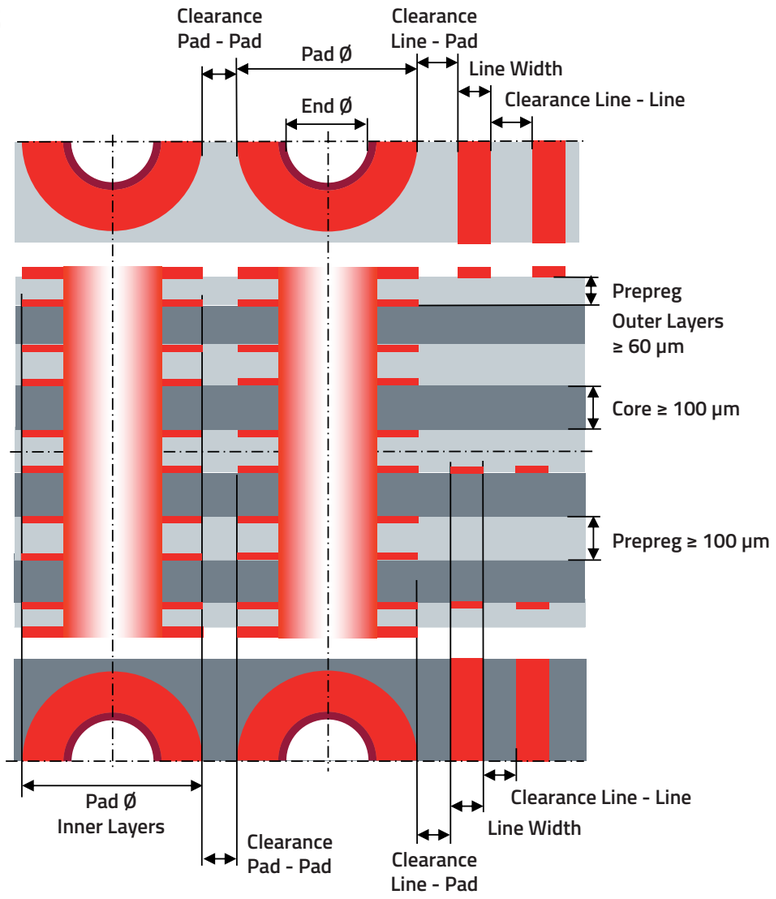
BASIC DESIGN RULES

Copper structures, spacings

OUTER LAYERS



INNER LAYERS

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.

OUTER LAYERS – COPPER SPACING

Starting foil thickness	Minimum copper thickness[1]		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
12 µm [3/8 oz.][2]	29,3 µm	34,3 µm	100 µm	80 µm	60 µm
17,1 µm [1/2 oz.]	33,4 µm	38,4 µm	120 µm	100 µm	60 µm
34,3 µm [1 oz.]	47,9 µm	52,9 µm	180 µm	160 µm	120 µm
68,6 µm [2 oz.]	78,7 µm	83,7 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

INNER LAYERS – COPPER SPACING

Starting foil thickness	Minimum copper thickness[3]		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
34,3 µm [1 oz.]	24,9 µm		120 µm	100 µm	60 µm
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

[3] IPC-6012E-EN Table 3-14: Internal Layer Foil Thickness after Processing

Drills, Drill Pads, Annular Rings, Clearances

PLATED THROUGH HOLES

Pad size	Remark	Aspect Ratio[4]	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

[4] "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.

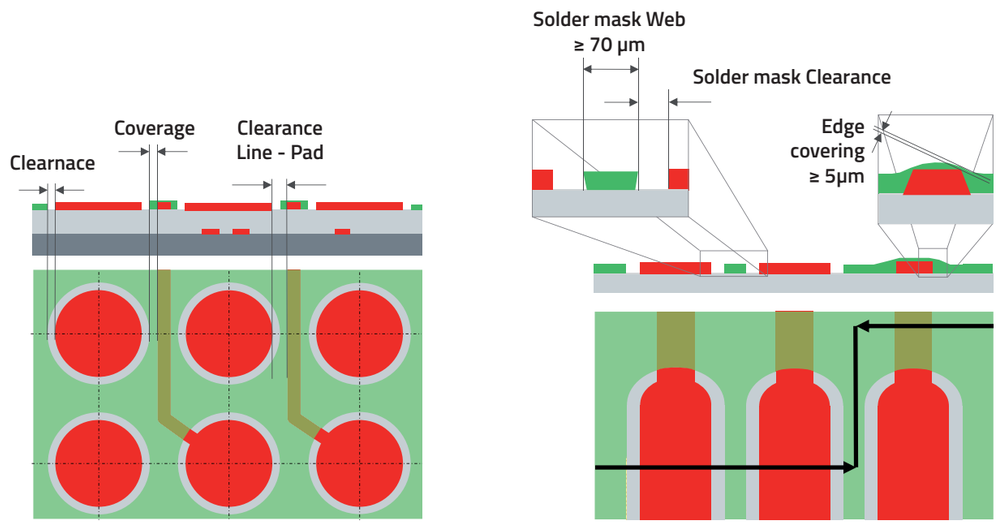
b. Spacing between holes
Minimum distances between holes (based on final diameter)

Hole-to-hole clearance same potential	300µm
Hole-to-hole clearance different potential	500µm
Distance NPTH-NPTH (Non Plated Through Hole)	350µm

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)

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.

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	

MORE SUPPORT THAN YOU EXPECT

Which base material qualities are used in a PCB can be seen from the stackup. Würth Elektronik offers cost- and production-**optimised standard stackups** on its website. Here you can also find all standards as digital stackup files for import into your EDA software.

www.we-online.com/basic-stackups

Do you have questions about fabrication data, tolerances, test documentation or packaging? In our **Technical Delivery Specification for printed circuit boards (TLS)** you will find our standards and recommendations for smooth and effective cooperation.

www.we-online.com/tds

Please contact us if you have higher requirements!
Finer parameters are possible in many cases by individual arrangement.

pcb@we-online.com

Version 2.2 / 12.2022 / 504809 EN