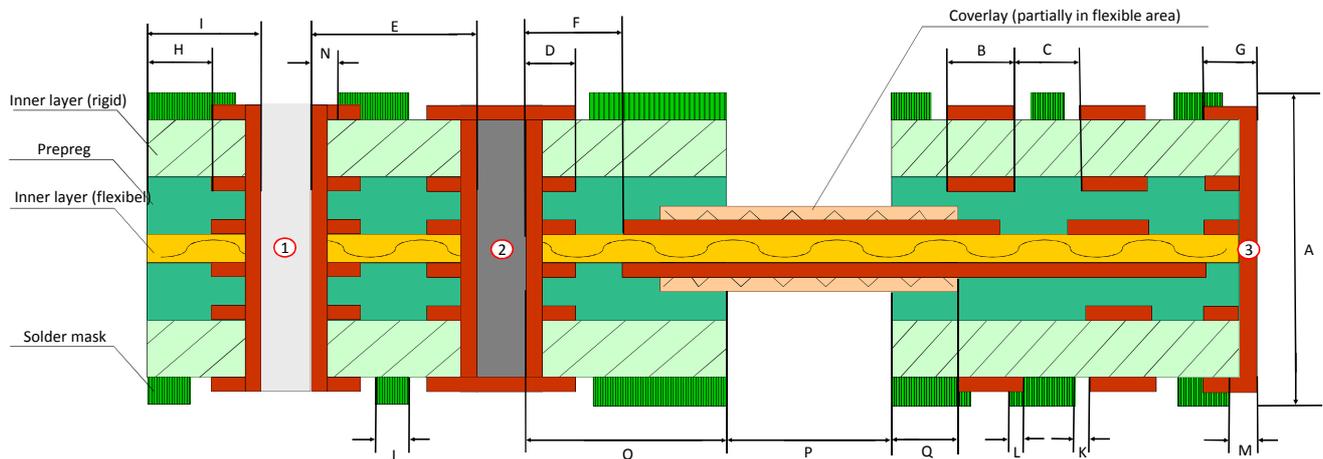


ILFA design rules for rigid-flex PCBs



ILFA PCBs		LEGEND	STANDARD	HIGH END (ON REQUEST)
General design rules				
Max. PCB dimensions			420x570mm	On request
Thickness rigid-flex		A	0.4 - 4.2mm	On request
Metallized holes & milled cut-outs (details refer to the diameter of the drilling tool)				
Drilling tool diameter	Deviations possible with press-fit technology		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
Edge metallizations		3	min. 2.0 mm distance from the flexible area	
Conductive pattern / remnant annular rings				
Trace width on inner & outer layers (µm)	Depending on copper thickness	B	Without plugging ≥75 with plugging ≥100	Without plugging ≥50 with plugging ≥75
Conductor spacing on inner & outer layers (µm)	Depending on copper thickness	C	Without plugging ≥75 with plugging ≥100	Without plugging ≥50 with plugging ≥75
Annular ring between end-Ø inner and outer layers (µm)		D	≥150	On request
Distance from hole to hole (µm)	Based on end-Ø	E	≥300	On request
Distance bore to adjacent conductive pattern (µm)	Based on end-Ø	F	≥250	On request
Overlap of edge metallization on outer layer (µm)	On inner layer recommended	G	≥300	
Distance of conductive pattern to milling contour (µm)		H	≥250	≥100
Distance from hole to milling contour (µm)	Based on end-Ø	I	≥400	On request
Soldermask				
Soldermask fillet width (µm)	Depending on solder mask type, colour, copper thickness	J	≥80	≥70
Soldermask clearance to copper (µm)		K	≥50	≥25
Soldermask overlap solder mask defined pads (µm)		L	≥50	≥25
Soldermask clearance of edge metallization (µm)		M	≥100	On request
Soldermask clearance via / component bore unplugged (µm)		N	≥70	On request
Special rigid-flex design rules				
Distance from bore to flex area (µm)	Based on end-Ø	O	≥700	
Length of flex area (µm)		P	≥2000	
Overlap of coverlay with rigid area (µm)		Q	500	500 - 1000
Minimum bending radius ² single bend (mm)	Without back bend		Thickness of flexible area X 1	On request
Minimum bending radius ² 4-12 cycles (mm)			Thickness of flexible area X 6	On request
Minimum bending radius ² dyn. stress (mm)			Thickness of flexible area X ≥10	On request

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.

²Bending radius: Thickness of the flexible area = addition of all materials (coverlay, adhesive, copper, base material). The information is only valid for a flexible core with maximum two copper layers.