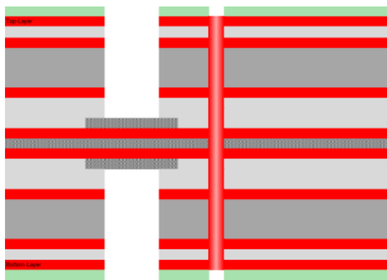
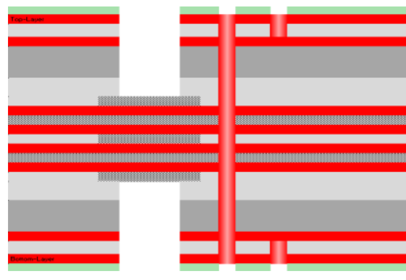
Design rules
Flex-rigid xRi – ≥ 2F – xRi

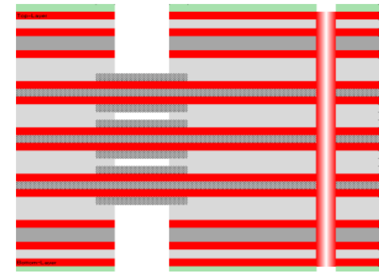
Application in accordance with IPC-2223 Use A: Flex-to-install
UL marking in accordance with UL94 or UL796F possible

These design rules apply to:

flex-rigid circuit boards with 2 or more copper layers on flexible polyimide material, internally located.


Example 8-layer (3Ri-2F-3Ri)

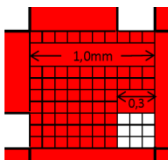

Example 8-layer (2Ri-4F-2Ri)


Example 14-layer (3Ri-4x2F-3Ri)

Nomenclature: F = flexible, Ri = rigid

Basic instructions

- Please comply with general standards such as IPC or IEC.
- Please note the valuable information and tips in the WE Flex-Rigid Design Guide*.
- Please refer to the WE Basic Design Guide for rules on conductor widths and spacings, via and pad sizes, and solder resist mask*.
- Filling of plated through holes (PTH):
  Never use open vias in solder areas! For PTH plugging (IPC Type III) always keep a clearance of 400µm to solder areas on both sides! In case of IPC Type VII (filled and capped) please ask for possible design rules (in special: line space parameters).
- Flexible circuit boards must be dried before they are assembled. Further information on this can be found on our website*.
- Copper openings in the ground or reference layers are required for drying.



Recommendation: copper openings 0.3 mm per 1 mm length of copper

- Flex-to-install bending radii: installation bending load in accordance with IPC-2223 up to 90° angle:
  - 2 copper layers: 10 x total thickness (IPC-2223 Section 5.2.4.2)
  - More than 2 copper layers: 20 x total thickness (IPC-2223 Section 5.2.4.3)
  - For use in more demanding conditions, please contact us.
- We will be happy to create the optimal delivery panel for you (best price).

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

Design rules
Flex-rigid xRi – ≥ 2F – xRi

Application in accordance with IPC-2223 Use A: Flex-to-install
UL marking in accordance with UL94 or UL796F possible

Material specifications

Material	Standard	Spec. Sheet	Description	Application
Flexible base material	IPC-4204	11	Polyimide without adhesive	Standard
Rigid material (core and prepreg)	IPC-4101	128	FR4 Tg 150°C, filled, non-halogen, low CTE(z)	Standard
LowFlow Prepreg	IPC-4101	128	Epoxy Prepreg Tg150	Standard
Solder resist	IPC-SM840		Green, photosensitive	Standard rigid areas
Coverlay / bond ply	IPC-4203	1 / 2	Polyimide cover or composite films, acrylic or epoxy adhesives	Standard: Partially in the flexible areas (also called "bikini")

Layer buildups

For standard layer buildups, please visit www.we-online.de/flex

customer	
pcb name	
WE-number	
engineer	
date	

WÜRTH ELEKTRONIK

Rigidflex 3Ri-2F-3Ri

PCB Thickness : 1,57 mm +/- 10% **Flex Thickness:** 0,16 mm +/- 0,05mm

Rigid area Structure	Flex area Thickness	Rigid area Thickness	Material description		Flex area Structure	Viatypes	Layer usage	Er	Impedance Z[Ohm] / Line / Space
Soldermask		15							
L1		45		Top-Layer					
		60	FR4 Tg150 HF						
L2		17							
		410	FR4 Tg150 HF						
L3		17							
		180	FR4 Tg150 HF						
	40		Coverlay						
L4	17	17							
	50	50	Polyimide						
L5	17	17							
	40		Coverlay						
		180	FR4 Tg150 HF						
L6		17							
		410	FR4 Tg150 HF						
L7		17							
		60	FR4 Tg150 HF						
L8		45		Bottom-Layer					
Soldermask		15							

