manufacturing?

1. PCB size and arrays
2. Copper price development and choice of materials
3. PCB stackup
4. Mechanical processing
5. Enhanced Technologies
6. More tips & tricks
7. Summary
AGENDA

How does your PCB layout influence the costs in PCB manufacturing?

1. PCB size and arrays
2. Copper price development and choice of materials
3. PCB stackup
4. Mechanical processing
5. Enhanced Technologies
6. More tips & tricks
7. Summary
THE SIZE OF THE PRINTED CIRCUIT BOARD

It's all a question of space!

The size of the PCB

- The smaller the PCB, the higher the number of PCBs on the production panel

- Error-proneness vs. yield:
  The smaller the PCBs, the better the yield per PCB, the lower the overhead to serve the desired delivery quantity

- Sustainability:
  The less material used, the smaller the footprint of the LP.
THE PCB ARRAY

How to utilize and occupy the manufacturing panel properly?

How is the manufacturing panel occupied with PCBs?

Background information:

- PCB materials are manufactured in large panels
  90% of EU and US manufacturers of FR4 uses these formats:
  - US-Format: 1.225 x 925 mm²
  - Uni-Format: 1.225 x 1.070 mm²

- 95% of PCB manufacturers in EU & US use these panel formats:
  - 460 x 305 mm² (1/8 US-Format) WE sample format
  - 606 x 458 mm² (1/4 US-Format) WE standard format
  - 606 x 528 mm² (1/4 Uni-Format) WE jumbo format
THE PCB ARRAY

How to utilize and occupy the manufacturing panel properly?

How is the manufacturing panel occupied with PCBs?

- Every PCB manufacturer needs a border for registration and labelling → Non-useable space!

Example: Single PCBs

routing or V-scorining

In this example: 33% more circuit boards on the production panel
THE PCB ARRAY

How to utilize and occupy the manufacturing panel properly?

How is the manufacturing panel occupied with PCBs?

- Every PCB manufacturer needs a border for registration and labelling → Non-useable space!

Example: Single PCBs – The smaller the PCB, the greater the effect!

<table>
<thead>
<tr>
<th>routing</th>
<th>or</th>
<th>V-scoring</th>
</tr>
</thead>
</table>

In this example: 56 PCBs vs. 85 PCBs
THE PCB ARRAY

How to utilize and occupy the manufacturing panel properly?

How is the manufacturing panel occupied with PCBs?

- Every PCB manufacturer needs a border for registration and labelling → Non-useable space!

Example: PCBs in array

routing or V-scoring

In this example: 100% more circuit boards on the production panel
THE PCB ARRAY

How to utilize and occupy the manufacturing panel properly?

Calculation basis:

- ML6 / Base material $T_g$ 150
- PCB size 20 x 40 mm²
- Array size 210 x 140 mm²
- 100 µm L/S
- 500 drills
- 0,20 mm smallest drill-Ø
- ENIG

PCBs on a production panel
Number of production panels (1,000 PCBs ordered) 216
PCBs in an array 5
PCB outline v-scored
PCB distance in array 0,00 mm
Price indicator 100%

160
7
20
5,00 mm
117%

128
8
16
10,00 mm
131%
THE PCB ARRAY

How to utilize and occupy the manufacturing panel properly?

How is the manufacturing panel occupied with PCBs?

