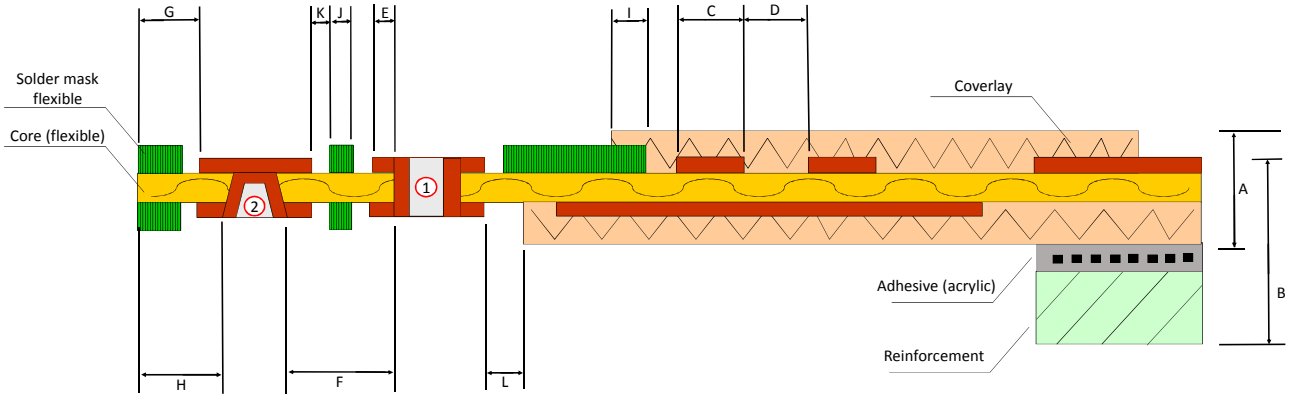
ILFA Design rules for single- and double-sided flexible PCB's



ILFA PCB's: General design rules		LEGEND	STANDARD	HIGH END (ON REQUEST)
Max. PCB dimensions			420x570mm	On request
Thickness of flexible area over all		A	≥0.065 mm (single sided) , ≥0.10 mm (double sided)	On request
Thickness of reinforced connector area		B	≥0.10 mm - 0.50 mm , Tolerance +/- 30 µm ≥ 0.50 mm , Tolerance +/- 10%	
Metallized holes & milled cut-outs (details refer to the diameter of the drilling tool)				
Drilling tool diameter			Specified end diameter + 100 µm	On request
Through hole		1	Aspect Ratio 1:8, minimum ∅ 75 µm	Aspect Ratio 1:10, minimum ∅ 50 µm
Blind Via (from top or bottom)		2	Aspect Ratio 1:1, minimum ∅ 75 µm	Aspect Ratio 1:1,2, minimum ∅ 50 µm
Conductive pattern / remnant annular rings				
Trace width min. (µm)	Depending on copper thickness	C	≥75	≥50
Conductor spacing min. (µm)	Depending on copper thickness	D	≥75	≥50
Annular ring to drilling tool-Ø (µm)		E	≥100	≥75
Distance from hole to hole (µm)[1]	Based on drilling tool-Ø	F	≥100	
Distance of conductive pattern to milling contour (µm)		G	≥300	≥100
Distance of metallized hole to milling contour (µm)	Based on drilling tool-Ø	H	≥350	≥200
Overlap of coverlay with solder mask (µm)	Only in a combined type	I	300	On request
Solder mask / coverlay				
Solder mask fillet width (µm)	With solder mask thickness ≤ 50 µm	J	≥100	On request
Solder mask clearance to copper (µm)		K	≥50	≥25
Coverlay clearance to copper (µm)		L	≥150	≥100
Material thicknesses (µm)				
Thickness of flexible polyimide (adhesiveless)	Preferably with rolled copper		Thickness: 25 - 150, Copper: 12 - 35	Copper: 9 ED-Copper, or Copper ≥70
Thickness of coverlay	DuPont FR or LF (preferably FR)		25, 38, 50, 75, 100, 150	
Thickness of acrylic adhesive for reinforcement (thermosetting)	DuPont FR or LF (preferably FR)		25, 50, 75	
Thickness of acrylic adhesive for reinforcement (transfer adhesive)	3M transfer adhesive		50 or 130	
Thickness of reinforcement	FR4 or Coverlay		Coverlay 25 - 150, FR4 50 - 3200	
Bending				
Minimum bending radius[2] single bend (mm)	Coverlay		Thickness of flexible area X 1	On request
Minimum bending radius[2] 4-12 cycles (mm)	Coverlay		Thickness of flexible area X 6	On request
Minimum bending radius[2] dyn. stress (mm)	Coverlay		Thickness of flexible area X ≥ 10	On request
Minimum bending radius[2] 4-12 cycles (mm)	Flexible solder mask		≥ 1,50 mm	On request

