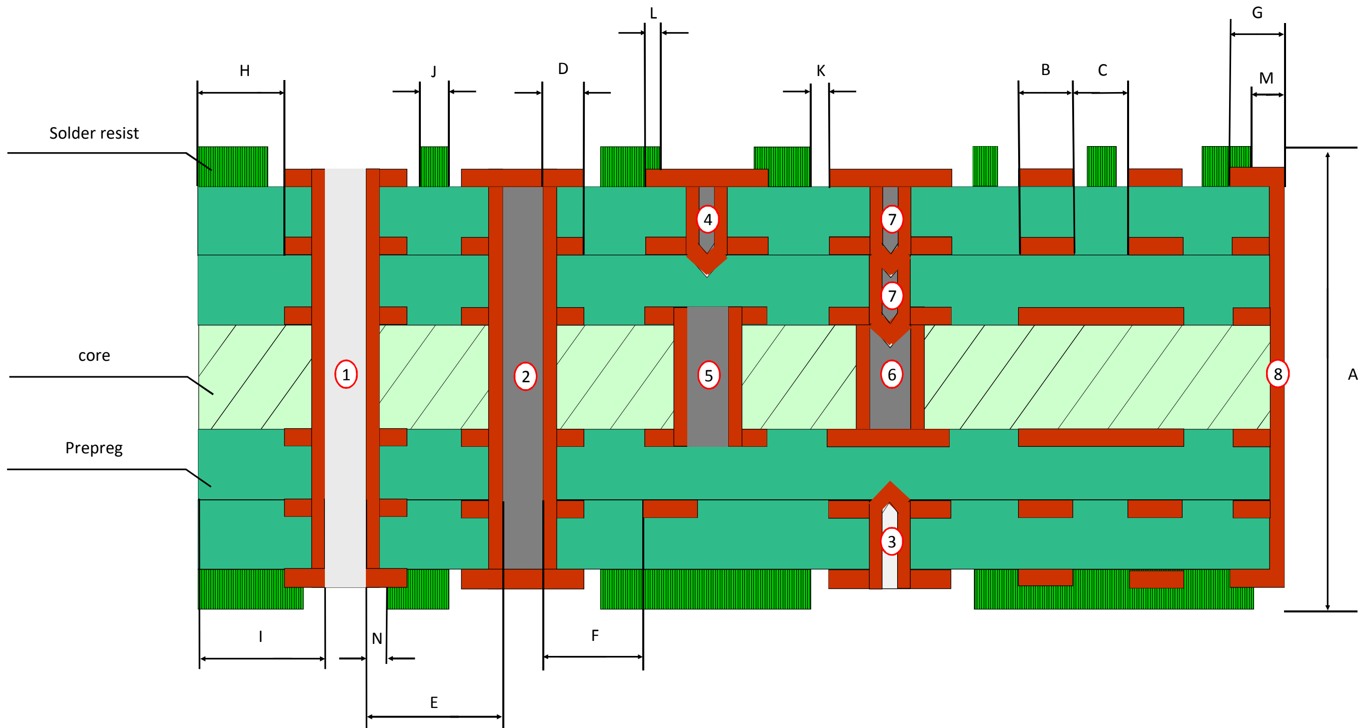
ILFA

WE CARE FOR CONNECTIONS

ILFA Design Rules for Multilayer



ILFA PCBs: **General design rules**		LEGEND	STANDARD	HIGH END (ON REQUEST)
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		Specified end diameter + 100 µm	on request
Through hole		1	Aspect Ratio 1:8, smallest ∅ 100 µm	Aspect Ratio 1:10, smallest ∅ 100 µm
Through hole plugged and capped[1]		2	Aspect Ratio 1:8, smallest ∅ 150 µm	Aspect Ratio 1:10, smallest ∅ 100 µm
Blind Via		3	Aspect Ratio 1:1, smallest ∅ 100 µm	Aspect Ratio 1:1.2, smallest ∅ 100 µm
Blind Via plugged and capped[1]		4	Aspect Ratio 1:1, smallest ∅ 150 µm	Aspect Ratio 1:1.2, smallest ∅ 150 µm
Buried Via plugged or resin filled[1]	depending on layout and ∅	5	Aspect Ratio 1:8, smallest ∅ 150 µm	Aspect Ratio 1:10, smallest ∅ 100 µm
Buried Via plugged and capped[1]		6	Aspect Ratio 1:8, smallest ∅ 150 µm	Aspect Ratio 1:10, smallest ∅ 100 µm
Stacked Via		7	Aspect Ratio 1:1, smallest ∅ 150 µm	Aspect Ratio 1:1.2, smallest ∅ 150 µm
Edge metallization		8		
Ladder pattern / Rest rings				
Conductor width on inner and outer layers (µm)	depending on copper thickness	B	Without Plugging ≥75, with Plugging ≥100	Without Plugging ≥50, with Plugging ≥75
LConductor 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-∅	E	≥300	on request
Distance hole to adjacent conductor pattern (µm)	related to end-∅	F	≥250	on request
Overlapping edge metallization on outer layer (µm)	recommended on inner layer	G	≥300	
Distance ladder pattern to milling contour (µm)		H	≥250	≥100
Distance hole to milling contour (µm)	related to end-∅	I	≥400	on request
Solder resist				
Lacquer web width (µm)	depending on paint type, color, copper thickness	J	≥80	≥70
Lacquer free to copper(µm)		K	≥50	≥25
Lacquer overlap. solder resist defined pads (µm)		L	≥50	≥25
Paint free edge metallization (µm)		M	≥100	on request
Paint free via/part hole unplugged (µm)		N	≥70	on request

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

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

VERSION: ILFA DM 3