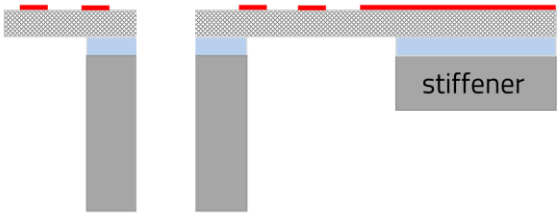
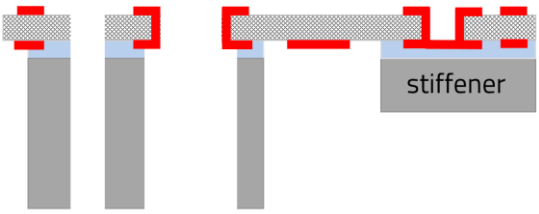
Design Rules PURE.flex
xF and xF-Ri

These design rules apply to:
Flexible circuit boards with 1 or 2 copper layers on flexible polyimide material, optionally with glued mechanical stiffener. Please use our SLIM.flex-Technology for up to 8 copper layers.
Application in accordance with IPC 2223 Use A: Flex-to-install, no UL marking.

Examples (shown without coverlay or solder mask):



1F-Ri: 1 layer with glued stiffener

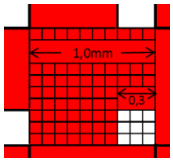


2F-Ri: 2 layers plated, with stiffener

Nomenclature: F = Flex, Ri = stiffener out of FR4

Basic information

- Please comply with general standards, such as IPC or IEC.
- Please take note of the useful information and tips in our RIGID.flex Design Guide at www.we-online.com/flex.
- Please see the WE BASIC Design Guide for rules for line widths, spacing, via and pad sizes, solder mask at www.we-online.com/basic.
- Essentially, marking print is not possible.
- Flexible circuit boards must be dried before they are assembled. Further information about this is available at www.we-online.com/starrflex.
- Copper removal is required in ground or reference layers for drying.
- Recommendation: Copper openings 0.3 mm per 1 mm length of copper.



- Flex-to-install bending radius: Installation Use A in accordance with IPC-2223 up to 90° bending angle:
  - 1 or 2 copper layers: 10 x total thickness (IPC-2223 section 5.2.3.3)
  - For use in more demanding conditions, please contact us.
- We will be happy to create the optimal delivery panel for you (best price!).

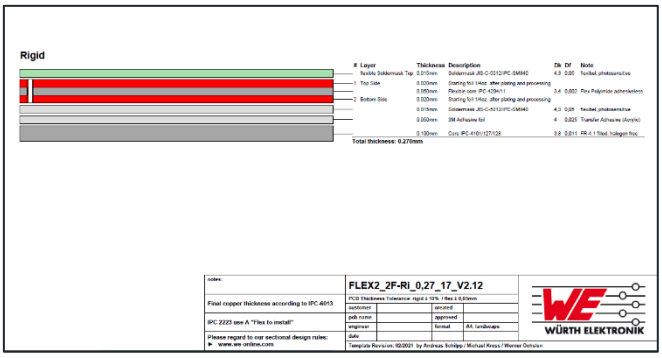
Design Rules PURE.flex
xF and xF-Ri

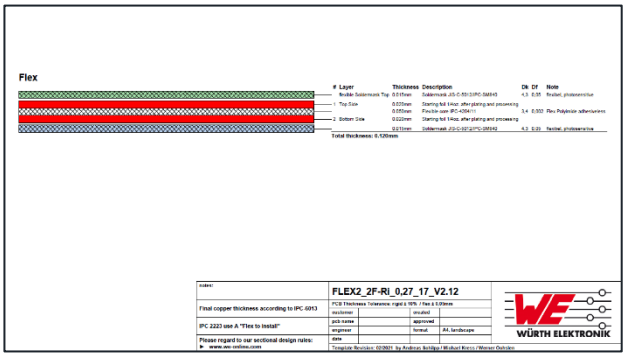
Material specifications

Material	Standard	Spec. sheet	Description	Application
Flexible base material	IPC-4204	11	Polyimide adhesiveless	Microvia, hand soldering
	IPC-4204	2	Polyimid with glue	
Rigid material	IPC4101	21	FR4 Tg135°C	Standard for stiffeners
Flex solder mask	JIS C 5012/ IPC-SM840		green, photosensitive	Standard
Coverlay	IPC-4203	1 / 2	Polyimide covering film 25µm, acrylic or epoxy glue (multilayer process)	Optional in place of flex solder mask (surcharge)

