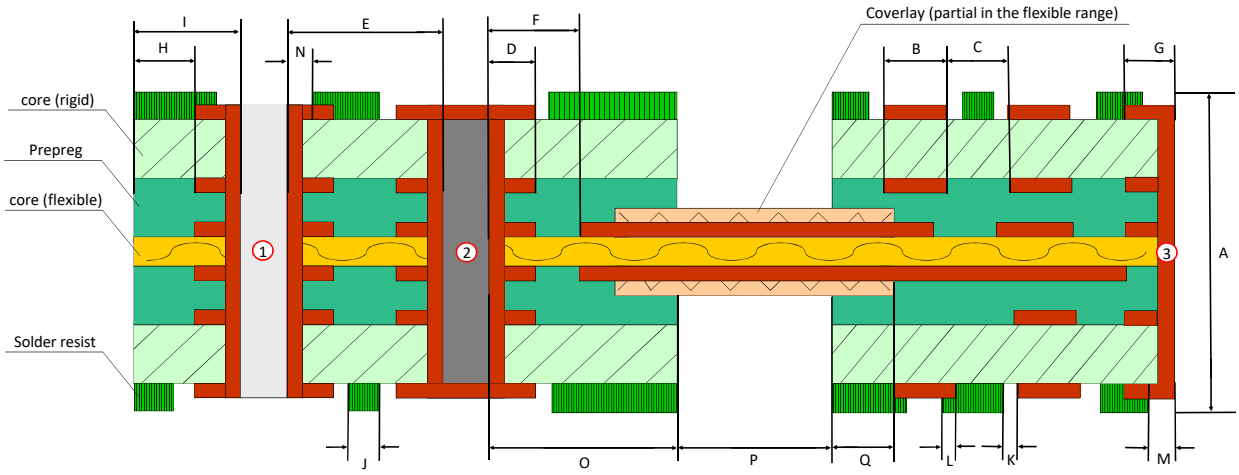
ILFA Design Rules for Rigid-Flex Printed Boards



ILFA PCBs: General design rules		LEGEND	STANDARD	HIGH END (ON REQUEST)
Max. PCB dimensions			420x570mm	on request
Thickness rigid flex		A	0.4 - 4.2mm	on request
Metallized bores & milled holes (data refer to the bore tool diameter)				
Drilling tool diameter	Possible deviation for press-fit		Specified final diameter+ 100 µm	on request
Through hole	technology	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
Edge metallization		3	min. 2.0 mm distance to the flexible area	
Ladder pattern / Rest rings				
Conductor width on inner and outer layers (µm)	depending on copper strength	B	Without Plugging ≥75, with Plugging ≥100	Without Plugging ≥50, with Plugging ≥75
Conductor spacing on inner and outer layers (µm)	depending on copper strength	C	Without Plugging ≥75, with Plugging ≥100	Without Plugging ≥50, with Plugging ≥75
Circum. rest ring to end inner & 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
Overlap. edge metallization on outer layer (µm)	recommended on inner layer	G	≥300	
Distance ladder pattern to milling contour (µm)		H	≥300	≥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
Special rigid flex design rules				
Distance hole to flex area (µm)	related to end-∅	O	≥700	
Length flex area (µm)		P	≥2000	
Overlap of the cover layer with rigid area (µm)		Q	500	500 - 1000
Minimum bending radius[2] one time bend (mm)	without backbend		Thickness of the flexible area X 1	on request
Minimum bending radius[2] 4-12 cycles (mm)			Thickness of the flexible area X 6	on request
Minimum bending radius[2] dynamic stress (mm)			Thickness of the flexible area X ≥10	on request

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

[2]Bending radius: thickness of the flexible area = Addition of all materials (cover layer, adhesive, copper, base material). The data applies only to a flexible core with max. two copper layers.