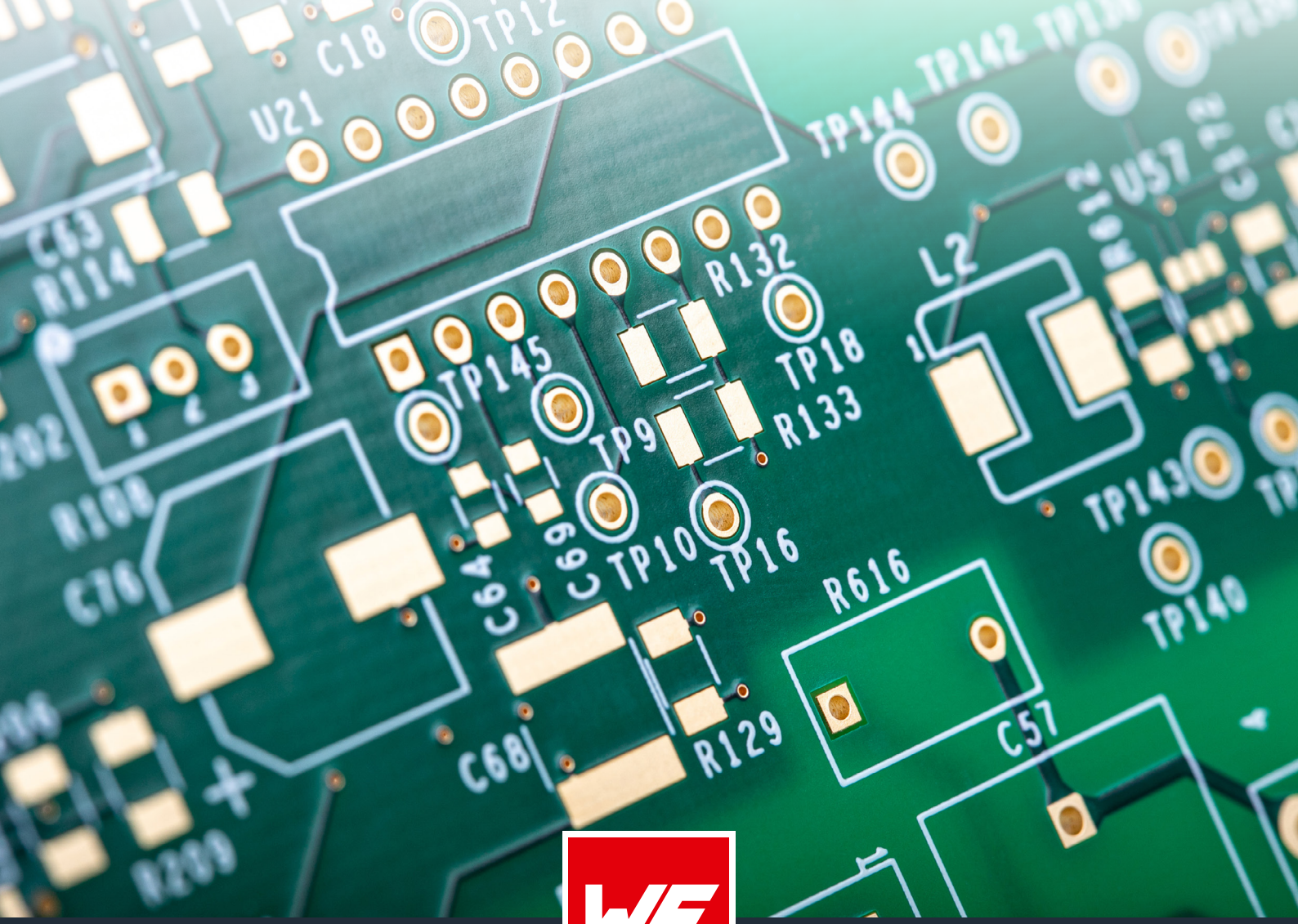