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

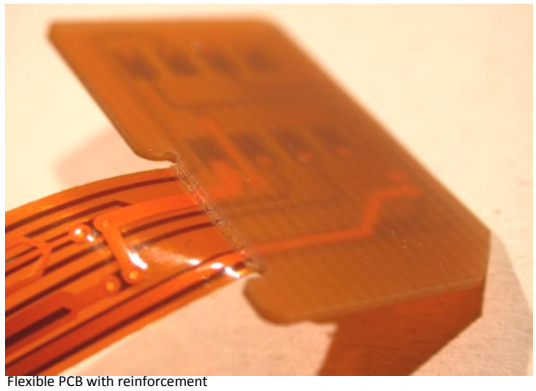
[1]Distance from hole to hole: the measure refers to the distance between holes with equal electric potential. The minimum for holes with different potentials results from the minimum annular ring and the minimum conductor spacing.
[2]Bending radius: thickness of flexible area = addition of all materials (coverlay, adhesive, copper, base material). The information is only valid for a flexible core with maximum two copper layers.

ILFA Design rules for single- and double-sided flexible PCB's

Examples of flexible PCB's

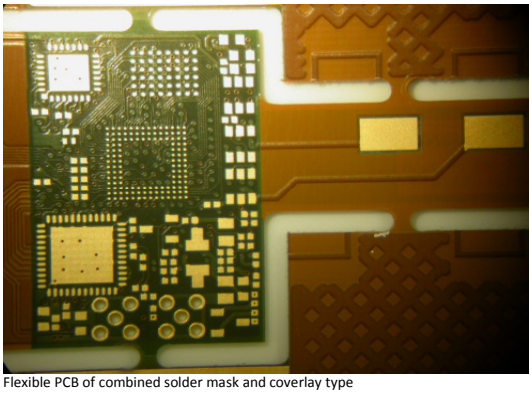
Reinforcements

In order to make mounting easier or to make a reliable plug connection possible, flexible PCB's are often reinforced.
The flexible PCB's can be partially furnished with a reinforcement. Depending on quantity and layout, 2 different adhesion types can be used. The cold type using transfer adhesive, where the reinforcements are applied manually by means of an adapter, has proven itself for minor quantities and small reinforced areas. For large-area reinforcements or high quantities an application by hot lamination is used.


Flexible PCB with reinforcement

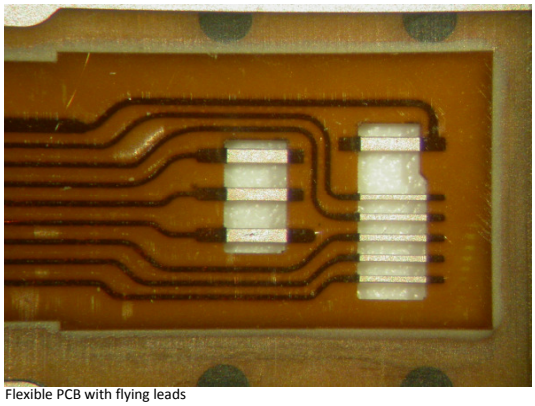
Flexible solder mask and/or coverlay

For isolation of outer conductors a flexible solder mask and/or coverlay can be used. The flexible solder mask is the low-cost option, being less flexible than coverlay, but due to photolithographic structuring it has the advantage that a small clearance to structures laid open is sufficient. This way solder mask fillets can also be accomplished between fine pitch components.
When high flexibility and dynamic bending stress are a matter, coverlay is the first choice. It is pre-structured with a laser and applied in a lamination process. Since tolerances greater than those with a solder mask arise in this process, the combined type is often used. Photostructurable solder mask in the component area and a highly flexible bond of polyimide and acrylic adhesive, commonly called coverlay. An overlap area must be arranged (see design rules pt. I).


Flexible PCB of combined solder mask and coverlay type

Flying leads (also floating leads)

A special technology are the so-called flying leads. The conductor copper is cleared from base material in partial areas from both sides. This is achieved by laser ablation. The flexible PCB can then be soldered on top of a rigid PCB or a component.


Flexible PCB with flying leads