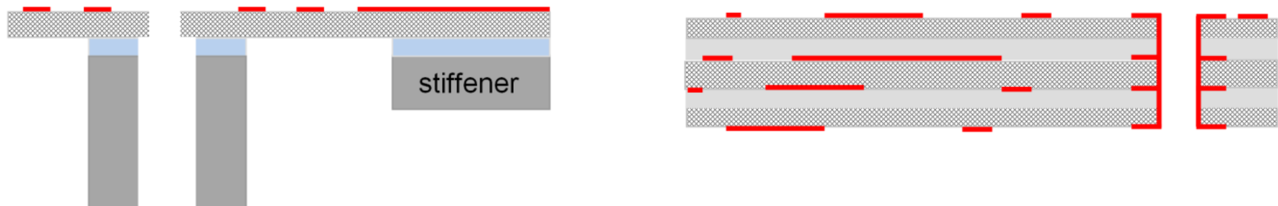
Design rules
Flex xF and TWINflex® xF – Ri

Application in accordance with IPC 2223 Use A: Flex-to-install
No UL marking

These design rules apply to:

flexible circuit boards with 1 to 4 copper layers on flexible polyimide material, optionally with glued mechanical stiffener

Examples (shown without coverlay or solder mask):



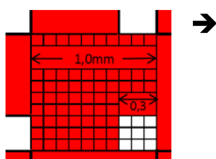
1F-Ri: 1 layer with glued stiffener

4F: Flexible multilayer with 4 layers, standard vias

Nomenclature: F = flexible, Ri = rigid

Basic information

- Please comply with general standards, such as IPC or IEC
- Please take note of the useful information and tips in the WE Flex-Rigid Design Guide *
- Please see the WE Basic Design Guide for rules for line widths, spacing, via and pad sizes, solder resist mask*
- Essentially, marking print is not possible.
- Flexible circuit boards must be dried before they are assembled. Further information about this is available in our Internet pages. *
- 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)
  - More than 2 copper layers: 20 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!)

* All documentation can be found online at: www.we-online.com/flex

Design rules
Flex xF and TWINflex® xF – Ri

Application in accordance with IPC 2223 Use A: Flex-to-install
No UL marking

Material specifications

Material	Standard	Spec. sheet	Description	Application
Flexible base material	IPC-4204	11	Polyimide adhesiveless	Microvia, hand soldering
	IPC-4204	2	Polyimide with glue	
Rigid material	IPC4101	21	FR4 Tg 135°C	Standard
Flex solder mask	JIS C 5012/ IPC-SM840		green, photosensitive	Standard
Coverlay	IPC-4203	1 / 2	Polyimide covering film 25 µm, acrylic or epoxy glue (multilayer process)	Optional in place of flex solder mask (surcharge)

Layer stack-up

Standard layer stack-ups, see www.we-online.com/flex

customer	
pcb name	
WE-number	
engineer	
date	

WÜRTH ELEKTRONIK

Twinflex® 2F-Ri

PCB Thickness : 0,29 mm +/-0,05mm Flex Thickness: 0,15 mm +/- 0,05mm

Rigid area Structure	Flex area Thickness	Rigid area Thickness	Material description	Flex area Structure	Viatypes	Layer usage	Impedance Er	Impedance Z[Ohm] / Line / Space
	15		photosensible flexible soldermask					
L1	35	35		Top-Layer				
	50	50	polyimid					
L2	35	35		Bottom-Layer				
	15	15	photosensible flexible soldermask					
adhesive foil		50	adhesive foil					
FR4 stiffener material		100	FR4					

Notes:

IPC 2223 use A "Flex to install"

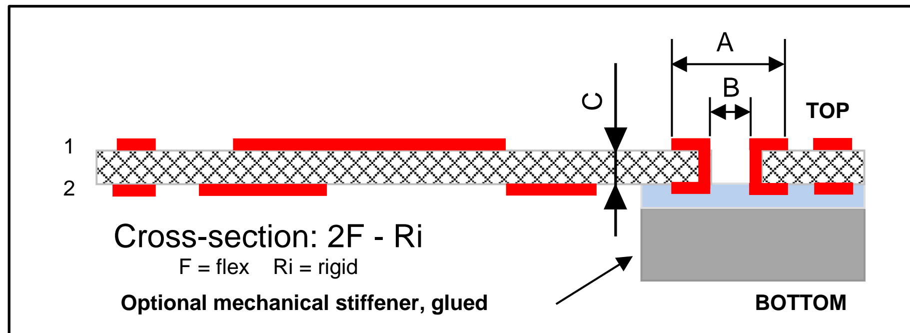
Please regard our sectional design rules xF and xF-Ri www.we-online.de

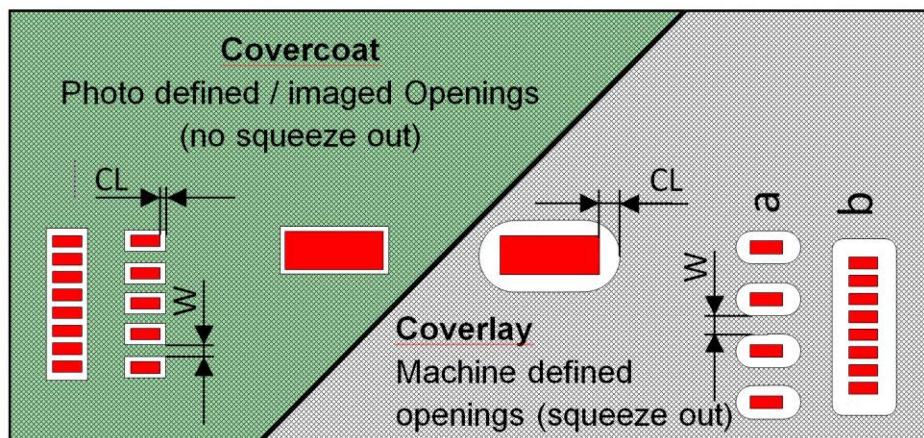
Standard design

1. Polyimide 50 µm adhesiveless, flexible PCB total thickness without stiffener 0.1 mm to 0.3 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 IPC6013
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

Design rules
Flex xF and TWINflex® xF – Ri

Application in accordance with IPC 2223 Use A: Flex-to-install
No UL marking





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

Symbol	Description	Technical standard	Advanced requirements
-	Distance copper to outline	≥ 300 µm	
-	Number x of copper layers (xF)	1 - 4	
C	Thickness of flexible material (polyimide, LCP on request)	50 µm	75 / 100 / (125)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	70µm circumferential	
CL (coverlay)	Minimum clearance of copper pad with coverlay (milled, lasered)	450µm circumferential	
W (solder mask)	Minimum bridge width photosensitive flex solder mask	70µm circumferential	
W (coverlay)	Minimum bridge width coverlay (milled, lasered)	500µm	
	Avoid vias in bending area!		
"ZIF"	**ZIF contacts thickness tolerance**	± 0.05 mm	
Usage of microvia technology: possible with 2F and 4F (staggered via):			
A (HDI)	Minimum pad diameter for microvia	350µm	300µm
B (HDI)	Finished Hole diameter lasered microvia	≈ 100µm	

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