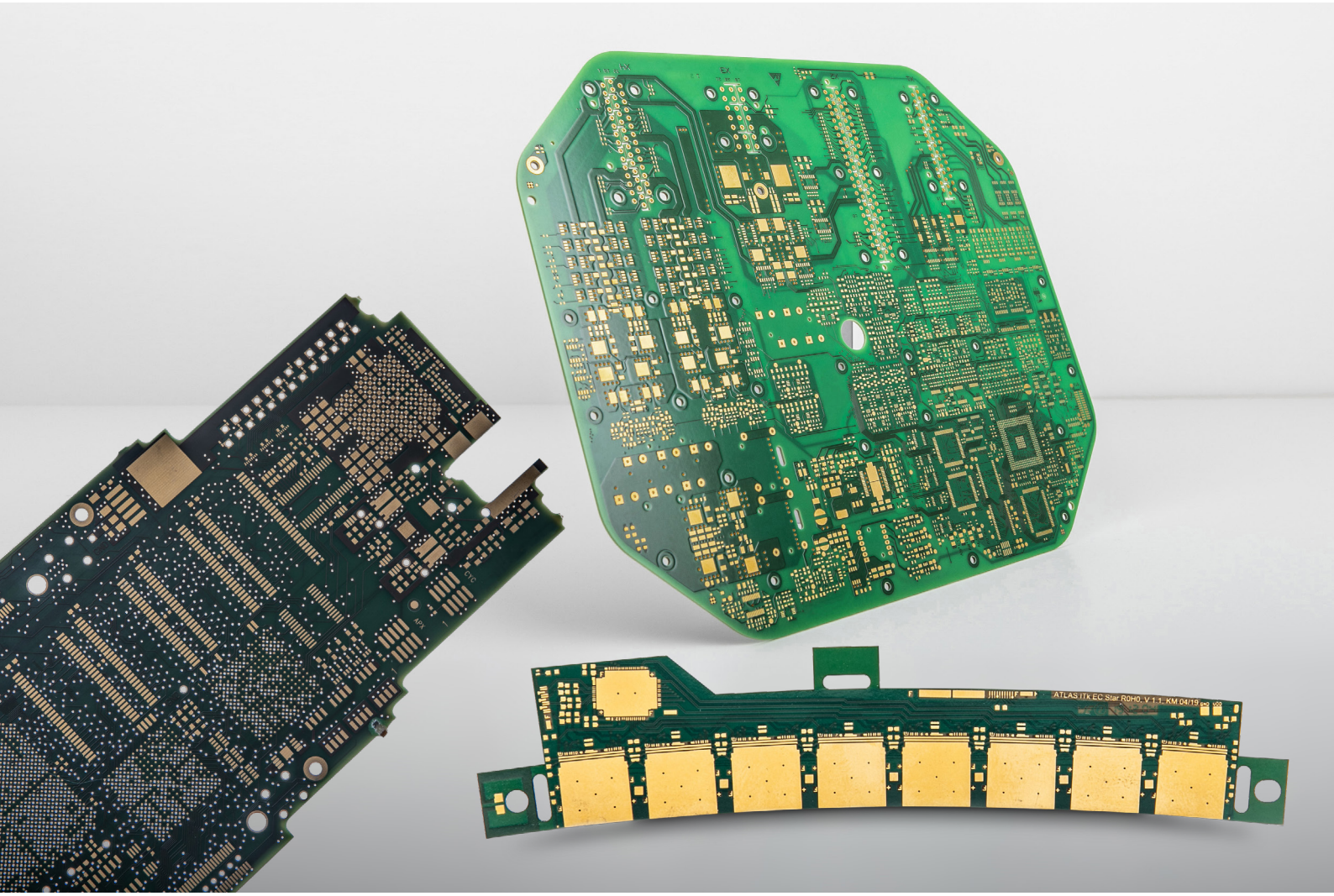


