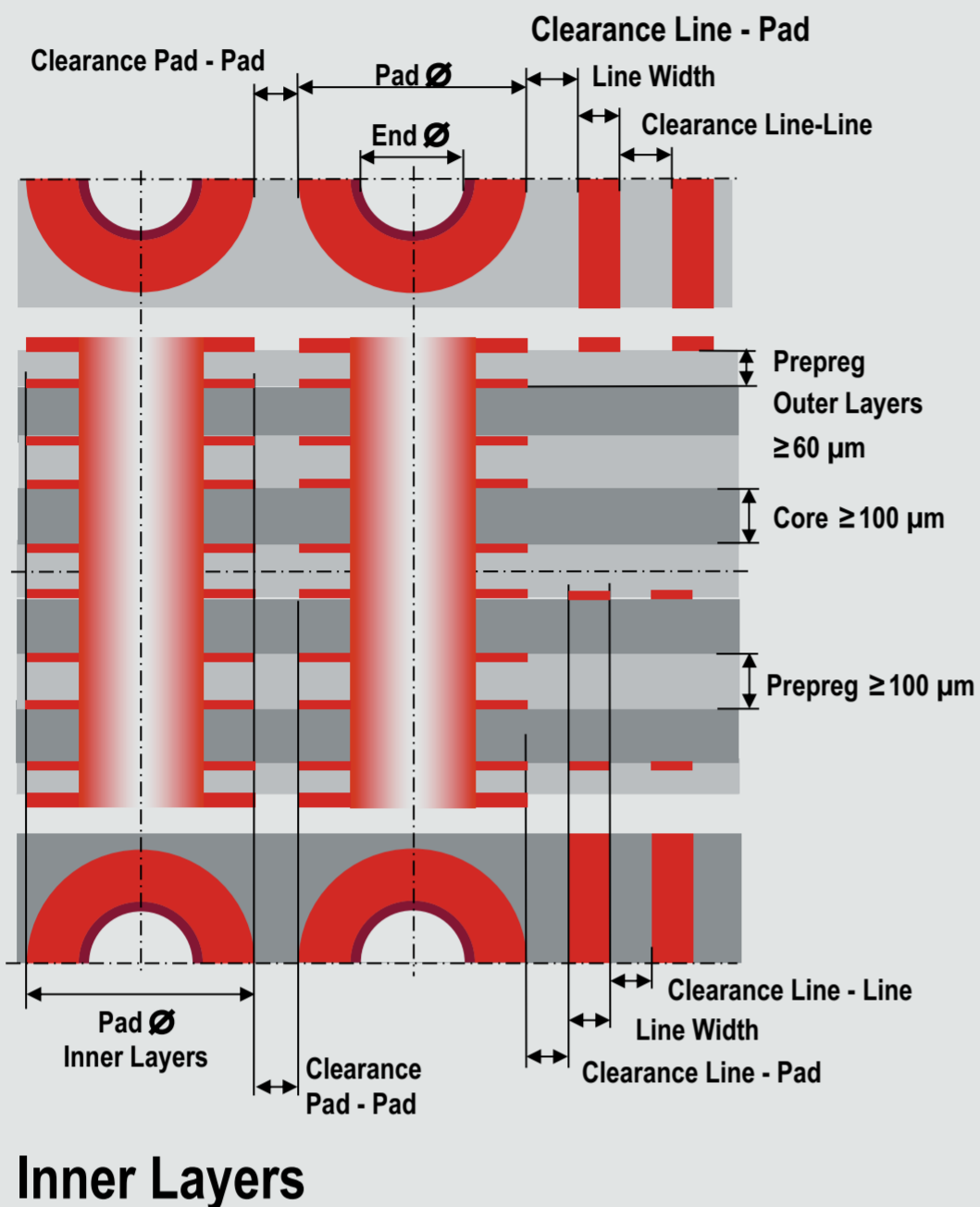


Basic Design Rules



Outer Layers



Outer layers – Track width and conductor spacing

Final copper thickness	Track width	Spacing
> 33,4 µm (IPC-6012)	100 µm	100 µm
70 µm	125 µm	160 µm
105 µm	150 µm	225 µm
ca. 25-30 µm ¹⁾	75 µm ¹⁾	75 µm ¹⁾

