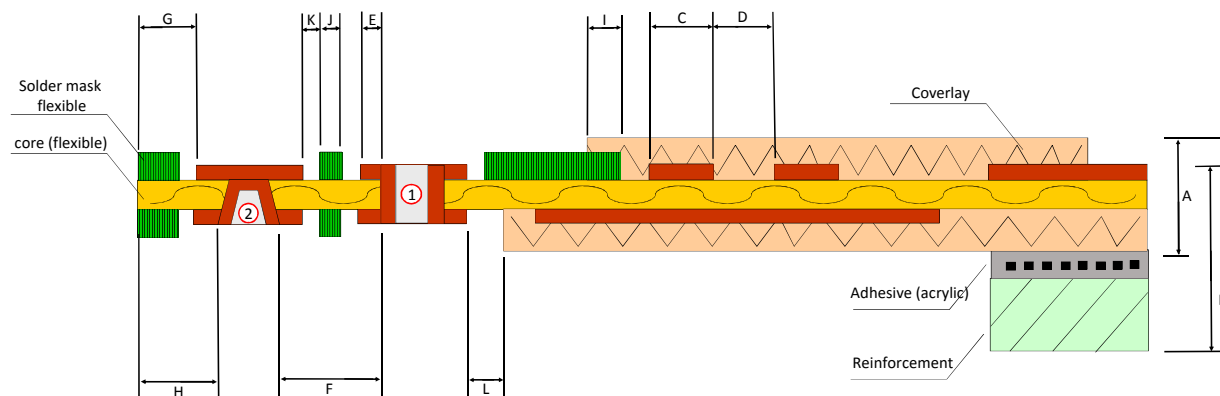


ILFA Design rules for single and double-sided flexible printed circuit boards



ILFA PCBs:

General design rules

Max. PCB dimensions
Thickness flexible area over all
Thickness reinforced area connector

LEGEND	STANDARD	HIGH END (ON REQUEST)
A	420x570mm ≥0.065 mm (single) , ≥0.10 mm (double-sided)	on request
B	≥0.10 mm - 0.50 mm , Tolerance +/- 30 μm ≥ 0.50 mm , Tolerance +/- 10%	on request

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

Drilling tool diameter
Through hole
Blind Via (from above, or below)

	specified end diameter + 100 μm	on request
1	Aspect ratio 1:8, smallest Ø 75 μm	Aspect ratio 1:10, smallest Ø 50 μm
2	Aspect ratio 1:1, smallest Ø 75 μm	Aspect ratio 1:1,2, smallest Ø 50 μm

Ladder pattern / Rest rings

Conductor width min. (μm) depending on copper thickness
Conductor spacing min. (μm) depending on copper thickness
Circumferential rest ring to drilling tool-Ø (μm)
Hole to hole distance (μm)¹ related to drilling tool-Ø
Distance ladder pattern to milling contour (μm)
Distance hole to milling contour (μm) related to drilling tool-Ø
Overlap Coverlay and Varnish (μm) Only for combination variant

C	≥75	≥50
D	≥75	≥50
E	≥100	≥75
F	≥100	
G	≥300	≥100
H	≥350	≥200
I	300	on request

Solder resist / Coverlay

Lacquer web width (μm) For coating thickness ≤ 50 μm
Lacquer free to copper (μm)
Coverlay clearance to copper (μm)

J	≥100	on request
K	≥50	≥25
L	≥150	≥100

Material thicknesses (μm)

Thickness Flexible Polyimide (adhesive-free) Preferred with rolled copper
Thickness Coverlay DuPont FR o. LF (preferred FR)
Thickness acrylic adhesive for Reinforcement DuPont FR o. LF (preferred FR)
Thickness acrylic adhesive for Reinf.(Transfer adhesive) 3M Transfer adhesive
Thickness Reinforcement FR4 or Coverlay

Thickness: 25 - 150, copper: 12 - 35	copper: 9 ED-copper, or copper ≥70
25, 38, 50, 75, 100, 150	
25, 50, 75	
50 o. 130	
Coverlay 25 - 150, FR4 50 - 3200	

Bending

Minimum bending radius² one time bend (mm) Coverlay
Minimum bending radius² 4-12 cycles (mm) Coverlay
Minimum bending radius² dyn. stress (mm) Coverlay
Minimum bending radius 4-12 cycles (mm) flexible solder resist

Thickness of flexible area X 1	on request
Thickness of flexible area X 6	on request
Thickness of flexible area X ≥ 10	on request
≥ 1,50 mm	on request

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

¹Hole-to-hole distance: The dimension refers to the distance between holes with the same electrical potential. The minimum dimension for boreholes with different potentials results from the minimum circumferential residual ring and the minimum conductor spacing.