<table>
<thead>
<tr>
<th>WE-Format</th>
<th>Sample format</th>
<th>Standard format</th>
<th>Jumbo format</th>
</tr>
</thead>
<tbody>
<tr>
<td>Technologies</td>
<td>All technologies</td>
<td>Basic, rigid-flex &amp; HDI</td>
<td>Basic &amp; HDI</td>
</tr>
<tr>
<td>Plant</td>
<td>Rot am See</td>
<td>Niedernhall</td>
<td>Schopfheim</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Used in Schopfheim for special constructions</td>
<td>Niedernhall on demand</td>
</tr>
<tr>
<td>Panel size</td>
<td>460 x 305 mm²</td>
<td>606 x 458 mm²</td>
<td>606 x 528 mm²</td>
</tr>
<tr>
<td>Usable area</td>
<td>426 x 271 mm²</td>
<td>572 x 424 mm²</td>
<td>570 x 500 mm²</td>
</tr>
<tr>
<td>Number of arrays</td>
<td>1</td>
<td>426 x 271 mm²</td>
<td>570 x 500 mm²</td>
</tr>
<tr>
<td>Best array for v-scored outlines are</td>
<td>572 x 424 mm²</td>
<td>500 x 285 mm²</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>426 x 271 mm²</td>
<td>213 x 135 mm²</td>
<td>285 x 250 mm²</td>
</tr>
<tr>
<td>2</td>
<td>271 x 213 mm²</td>
<td>286 x 212 mm²</td>
<td>250 x 190 mm²</td>
</tr>
<tr>
<td>4</td>
<td>213 x 135 mm²</td>
<td>212 x 190 mm²</td>
<td>250 x 190 mm²</td>
</tr>
<tr>
<td>6</td>
<td>142 x 135 mm²</td>
<td>142 x 135 mm²</td>
<td>142 x 190 mm²</td>
</tr>
<tr>
<td>8</td>
<td>135 x 106 mm²</td>
<td>212 x 143 mm²</td>
<td>250 x 142 mm²</td>
</tr>
<tr>
<td>9</td>
<td>142 x 90 mm²</td>
<td>142 x 90 mm²</td>
<td>142 x 166 mm²</td>
</tr>
<tr>
<td>12</td>
<td>106 x 90 mm²</td>
<td>142 x 90 mm²</td>
<td>142 x 166 mm²</td>
</tr>
<tr>
<td>15</td>
<td>90 x 85 mm²</td>
<td>141 x 114 mm²</td>
<td>141 x 114 mm²</td>
</tr>
</tbody>
</table>

Tipps:
- Edge of array min. 5 mm
- Edge of array 8 - 10 mm for routed outlines
- 2 edges with 5 – 10 mm for v-scored outlines
- Size of array should be based on thickness of PCB (the thinner the smaller)
AGENDA

How does your PCB layout influence the costs in PCB manufacturing?

1. PCB size and arrays
2. Copper price development and choice of materials
3. PCB stackup
4. Mechanical processing
5. Enhanced Technologies
6. More tips & tricks
7. Summary
DEVELOPMENT OF COPPER PRICE

Role of material price in PCB price

Copper price

Developments on the London commodity exchange

Time period: Jan. 2016 to April 2023

DOUBLED from 2016 to date

Source: http://www.boerse.de – data downloaded on 03.04.2023
DEVELOPMENT OF COPPER PRICE

Role of material price in PCB price

Comparison of material purchasing prices for FR4  (as of July 2020 / as of April 2023)

Copper plays an important role in the price of PCBs!

Hence the question: What is necessary or what is possible?
MATERIAL PRICE

Role of material price in PCB price

Comparison of the different solder surfaces

<table>
<thead>
<tr>
<th>Solder Surface</th>
<th>Cost Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>HAL</td>
<td>1.34</td>
</tr>
<tr>
<td>HAL LF</td>
<td>1.00</td>
</tr>
<tr>
<td>Chem. Sn 0.8µm</td>
<td>1.18</td>
</tr>
<tr>
<td>Chem. Sn 1µm</td>
<td>1.33</td>
</tr>
<tr>
<td>Chem. Ag</td>
<td>1.44</td>
</tr>
<tr>
<td>ENIG</td>
<td>2.50</td>
</tr>
<tr>
<td>Plated Au ENEPIG</td>
<td></td>
</tr>
</tbody>
</table>

