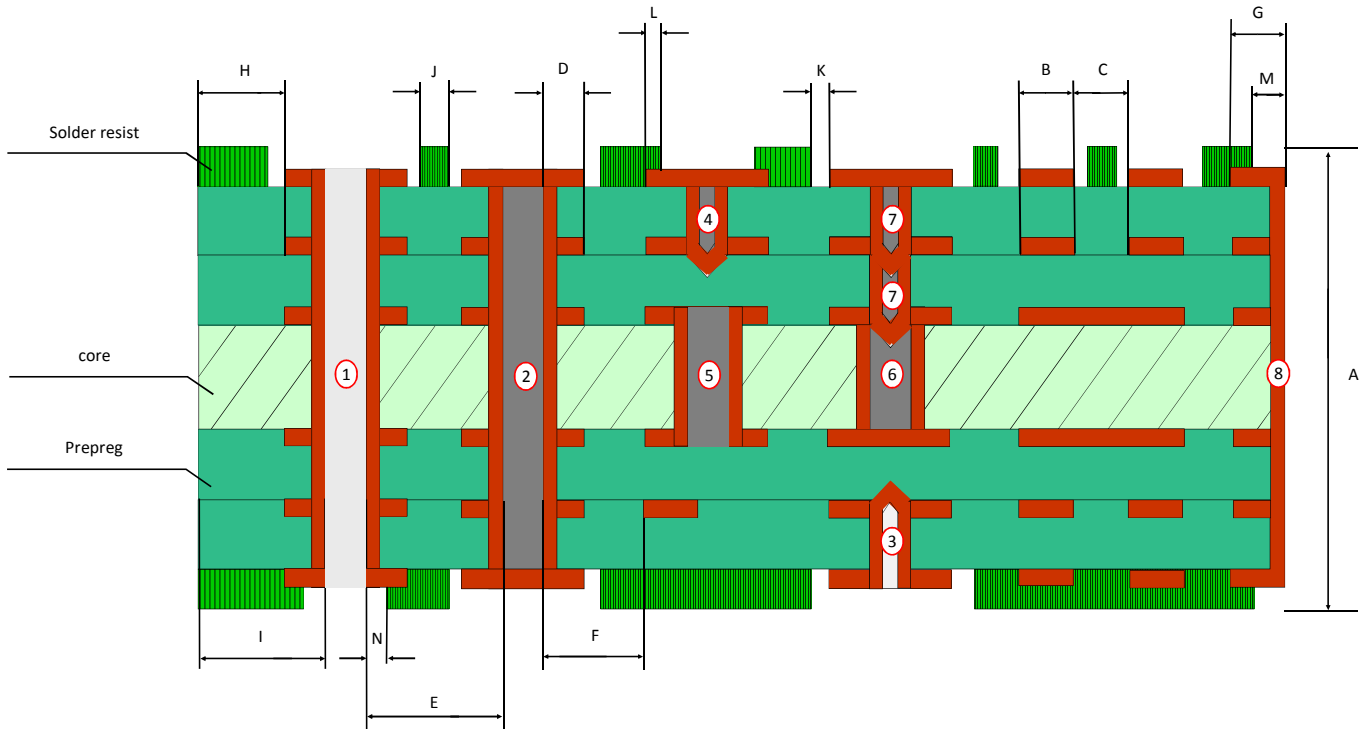


ILFA Design Rules for Multilayer



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 bores & milled holes (data refer to the bore tool diameter)

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

Ladder pattern / Rest rings

Conductor width on inner and outer layers (µm)	depending on copper thickness	LEGEND	STANDARD	HIGH END (ON REQUEST)
Conductor spacing on inner and outer layers (µm)	depending on copper thickness	C	Without plugging ≥ 75, with plugging ≥ 100	Without plugging ≥ 50, with plugging ≥ 75
Circumferential rest ring to end Ø inner and outer layer (µm)		D	≥ 150	on request
Hole to hole distance (µm)	related to end-Ø	E	≥ 300	on request
Distance hole to adjacent conductor pattern (µm)	related to end-Ø	F	≥ 250	on request
Overlapping edge metallization on outer layer (µm)	recommended on inner layer	G	≥ 300	on request
Distance ladder pattern to milling contour (µm)		H	≥ 250	≥ 100
Distance hole to milling contour (µm)	related to end-Ø	I	≥ 400	on request

Solder resist

Lacquer web width (µm)	depending on paint type, color, copper thickness	LEGEND	STANDARD	HIGH END (ON REQUEST)
Lacquer free to copper (µm)	thickness	J	≥ 80	≥ 70
Lacquer overlap, solder resist defined pads (µm)		K	≥ 50	≥ 25
Paint free edge metallization (µm)		L	≥ 50	≥ 25
Paint free via/part hole unplugged (µm)		M	≥ 100	on request
		N	≥ 70	on request

Other options are possible. Your layer structure does not meet the standard? We are happy to help.

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