Standard Stackups

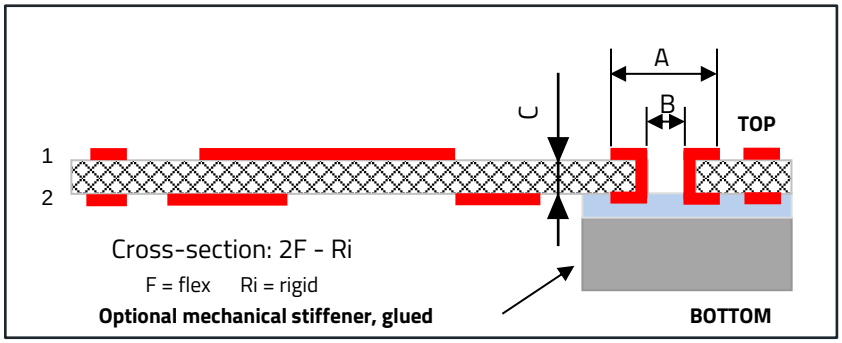
The standard stackups you will find under www.we-online.com/pureflex.

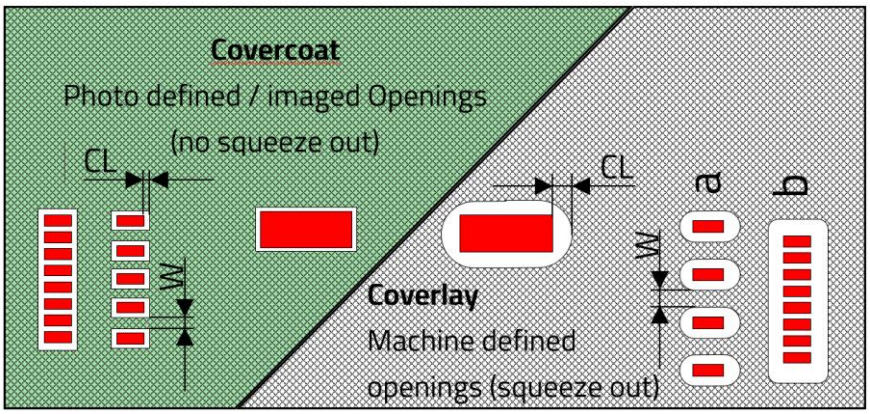




Standard design

1. Polyimide 50 µm adhesiveless, flexible PCB total thickness without stiffener 0.1 mm to 0.12 mm (depending on number of layers)
2. Base Copper thickness inner layers 18 µm, exterior layers 9 µm + electroplating
3. Flexible photosensitive solder resist green
4. Standard vias, plating thickness in accordance with IPC-6013
5. Outline lasered or milled, smallest milling diameter 1.6 mm. V-scoring not permitted!
6. Solderable surface ENIG (electroless Nickel – immersion Gold)
7. Packaged in ESD shrink wrap





a: single Coverlay opening larger than pad **b:** window opening

Symbol	Description	Technical Standard	Advanced requirements	SLIM.flex
-	Distance copper to outline	≥ 300 µm	≥ 100 µm	
-	Number x of copper layers (xF)	1 to 2		3 to 8
C	Thickness of flexible material (Polyimide)	50 µm	75/100/ (125) µm	5 µm
-	Thickness of cold-bonded stiffener made of FR4 material	0.1 – 0.5 mm	0.5 – 0.8 mm	
-	Thickness of glue for stiffener	50 µm		
CL (Solder mask)	Minimum clearance of copper pad with photosensitive flex solder mask	50 µm circumferential		
CL (Coverlay)	Minimum clearance of copper pad with coverlay (milled, lasered)	450 µm circumferential		
W (Solder Mask)	Minimum bridge width photosensitive flex solder mask	70 µm circumferential		
W (Coverlay)	Minimum bridge width coverlay (milled, lasered)	500 µm		
Avoid vias in bending areas!				
„ZIF"	**ZIF contacts thickness tolerance**	± 0.05 mm		
Usage of microvia technology: possible with 2 or more layers:				
A (HDI)	Minimum pad diameter for microvia	300 µm	250 µm	200 µm
B (HDI)	Finished hole diameter lasered microvia	≈ 100 µm		70 µm

Further specifications available on request, please contact us: flex@we-online.com