- **ENIG**
  - Ni 5.5µm
  - Au 0.075µm

- **Electroplated Au**
  - Ni 4-7µm
  - Au 1-3 µm
  - << Gold 40 times thickness>>

- **ENEPIG**
  - Ni 4-7µm
  - Au bis 0.06µm
  - Pd 0.05 bis 0.25µm
  - << Palladium on top >>
MATERIAL PRICE

Role of material price in PCB price

Electroplated Gold

Usage of electroplated Gold
- often used for contacts as an abrasion resistant surface
- mostly selective in combination with ENIG
- with thicknesses up to 4 µm

Price indicator: up to 500% or more (depends on the current price of gold)

Source: http://www.boerse.de – data downloaded on 03.04.2020
AGENDA

How does your PCB layout influence the costs in PCB manufacturing?

1. PCB size and arrays
2. Copper price development and choice of materials
3. PCB stackup
4. Mechanical processing
5. Enhanced Technologies
6. More tips & tricks
7. Summary
How does the PCB construction influence the price?

Comparison of a 4-layer multilayer with different thicknesses

- **Standard:** 1.55 mm / 1.60 mm
- **Optimum:** 1.00 mm
- **Further standards:** 0.80mm / 2.00 mm / 2.40 mm

**Price indicators**
- 107% for 0.50mm thickness
- 96% for 1.00mm thickness
- 100% for 1.60mm thickness
- 137% for 3.20mm thickness

**Layers and Materials**
- ML4_TG150_0.50_35: 0.10mm-035+035, 4x prepreg 1080
- ML4_TG150_1.00_35: 0.41mm-035+035, 4x prepreg 2116
- ML4_TG150_1.60_35: 0.71mm-035+035, 4x prepreg 7628
- ML4_TG150_3.20_35: 2.40mm-035+035, 4x prepreg 7628
How does the PCB construction influence the price?

Comparison of a 6-layer multilayer: 1,60 mm standard vs. individual stackup

**Standard stackup**
- 2x 0.36mm-035+035
- 6x prepreg 2116

**Specific stackup**
- 2x 0.10mm-035+035
- 6x prepreg 2116
- 8x prepreg 7628

**Price indicator**
- Standard stackup: 100%
- Specific stackup: 116%

Additional costs:
- Handling of thin laminate
- 4 prepregs more in stackup
How does the PCB construction influence the price?

Comparison of a 6-layer multilayer: 1,60 mm standard vs. individual stackup

**Standard stackup**
- 2x 0.36mm-035+035
- 6x prepreg 2116
- Price indicator 100%

**Core-based stackup**
- 3x 0.20mm-035+035
- 4x prepreg 2116
- 2x prepreg 7628
- Price indicator 122%

**Additional cost:**
- Multiple exposures of the outer layer cores (processed like an 8-layer PCB)
- More cores

**Further cost drivers**
- Filling cores in stackup
AGENDA

How does your PCB layout influence the costs in PCB manufacturing?

1. PCB size and arrays
2. Copper price development and choice of materials
3. PCB stackup
4. Mechanical processing
5. Enhanced Technologies
6. More tips & tricks
7. Summary
MECHANICAL PROCESSING

Which influence do the drilling tools have on the PCB costs?

Panel stacking for mechanical processing

using „via drilling“ as an example

<table>
<thead>
<tr>
<th>Tool life</th>
<th>Panel stacking</th>
<th>Time to drill 15.000 vias</th>
<th>Usage of drill bits for 15.000 vias</th>
<th>Price indicator (drilling process only)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.250 strokes</td>
<td>stack of 3</td>
<td>0,2 h</td>
<td>4</td>
<td>100%</td>
</tr>
<tr>
<td>1.000 strokes</td>
<td>stack of 2</td>
<td>0,4 h</td>
<td>7,5</td>
<td>200%</td>
</tr>
<tr>
<td>500 strokes</td>
<td>stack of 1</td>
<td>0,8 h</td>
<td>30</td>
<td>460%</td>
</tr>
</tbody>
</table>
MECHANICAL PROCESSING

Which influence do the drilling tools have on the PCB costs?