BASIC
DESIGN RULES

EN

WÜRTH ELEKTRONIK MORE THAN YOU EXPECT

CONTENTS



1 SCOPE OF APPLICATION

These design rules apply in principle to BASIC technology, i.e. for single-sided, double-sided and multilayer PCBs. The design parameters (copper structures or spacing, holes, solder mask, markings and solder surfaces) also apply to all PCB technologies, for example HDI Microvia, Embedding Technology or Flex solutions.

2 BASIC NOTES

a. Please refer to general standards like IPC or IEC.
   As standard we manufacture according to IPC-A-600 class 2.
b. Please refer to our **technical delivery specification** (www.we-online.com/tds)
c. Holes in soldering surfaces:
   Do not use open holes in soldering surfaces! Keep a distance of at least 400 µm distance on both sides from solder pads to holes that are to be plugged (plugged via, IPC type III). For vias according to IPC type VII (filled and capped) please consult us for allowed design rules (conductor spacing)!
d. Plugging und Filling
   In our **Design Guide "Plugging/Filling/Tenting"** (https://www.we-online.com/designguidepluggingfillingtenting) we have, in addition to basic definitions of terms, also explained the individual via variants with their respective goals clearly summarized for you. This will help you to find the right solution for each of your applications - within IPC-4761 and beyond.
e. We will be happy to create an optimum delivery panel for you (best price!)

3 BASIC BASE MATERIAL

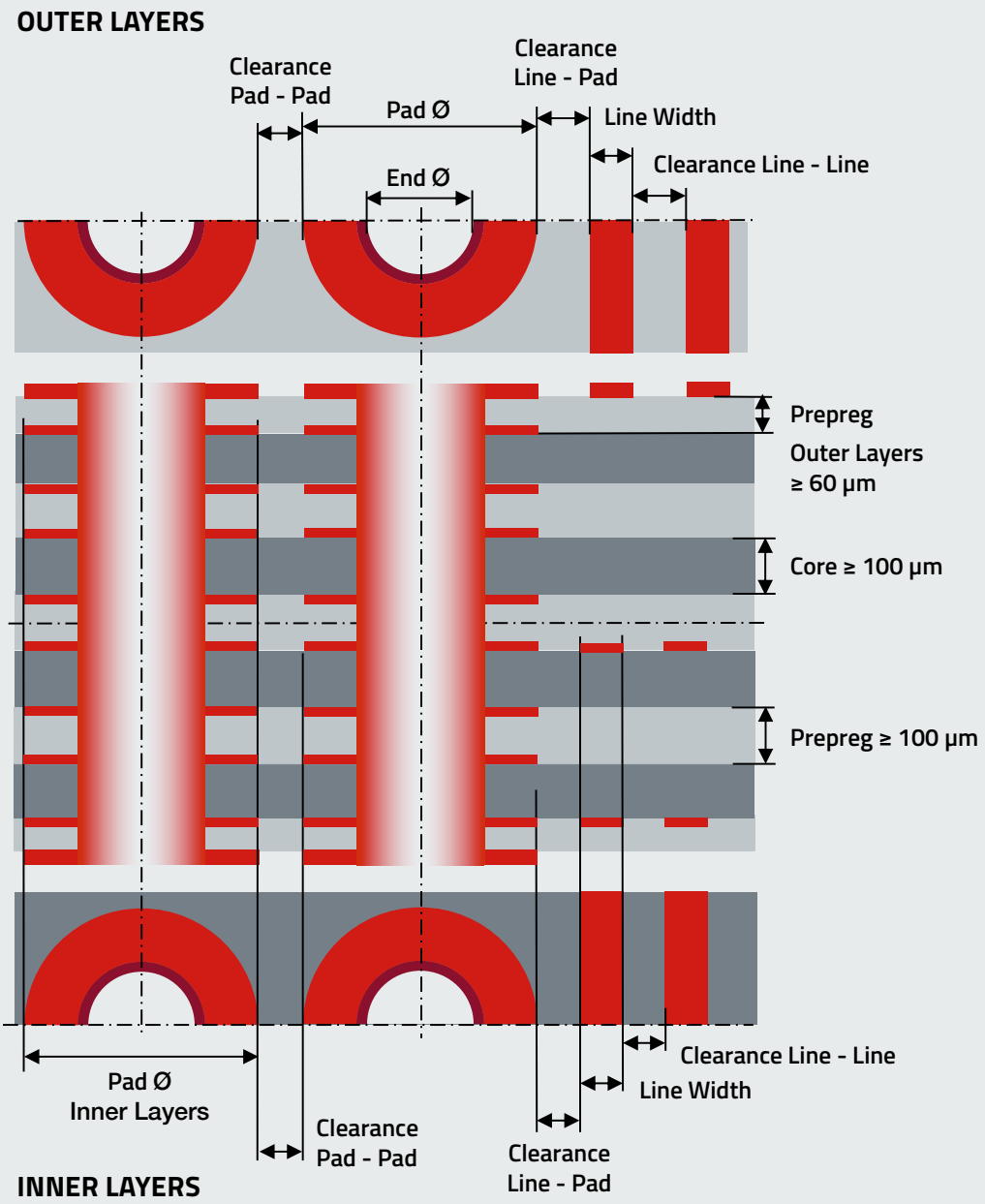
All base materials used are IPC compliant. Rigid base materials are specified in IPC-4101 and its specification sheets. For standard FR-4.0 with Tg135, for example, specification sheet 21 applies, higher grade FR-4.1 for example in specification sheets 128 (92, 94, 127) for elevated operating temperatures with Tg150 and low CTE (Coefficient of Thermal Expansion). Our portfolio is rounded off with CEM-1 materials (specification sheets 10, 12, 14, 15, 81) and CEM-3 materials (specification sheets 16, 35).

