Design Rules
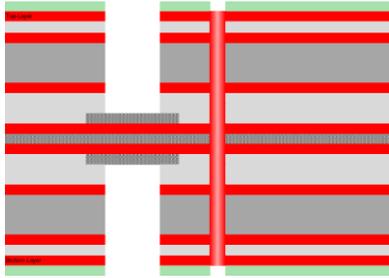## RIGID.flex xRi-≥2F-xRi
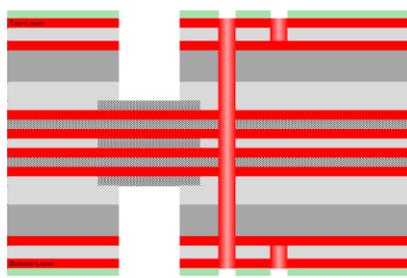
These design rules apply to:

RIGID.flex PCBs with two or more copper layers, internally located on flexible polyimide foil material. Application in accordance with IPC 2223 Use A: Flex-to-install, UL marking according UL94 and UL796F possible.
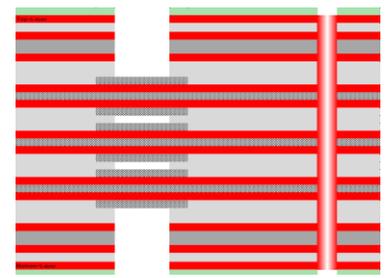
Examples:

3Ri-2F-3Ri: 8 layers

2Ri-4F-2Ri + HDI 1-6-1: 8 layers

3Ri-2F-2F-2F-3Ri: 12 layers

Nomenclature: Ri = Rigid, F = Flex

Basic instructions

- Please comply with general standards, such as IPC or IEC
- Please refer to the valuable hints and tips in our RIGID.flex Design Guide at www.we-online.com/flex .
- Please refer to our BASIC Design Rules for rules on conductor widths, spacings, via and pad sizes as well as solder mask at www.we-online.com/basic.
- Filling of PTHs (plated through holes):
  Do not use open holes in solder pads! Keep at least 400 µm distance from solder pads to holes to be plugged on both sides (Via plugging, IPC-4761 type III). For vias according to IPC-4761 type VII (filled and capped) please consult us for allowed design rules (conductor spacing)!
- Lift-off areas - attention: NO copper layout below the flex and NO vias permitted in these areas!
- Flexible and rigid-flexible circuit boards must be dried before they are assembled. Further information about this is available at www.we-online.com/dryingprocess.
- Copper removal is required in ground or reference layers for drying.
  - o 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:

  - 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 5.2.4.3)
  - For use in more demanding conditions, please contact us.
- We will be happy to create the optimal delivery panel for you (best price!).
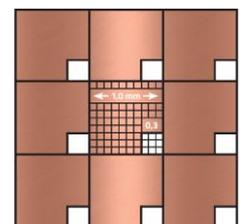
Material specifications

Material	Standard	Spec. sheet	Description	Application
Flexible base material	IPC-4202	11	Polyimide adhesive less	Standard
Rigid base material (cores, prepregs)	IPC-4101	128	FR4.1 Tg 150 °C, filled; low-halogen, low CTE(z)	Standard for RIGID.flex
LowFlow Prepreg	IPC-4101	128	FR4.1 Tg 150 °C	Standard
Soldermask	IPC-SM840		green, photosensitive	Standard lacquer in the rigid areas
Coverlay, bond ply	IPC-4203	1 / 2	Polyimide coverlay or composite films, acrylic or epoxy glue	Standard: Partially in the flexible areas (also called "bikini")

Standard Stackups

The standard stackups you will find under www.we-online.com/flex.
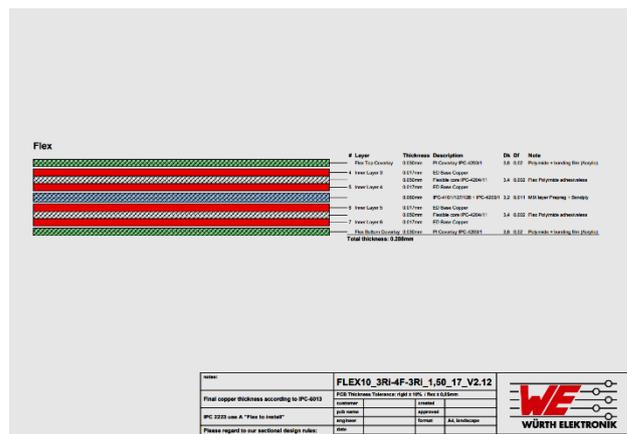
Combination with microvia technique (from 2Ri-yF-2Ri) and buried via technique is possible:

See WE HDI Microvia Design Guide.

