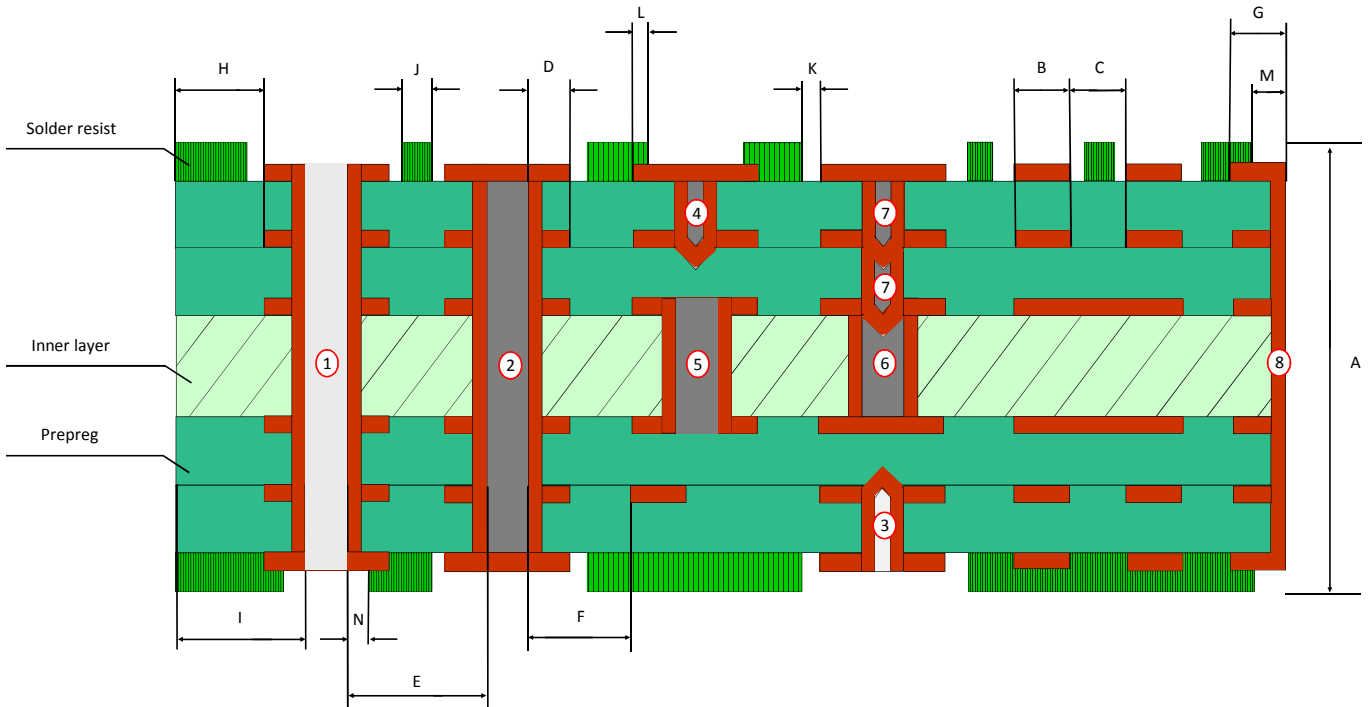


ILFA design rules for multilayer PCB's



ILFA PCBs	LEGEND	STANDARD	HIGH END (ON REQUEST)
General design rules			
Max. PCB dimensions		420x570mm	On request
Thickness multilayer	A	0.3 - 4.2mm	On request

Metallized holes & milled cut-outs (details refer to the diameter of the drilling tool)

Drilling tool diameter	Possible deviation with press-fit technology	LEGEND	STANDARD	HIGH END (ON REQUEST)
			Specified end diameter + 100 µm	On request
Through hole		1	Aspect ratio 1:8, minimum Ø 100 µm	Aspect ratio 1:10, minimum Ø 100 µm
Through hole, plugged and capped ¹		2	Aspect ratio 1:8, minimum Ø 150 µm	Aspect ratio 1:10, minimum Ø 100 µm
Blind via		3	Aspect ratio 1:1, minimum Ø 100 µm	Aspect ratio 1:1.2, minimum Ø 100 µm
Blind via, plugged and capped ¹		4	Aspect ratio 1:1, minimum Ø 150 µm	Aspect ratio 1:1.2, minimum Ø 150 µm
Buried via, plugged or resin filled ¹	Depending on layout and Ø	5	Aspect ratio 1:8, minimum Ø 150 µm	Aspect ratio 1:10, minimum Ø 100 µm
Buried via, plugged and capped ¹		6	Aspect ratio 1:8, minimum Ø 150 µm	Aspect ratio 1:10, minimum Ø 100 µm
Stacked via		7	Aspect ratio 1:1, minimum Ø 150 µm	Aspect ratio 1:1.2, minimum Ø 150 µm
Edge metallizations		8		

Conductive pattern / remnant annular rings

Trace width on inner & outer layers (µm)	Depending on copper thickness	LEGEND	STANDARD	HIGH END (ON REQUEST)
Conductor spacing on inner & outer layers (µm) <td>Depending on copper thickness</td> <td>B</td> <td>Without plugging ≥75 with plugging ≥100</td> <td>Without plugging ≥50 with plugging ≥75</td>	Depending on copper thickness	B	Without plugging ≥75 with plugging ≥100	Without plugging ≥50 with plugging ≥75
Annular ring between end-Ø inner and outer layers (µm)		C	Without plugging ≥75 with plugging ≥100	Without plugging ≥50 with plugging ≥75
Distance from hole to hole (µm)	Based on end-Ø	D	≥150	On request
Distance bore to adjacent conductive pattern (µm)	Based on end-Ø	E	≥300	On request
Overlap of edge metallization on outer layer (µm)	On inner layer recommended	F	≥250	On request
Distance of conductive pattern to milling contour (µm)		G	≥300	On request
Distance from hole to milling contour (µm)	Based on end-Ø	H	≥250	≥100
		I	≥400	On request

Soldermask

Soldermask fillet width (µm)	Depending on solder mask type, colour, copper thickness	LEGEND	STANDARD	HIGH END (ON REQUEST)
Soldermask clearance to copper (µm)		J	≥80	≥70
Soldermask overlap solder mask defined pads (µm)		K	≥50	≥25
Soldermask clearance of edge metallization (µm)		L	≥50	≥25
Soldermask clearance via / component bore unplugged (µm)		M	≥100	On request
		N	≥70	On request

Other options are possible. Your layer structure is not standard? We will be happy to help you.

¹Plugging is possible from a circuit board thickness of ≥0.3 mm excl. copper thickness. PCBs with external, flexible base materials, or materials without glass fabric cannot be plugged.