4 BASIC STANDARD STACKUPS

For the BASIC technology, we offer **standard stackups** on our website that are usual in the market and cost-optimized. Here you will also find all standards as digital stackup files for import into your EDA software. You can use these standards as a basis for your solutions and modify them according to your requirements.

5 COPPER STRUCTURES, SPACINGS

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 <4mil 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.



The derivation and illustration of the table values has changed compared to the old documents.

1. For outer layers, the copper foil thickness before metallization is selected here in accordance with IPC-6012 and no longer the final copper layer thickness.
2. The column "nominal thickness" is for orientation only
3. In addition to the STANDARD category (possible in all plants), there is now another ADVANCED category.
4. The minimum strucutres have not worsened in any case, are now possibly to be found in the new category ADVANCED.

Outer layers – conductor spacing						
Starting foil thickness	**Minimum copper thickness[1]**		**Nominal final thickness**	**Minimum conductor spacing Standard**	**Minimum conductor spacing Advanced**	**Minimum possible line width**
	IPC-class 1, 2	IPC-class 3				
8,5 µm [1/4 oz.][2]	26,2 µm	31,2 µm		100 µm	75 µm	60 µm[3]
12 µm [3/8 oz.][2]	29,3 µm	34,3 µm		100 µm	80 µm	60 µm[3]
17,1 µm [1/2 oz.]	33,4 µm	38,4 µm	35 µm	120 µm	100 µm	60 µm[3]
34,3 µm [1 oz.]	47,9 µm	52,9 µm	70 µm	180 µm	160 µm	120 µm
68,6 µm [2 oz.]	78,7 µm	83,7 µm	105 µm	275 µm	225 µm	125 µm
102,9 µm [3 oz.]	108,6 µm	113,6 µm		390 µm	320 µm	150 µm
1) IPC-6012E-EN Table 3-15: External Conductior Thickness after Plating						
2) Extra cost: No standard copper foil						
3) Outer layers: only possible with uniform circuit layout						
Inner layers - conductor spacing						
Starting foil thickness	**Minimum copper thickness[4]**			**Minimum conductor spacing Standard**	**Minimum conductor spacing Advanced**	**Minimum possible line width**
	IPC-class 1, 2, 3					
17,1 µm [1/2 oz.]	11,4 µm			100 µm	75 µm	60 µm[3]
34,3 µm [1 oz.]	24,9 µm			120 µm	100 µm	60 µm[3]
68,6 µm [2 oz.]	55,7 µm			180 µm	150 µm	125 µm
102,9 µm [3 oz.]	86,6 µm			250 µm	225 µm	175 µm
4) IPC-6012E-EN Table 3-14: Internal Layer Foil Thickness after Processing						

Tolerances of copper structures
Outer layers: see Technical delivery specification chapter 3.3
Inner layers: see Technical Delivery Specification chapter 4.3