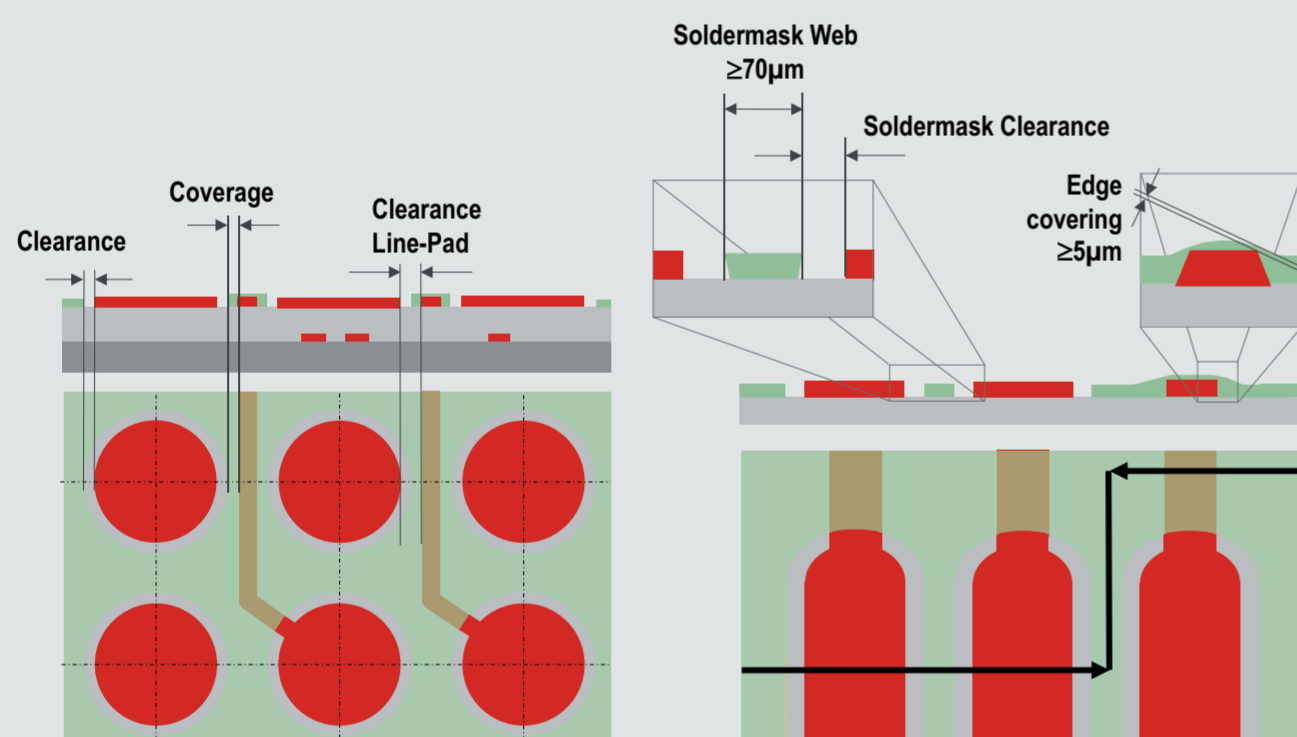
Inner layers – Track width and conductor spacing

Final copper thickness	Track width	Spacing
17,5 µm / ½ oz/ft ²	100 µm 75 µm ¹⁾	100 µm 75 µm ¹⁾
35 µm / 1 oz/ft ²	100 µm	100 µm
70 µm / 2 oz/ft ²	125 µm	150 µm
105 µm / 3 oz/ft ²	175 µm	225 µm

¹⁾ Technically possible. Due to cost reasons only advisable when absolutely necessary.

Plated through hole vias

Pad size	Note	Drill tool	Final hole diameter	Tolerance (Standard)	Copper clearance inner layer plane without pads	Solder mask opening Ø
0.60 mm	Preferred	0.35 mm	0.25 mm	+0.10/ -0.05 mm	≥ 0.80 mm	Tool-Ø + 0.15 mm
0.55 mm		0.30 mm	0.20 mm		≥ 0.75 mm	Tool-Ø + 0.15 mm
0.50 mm (Cu max. 35 µm)	Max. 12 layers Max. ca. 1.80 mm board thickness	0.25 mm	0.15 mm		≥ 0.70 mm	Tool-Ø + 0.15 mm
0.45 mm (Cu max. 35 µm)	For stack-ups with lower complexity	0.25 mm (0.20 mm)	0.15 mm		≥ 0.70 mm	Tool-Ø + 0.15 mm



Solder mask

	Standard	Advanced
Clearance	≥ 50 µm	35 µm
Coverage	50 µm	40 µm
Solder mask web	≥ 70 µm	—
Via-opening	See table „Plated through hole vias“	

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

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/downloads



Further design parameters

Clearance to copper

Copper clearance to routed board edge	≥ 0.23 mm
Copper clearance to scored board edge	≥ 0.45 mm for board thickness 1.60 mm
Copper clearance to NPT hole	≥ 0.3 mm circumferential

Markings are possible in different ways.

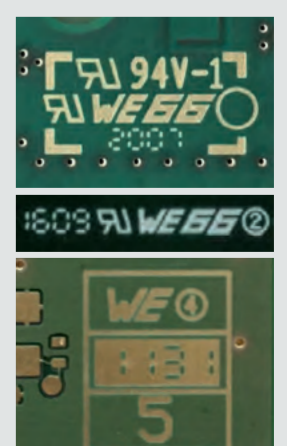
For all markings, the minimum permitted character heights defined below must be kept.

Markings

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 ²⁾	
18 µm	≥ 1.0 mm	≥ 1.5 mm	≥ 1.5 mm	
35 µm	≥ 1.5 mm			
70 µm	≥ 2.0 mm			
105 µm	≥ 2.5 mm			

²⁾ Line thickness 0.1 mm; Distance to soldermask opening 0.1 mm

Examples:



Please contact us if you have higher requirements!
Finer parameters are possible in many cases by individual arrangement.
cbt@we-online.de