- Buried vias after consultation by modification of the stackup (additional costs due to additional multilayer process).

Please contact us, we will find a solution for your needs: flex@we-online.com.
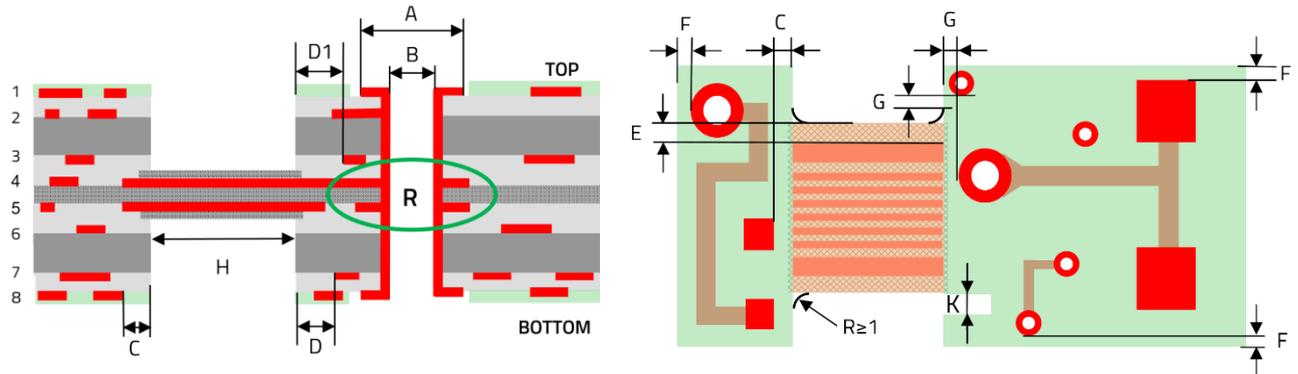
Standard design

1. Flexible foil out of Polyimide 50 µm thick, adhesive less, ED copper on both sides
2. PCB total thickness 0.8 mm to 1.55 mm
3. Copper layer thickness on inner layers 18 µm, outer layers 12 µm + plated copper
4. Partial coverlay technique in flexible are (also called "bikini")
5. All rigid areas with standard solder resist green
6. Standard PTH (Plated Through Holes)
7. Smallest milling diameter 1.6 mm
8. Solderable surface ENIG
9. Packaged in ESD shrink wrap

Design Rules

RIGID.flex xRi-≥2F-xRi

Stackup 3Ri-2F-3Ri

Top view

Symbol	Description	Technical Standard	Advanced requirements
	Line widths and spacings	see WE BASIC Design Rules!	
A	Minimum via pad diameter →**For all Pad-connections Teardrops are recommended!**	see WE BASIC Design Rules!	
B	Final diameter of PTH	see WE BASIC Design Rules!	
R	→ **NFP: Non functional / non-used pads do NOT remove!!**		
C	Spacing, Cu – outer layer to flex-rigid transition (bottom)	≥300 µm	
D	Spacing, Cu – inner layer to flex-rigid transition	≥500 µm	
D1	Spacing, Cu – to flex-rigid transition: inner layers next to flex	≥1000 µm	
E	Distance of conductor to the flexible contour	≥300 µm	
F	Spacing, exposed Cu – outside of flex-rigid transition	≥300 µm	
G	**Flexible area 2F**: Distance of via pad to flex-rigid transition	≥1500 µm	≥1000 µm
G	**Flexible area >2F**: Distance of via pad to flex-rigid transition	≥2000 µm	≥1500 µm
G	*For your information: Recommendation in IPC-2223D 5.2.2.3:*	*3.18 mm+ ½ pad diameter*	
H	Length of the flex area with 2F (for > 2F please contact us) Airgap (2F+2F…), typical	≥5 mm ≥50 mm	≥2.5 mm -
K	Minimum recess width directly at the flex area	1.6 mm	1.0 mm
„K"	Outline manufacturing of flex area: **No scoring permitted!**		
„ZIF"	**ZIF contacts thickness tolerance** (material of stiffener)	± 0.05 mm (FR4)	± 0.03 mm (PI) special stackup
-	Combination with microvia (from 2Ri-yF-2Ri) and buried via technique is possible	see HDI Microvia Design Guide!	

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