6 DRILLS, DRILL PADS, ANNULAR RINGS, CLEARANCES

Plated Through Holes						
Pad size	**Remark**	**Aspect Ratio[1]**	**Drill tool diameter**	**Finished hole diameter**	**Tolerance (Standard)**	**Copper clearance plane on inner layer without Pad**
0,60 mm	**Standard**	≤ 8:1	0,35 mm	**0,25 mm**	+0,1/-0,05 mm	≥ 0,80 mm
0,55 mm			0,30 mm	**0,20 mm**		≥ 0,75 mm
0,50 mm (Cu max. 35 µm)	Max. ca. 12 layers Max. ca. 1,80 mm PCB thickness		0,25 mm	**0,15 mm**		≥ 0,70 mm
0,45 mm (Cu max. 35 µm)	For less complex Layer stackups		0,25 mm (0,20 mm)	**0,15 mm**		≥ 0,70 mm

1) Aspect Ratio" 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.

a. Drill diameter

The PCB design specifies the via design by defining the via hole size and the via pad size. The hole size in the manufacturing data represents the final diameter that is specified for the finished PCB. For the drill tool (drill tool diameter), a larger diameter is always selected for the drilling tool (drill tool diameter), because the hole diameter becomes smaller after drilling due to the deposition of copper and solder surface in the hole barrel. Therefore, the pad size must also not be selected too small.

b. Spacing between holes

Minimum distances between holes (based on final diameter)	
Hole-to-hole clearance same potential	300 µm
Hole-to-hole clearance different potential	500 µm
Distance NPTH-NPTH (Non Plated Through Hole)	350 µm

c. Copper layer thicknesses in PTHs, blind and buried vias

see IPC-6012E-EN, Tables 3-4ff: Minimum Requirements for Surface and hole copper Plating.

d. Spacing Copper to contour

Routed board edge: ≥ 0,23 mm
V-scored board edge: ≥ 0,45 mm (for board thickness 1,60 mm)

e. Teardrops

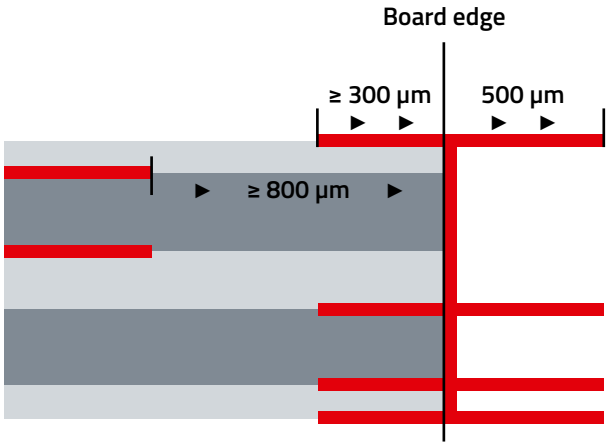
Usage of teardrps at any pad connection is recommended

7 EDGE PLATING

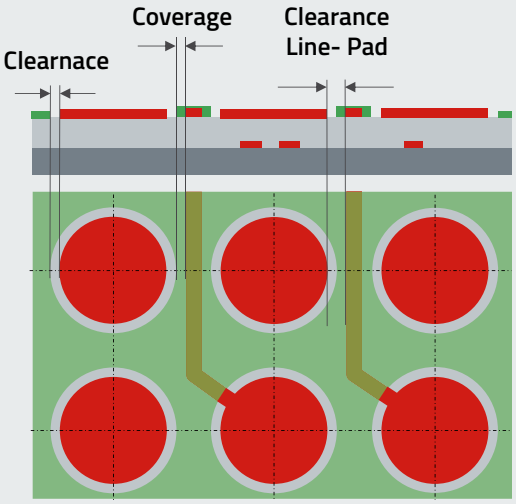
We offer edge metallization (sideplating) for the edges of your printed circuit boards. For a correct production we ask you to observe the design parameters:

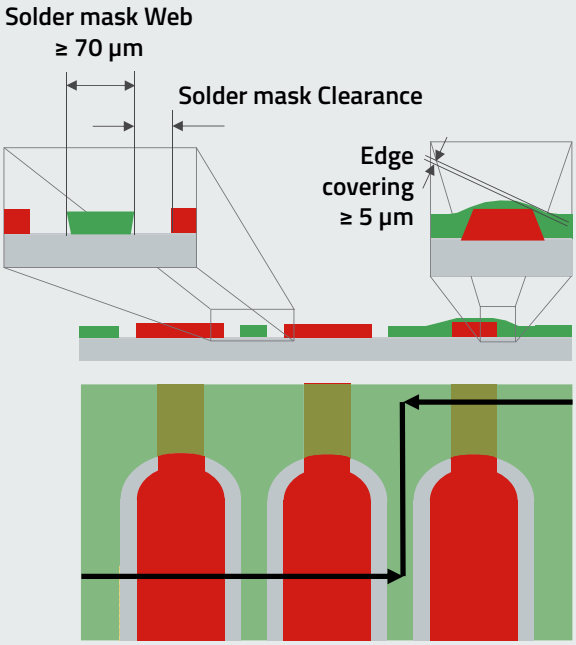
In your layout data, the PCB edge to be metallized must be designed with 500 µm protruding copper and at least 300 µm connection to the edge.

Layers that are not to be connected should have a clearance of min. 800 µm on the outer contour.



8 SOLDER MASK





Solder Mask		
	Standard	**Advanced**
Clearance	**≥ 50 µm**	35 µm
Coverage	**50 µm**	40 µm
Solder mask web	**≥ 70 µm**	–
Via-opening	final diameter +0,25 mm	

Manufacture without solder mask clearances involves additional effort and is not recommended due to quality reasons.

Our standard for soldermask is green photosensitive soldermask that meets the requirements of IPC-SM-840 Class T and H. Other colors such as white, black, blue, red or yellow can also be applied on request. Depending on the production location, the execution is as a colored solder mask instead of the green solder mask or as an additional print over the standard green solder mask. Please ask for these options, we will provide you with a quote.

9 PEELABLE MASKING

A peelable protective solder resist for protecting drill holes or assembly pads in wave soldering processes is applied by screen printing.
Coating thickness = 300 µm +/-200 µm (special tolerances possible after clarification). 1 mm circumferential distance to solder structures not to be covered (special tolerances possible after clarification).

10 MARKINGS

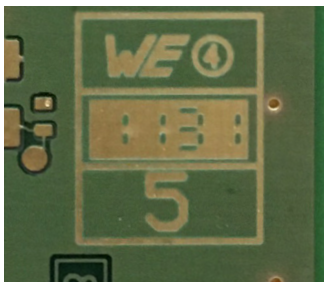
Markings (Legend Print)	Standard	Advanced
Silkscreen clearance to copper (Via pads, SMD pads, conductor)	300 µm	
Silkscreen clearance to the edge of the NPTH	300 µm	
Minimum line width and length of Print	150 µm	100 µm (white)
Clearance between markings	200 µm	

Each printed circuit board must be clearly identifiable with regard to the manufacturer.

- As a standard, your PCB receives a WE logo including date code (yy/ww) for tracking purposes.
- Data Matrix Code according to ISO/IEC 16022 with dynamic contents for individual marking is possible on request.
  Learn more here: www.we-online.com/DMCflyer
- If a UL marking is required, a suitable area must be provided in the data. As standard, the UL marking consists of manufacturer's identification plus UL type designation and factory identification. Optionally, the flammability class and the cURus logo can be included.
- Please note that our UL marking includes the WE logo as standard. This is not advertising. The logo is an official part of the UL marking.
- Alternatively, the Würth Elektronik UL file number "E76251" can be used instead of the WE logo, which requires more space.

Markings are possible in different ways. For all markings, the minimum permitted character heights defined below must be kept.

Examples:



Minimum character height as				
Copper (Size depends on the base copper)		Clearance in solder mask		Legend print Colour white
Base copper		Preferred on base material	Over copper, NOT HAL!	On soldermask
18 µm	**≥ 1,0 mm**	**≥ 1,0 mm**	**≥ 1,5 mm**	**≥ 1,5 mm**
35 µm	**≥ 1,5 mm**			
70 µm	**≥ 2,0 mm**			
105 µm	**≥ 2,5 mm**			

Würth Elektronik GmbH & Co. KG
Circuit Board Technology
Salzstr. 21
74676 Niedernhall · Germany
Phone +49 7940 946-0
BASIC@we-online.com
www.we-online.com

Version 2.0 / 04.2023
503633 EN