Comparison:

Ø 0,5 mm, Ø 0,35 mm und Ø 0,25 mm drill bits on 5 mm x 5 mm checkered paper
MECHANICAL PROCESSING

Which influence do the drilling tools have on the PCB costs?

- Ø 0.2 mm (0.55 € per bit)
  Tool life: 750 strokes
  Drilling frequency: 3 / s

- Ø 0.3 mm (0.50 € per bit)
  Tool life: 1,000 strokes
  Drilling frequency: max. 8 / s

- Microvia
  Ø 0.125 mm
  Drilling frequency: 150–180 / s
MECHANICAL PROCESSING

Copper- or resin-filled Microvias

Filling of Microvias or not? This is the question here!

IPC-7095C – Table A-3 – Class III:
Max. „22% of the image diameter”

The formation of voids depends, among other things, on:
- Flux / solder paste
- Temperature profile of the solder process
- Uniform heating or through-heating of the circuit board

Every user has to define for himself how to manufacture!
MECHANICAL PROCESSING

Copper-filled Microvias

Sequence Cu-filling process (Source/publication: MacDermid Enthone Electronic Solutions / 2018)

1. Wetter/Suppressor: Molecules occupy the surface and block the deposition of Cu
2. Leveler: Molecules accumulate at the location of the highest current density and block the deposition of Cu
3. Brightener: Brightener for the reduction of Cu crystal sizes

Time sequence (Source/publication: KAIST / 2019)

Process takes factor 2-3 longer compared to standard
MECHANICAL PROCESSING

What influence does the HDI layer construction have?

- Microvias from layer 1 to layer 3
- Internal staggered Microvias
- mech. drilled buried vias

1. Single lamination
2. Two sequential laminations
3. Three sequential laminations

<table>
<thead>
<tr>
<th>Complexity</th>
<th>Cost Percentage</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 + 6 + 1</td>
<td>100%</td>
</tr>
<tr>
<td>2 + 4 + 2</td>
<td>115%</td>
</tr>
<tr>
<td>1 + 6b + 1</td>
<td>120%</td>
</tr>
<tr>
<td>2 + 4(6b) + 2</td>
<td>142%</td>
</tr>
<tr>
<td>1 + 6b + 1</td>
<td>150%</td>
</tr>
<tr>
<td>2 + 4b + 2</td>
<td>175%</td>
</tr>
</tbody>
</table>

WEBINAR | 05.04.2023
MECHANICAL PROCESSING

Which influence do the routing tools have on the PCB costs?

Panel stacking for mechanical processing

using “routing” as an example

<table>
<thead>
<tr>
<th>Tool life</th>
<th>Tool diameter (mm)</th>
<th>Panel stacking</th>
<th>Price indicator</th>
</tr>
</thead>
<tbody>
<tr>
<td>15 m</td>
<td>2,00</td>
<td>stack of 3</td>
<td>100%</td>
</tr>
<tr>
<td>5 m</td>
<td>1,50</td>
<td>stack of 2</td>
<td>270%</td>
</tr>
<tr>
<td>3 m</td>
<td>1,00</td>
<td>stack of 1</td>
<td>1200%</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Feed</th>
<th>Tools for 100 PCBs with 0,5m routing path</th>
</tr>
</thead>
<tbody>
<tr>
<td>1,2 m/min</td>
<td>1,1</td>
</tr>
<tr>
<td>0,7 m/min</td>
<td>5</td>
</tr>
<tr>
<td>0,3 m/min</td>
<td>16,7</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Time to route 100 PCBs with 0,5m routing path</th>
</tr>
</thead>
<tbody>
<tr>
<td>0,2 h</td>
</tr>
<tr>
<td>0,6 h</td>
</tr>
<tr>
<td>2,8 h</td>
</tr>
</tbody>
</table>
MECHANICAL PROCESSING

What else has an influence on the price of PCBs?