²Bending radius: thickness of the flexible area = addition of all materials (Coverlay, adhesive, copper, basic material). The specifications apply only to a flexible core with max. two copper layers.

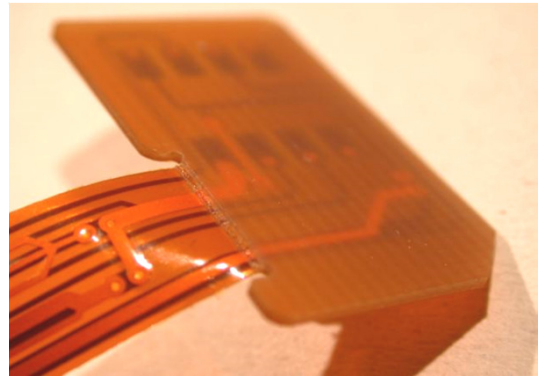
ILFA Design rules for single and double-sided flexible printed circuit boards

Examples of flexible PCBs

Reinforcements

In order to simplify assembly or to enable a reliable plug-in connection, flexible printed circuit boards are often reinforced.

The flexible printed circuit boards can be partially reinforced. Depending on the quantity and layout, two adhesive variants can be used. The cold variant using transfer adhesive, in which the reinforcements are applied manually with the aid of an adapter, has proved successful for small quantities and small reinforcement areas. For large-area reinforcements or high quantities, application is by hot lamination.

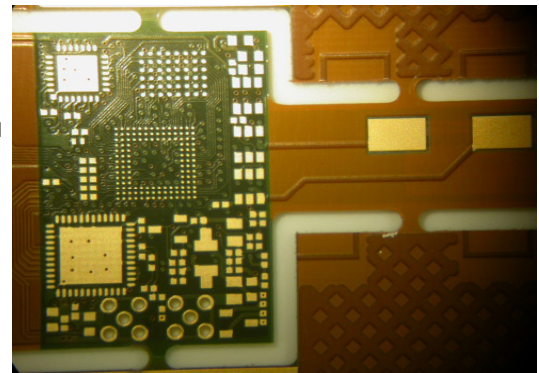


Flexible printed circuit board with reinforcement

Flexible solder resist and/or overlay

Flexible solder resist and/or Coverlay can be used to insulate the outer conductors. The flexible resist is the low-cost variant, is less flexible than Coverlay, but offers the advantage of photolithographic structuring that a small amount of clearance around exposed structures is sufficient. In this way, resist ridges can also be realized between FinePitch components.

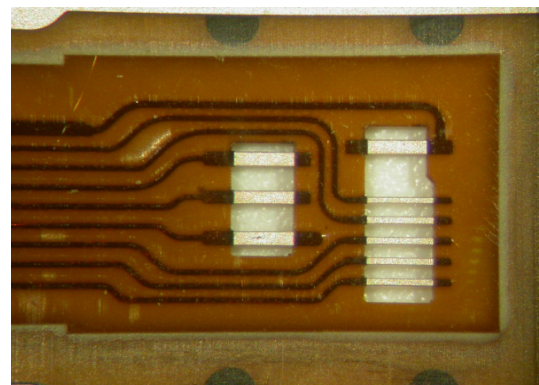
When high flexibility and dynamic bending stresses are required, Coverlay is the first choice. This is pre-structured by laser and applied in a lamination step. Since this process involves greater tolerances than varnish, the combination variant is often used. In the assembly area, the photo-structurable coating, and in the bending area, the highly flexible composite of polyimide and acrylate adhesive, the so-called Coverlay. An overlap area must be provided.



Flexible printed circuit board with combination variant lacquer and coverlay

Flying Leads (also known as Floating Leads)

Flying leads are a special technology. The copper of the conductors is exposed from base material on both sides in partial areas. This is achieved by laser ablation. The flexible printed circuit board can then be soldered from above onto a rigid printed circuit board or a component.



Flexible printed circuit board with flying leads