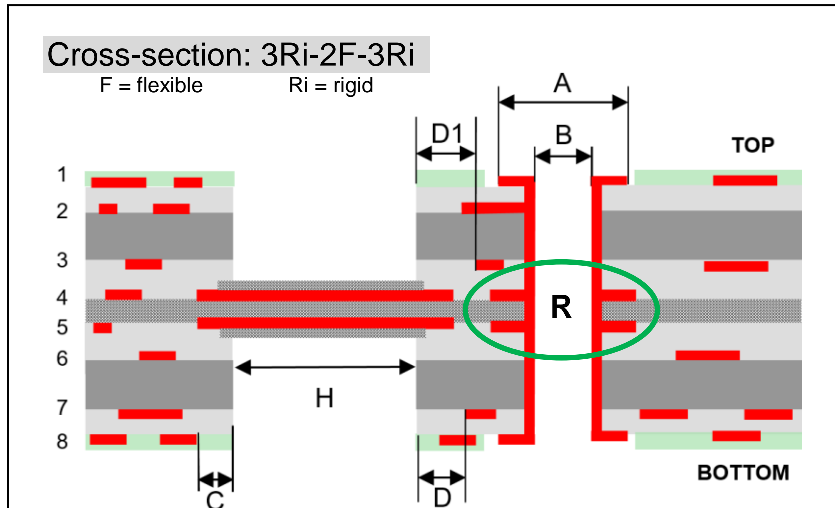
Standard design

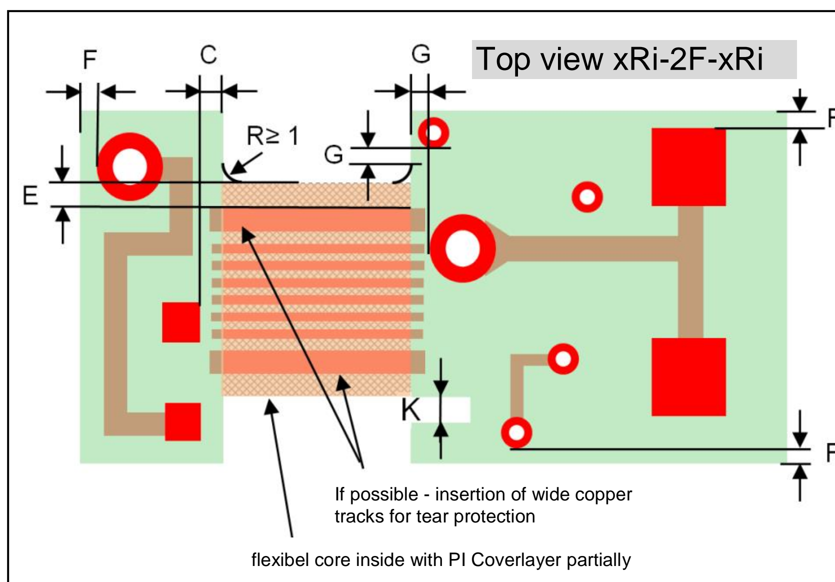
1. Polyimide 50 µm without adhesive, ED copper, PCB total thickness 1.0 mm to 1.55 mm
2. Copper layer thickness inner layer 18 µm, outer layer 12 µm + electroplating
3. Partial coverlay technique (also called "bikini")
4. Photosensitive solder resist green
5. Standard vias
6. Minimum milling diameter 1.6 mm
7. Solderable surface ENIG (electroless nickel immersion gold)
8. Packaged in ESD shrink wrap

Combination with microvia technique (from 6 layers) and buried via technique possible:
following the WE HDI Design Guide!

Design rules
Flex-rigid xRi – ≥ 2F – xRi

Application in accordance with IPC-2223 Use A: Flex-to-install
UL marking in accordance with UL94 or UL796F possible





Underwriters Laboratories Inc.®

c US

Symbol	Description	Technical standard	Advanced requirements
	Conductor widths and spacings	See WE Basic Design Guide!	
A	Minimum via pad diameter ➔➔ **Teardrops recommended** 🡐🡐	See WE Basic Design Guide!	
B	Final diameter of continuous vias	See WE Basic Design Guide!	
R	**➔➔ NFP: Do not remove non-functional pads!! 🡐🡐**		
C	Spacing, Cu – outer layer to flex-rigid transition	≥ 300 µm	
D	Spacing, Cu – to flex-rigid transition: remote inner layers	≥ 500 µm	
D1	Spacing, Cu – to flex-rigid transition: inner layers next to flex	≥ 1000 µm	
E	Distance of the conductor to the flexible contour	≥ 300 µm	
F	Spacing, exposed Cu – outside the flex-rigid transition	≥ 300 µm	
G	**2F**: Distance of via pad to flex-rigid transition	≥ 1500 µm	1000 µm
G	**>2F**: Distance of via pad to flex-rigid transition	≥ 2000 µm	1500 µm
G	**Recommendation IPC-2223C: 3.18 mm+ ½ pad diameter**		
H	Length of the flex area with 2F (please contact us if > 2F)	≥ 5 mm	≥ 2.5 mm
K	Minimum recess width directly in the flex area	1.6 mm	1.0 mm
"K"	**Countering of the flex area: no scoring permitted!**		

➔ Further specifications are possible on request. Please contact us at: flex@we-online.com