Routing contours

Complex routing contours can lengthen the routing paths and have a negative influence on the routing tool diameter.

- Standard routing contour:
  - 4x change in direction
  - routing tool 2.4 mm

- Complex routing contour:
  - approx. 30x change in direction
  - high routing time
  - routing tool 1.8 mm

HOW DOES YOUR PCB LAYOUT INFLUENCE THE COST IN PCB MANUFACTURING?
WEBINAR | 05.04.2023
AGENDA

How does your PCB layout influence the costs in PCB manufacturing?

1. PCB size and arrays
2. Copper price development and choice of materials
3. PCB stackup
4. Mechanical processing
5. Enhanced Technologies
6. More tips & tricks
7. Summary
ENHANCED TECHNOLOGIES

What else has an influence on the price of PCBs?

Further cost drivers!

- PCB thickness / layer count
  → not only relevant for drilling & routing...

- Number of laminations

- Edge plating / side plating

- Castellated holes / Castellation
ENHANCED TECHNOLOGIES

What else has an influence on the price of PCBs?

Further cost drivers!

- Coloured solder resists
  - White / black / red / blue
  - The problem: demand extremely low
  - Question: Does it always have to be solder resist - or is it sufficient to mark PCBs with a coloured legend printing (e.g. additional red/yellow for prototypes / samples without series approval)

- Legend printing
  - How small must be printed?
  - Danger: printing onto pads
What else has an influence on the price of PCBs?

**Required or needed impedances**

- Impedance watching / controlled dielectric
  - Calculated stackup and tracks, no TDR coupons

- Impedance control
  - additional TDR coupon (or coupons) on manufacturing panel
  - reduced number of PCBS on manufacturing panel
ENHANCED TECHNOLOGIES

What else has an influence on the price of PCBs?

Filled and Capped Via (IPC 4761 – Type VII Via)

Via filled with resin and over-plated with Cu

Necessary or to be avoid with intelligent design?

usage:
  e.g. cooling areas
  vias in solder pads
  vacuum tightness
ENHANCED TECHNOLOGIES

What else has an influence on the price of PCBs?

Request: IPC Class 3

The requirement of 25µm copper in the barrel is often mistaken with the requirement of IPC Class 3 production:

- 25µm barrel copper is only a part of the requirement of IPC Class 3

- Tougher test criteria in accordance with IPC Class 3 lead to
  - a lower yield
  - less space on the production panel due to more coupons
  - higher inspection effort, e.g. evaluation of several coupons to check the "inner values"
  - higher price!
ENHANCED TECHNOLOGIES

How about rigid-flex stackups?

Flex-Rigid 1F-5Ri / HDI 1-4-1

- Single sided vs. double sided effort for mechanical depth milling
- Huge price differences for the flex material: copper on one or both sides
- Screen-printed flexible solder resist is cheaper than routed and laminated coverlay
- For higher reliability with xRi-2F-xRi: Partial coverlay (Bikini coverlay) required

Flex-Rigid 2Ri-2F-2Ri / HDI 1-4-1
AGENDA

How does your PCB layout influence the costs in PCB manufacturing?

1. PCB size and arrays
2. Copper price development and choice of materials
3. PCB stackup
4. Mechanical processing
5. Enhanced Technologies
6. More tips & tricks
7. Summary
**MORE TIPS & TRICKS**

**Complexity**

**Total Cost of Ownership / System costs using the example of a system with 4 PCBs**

<table>
<thead>
<tr>
<th>Rigid PCBs with connectors</th>
<th>Rigid-flex PCB</th>
<th>Profit</th>
</tr>
</thead>
<tbody>
<tr>
<td>▪ 4 layouts</td>
<td>▪ 1 layout</td>
<td>▪ Only one layout routing logic</td>
</tr>
<tr>
<td>▪ handling of 7 parts</td>
<td>▪ handling of 1 part</td>
<td>▪ No uneven quantities of parts in storage</td>
</tr>
<tr>
<td>▪ min. 4 soldering processes</td>
<td>▪ min. 1 soldering process</td>
<td>▪ Less set-up time during assembly</td>
</tr>
<tr>
<td>▪ min. 4 test runs</td>
<td>▪ min. 1 test run</td>
<td>▪ Less time spent in testing</td>
</tr>
<tr>
<td>▪ logistics for 7 parts</td>
<td>▪ logistics for 1 part</td>
<td>▪ Logistics and handling simplified in complexity</td>
</tr>
</tbody>
</table>

**Profit**

- Only one layout routing logic
- No uneven quantities of parts in storage
- Less set-up time during assembly
- Less time spent in testing
- Logistics and handling simplified in complexity
MORE TIPS & TRICKS

Reduce prejudices!

An HDI PCB is too expensive! IS that right? Fact check!

Fact 1 – Fan-out | Based on BGA with Pitch 0,8mm / 400 Pins / 10 rows

- Plated Through Holes
- HDI-Microvia

- By using Microvias, the number of fan-out layers required can be significantly reduced

Fact 2

By using Microvias, the LP size can usually be reduced by 10-20%

⇒ More PCBs on a production panel
MORE TIPS & TRICKS

Error prevention

Pad connections to close to the pad

Avoiding pseudo errors in DRC or AOI
MORE TIPS & TRICKS

Error prevention

Clearance to outline of planes, lines and holes incl. pads

Follow the design rules!
Non-compliance often leads to non-manufacturability or at least higher costs!
MORE TIPS & TRICKS

Special processes

Special request in packaging

**Manufacturing**
- Double-sided PCB semi-circular
- 198 PCBs per production panel
- Order quantity: 2,600 PCBs → 14 production panel

**First packaging unit**
- 4 PCBs on top of each other
- Fixed by hand with an adhesive strip

**Second packaging unit**
- 10 packages of the first unit
- Dimensions of base carton exactly specified

Result: 10 hours of packaging effort
AGENDA

How does your PCB layout influence the costs in PCB manufacturing?

1. PCB size and arrays
2. Copper price development and choice of materials
3. PCB stackup
4. Mechanical processing
5. Enhanced Technologies
6. More tips & tricks
7. Summary
HOW DOES YOUR PCB LAYOUT INFLUENCE THE COSTS IN PCB MANUFACTURING

Things to consider when manufacturing PCBs:

<table>
<thead>
<tr>
<th>design complexity</th>
<th>materials</th>
<th>processes</th>
<th>sizes / volumes</th>
<th>further more</th>
</tr>
</thead>
<tbody>
<tr>
<td>application components</td>
<td>copper</td>
<td>number of processes</td>
<td>outline</td>
<td>cost for tools</td>
</tr>
<tr>
<td>layer count</td>
<td>glass types</td>
<td>sequential built-ups</td>
<td>panel occupation</td>
<td>packaging e.g. MBB</td>
</tr>
<tr>
<td>structures lines/spaces</td>
<td>resins (Tg, PI, RF...)</td>
<td>via filling</td>
<td>design of array</td>
<td>test/ qualifications e.g. PPAP</td>
</tr>
<tr>
<td>Via-types dimensions blind/buried</td>
<td>solder surfaces (ENIG etc.)</td>
<td>YIELD time</td>
<td>X-out rules</td>
<td>registration e.g. UL</td>
</tr>
</tbody>
</table>
THANK YOU VERY MUCH FOR YOUR ATTENTION!

What kind of Application do you have? How can WE support You?

Contact:
Würth Elektronik GmbH & Co. KG
Advanced Solutions Center
+49 7940 946-1234
asc@we-online.com