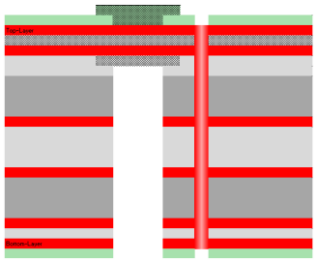
Design Rules

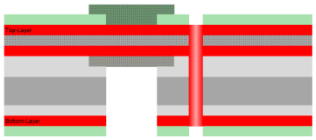
RIGID.**flex** 2F-xRi

These design rules apply to:
RIGID.flex PCBs with two copper layers on flexible polyimide material, externally located.
Application in accordance with IPC 2223 Use A: Flex-to-install, UL marking according UL94 and UL796F possible.

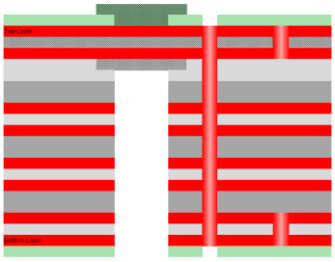
Examples:



2F-4Ri: 6-layers



2F-1Ri: 3-layers

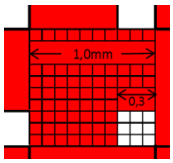


2F-6Ri + HDI 1-6-1: 8-layers

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.
  - 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)
  - 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

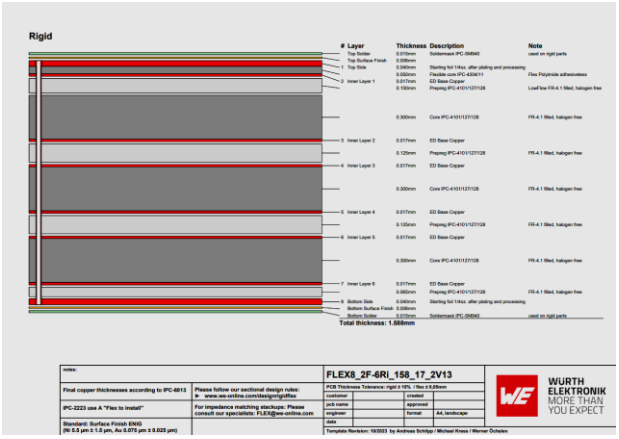
RIGID.**flex** 2F-xRi

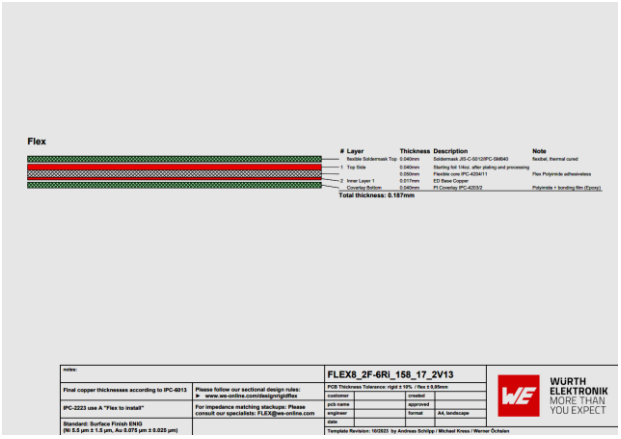
Material specifications

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
Flexible solder mask	JIS C 5012/ IPC-SM840		green	Partially in the flex area or over the entire area of the flex side
Coverlay	IPC-4203 JPCA-DG04	1 / 2	Polyimide coverlay film, acrylic or epoxy glue	Standard covering of flexible area on layer 2, optional in place of flex solder mask on top layer(surcharge)

Standard Stackups

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





Combination with microvia technique and buried via technique is possible:
See WE HDI Microvia Design Guide.

- Flex side: Microvias through Polyimide core outside the bending area.
- Buried vias after consultation by modification of the stackup (additional costs due to additional multilayer process). Buried Vias starting on layer 2 are generally not possible with this stack-up approach! Please contact us, we will find a solution for your needs: flex@we-online.com.

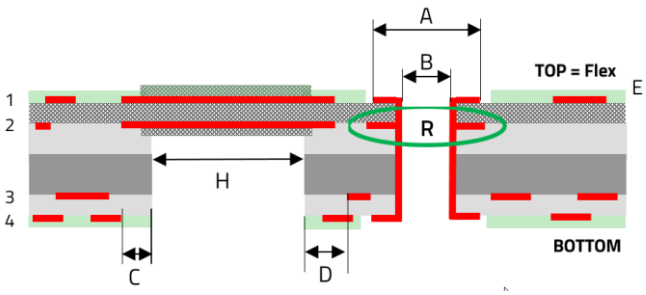
Standard design

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. Low-Flow Prepreg between flexible and rigid materials
5. Flexible area partially covered by Coverlay film the on layer 2
6. Flexible soldermask green in bending area on TOP layer or on entire flex side
7. Bottom (not flex side) or optional all rigid areas with standard solder resist green
8. Standard PTH (Plated Through Holes)
9. Smallest milling diameter 1.6 mm
10. Solderable surface ENIG
11. Packaged in ESD shrink wrap

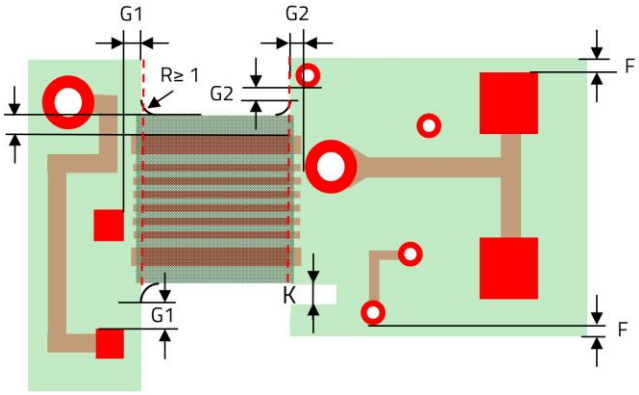
Design Rules

RIGID.**flex** 2F-xRi

Stackup 2F-2Ri



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	
E	Distance of conductor to the flexible contour	≥300 µm	
F	Spacing, exposed Cu – outside of flex-rigid transition	≥300 µm	
G1	Flexible lacquer: Spacing, pad / exposed Cu to flex-rigid transition (top)	≥800 µm	≥400 µm
G1	PI-Coverlay: Spacing to flex-rigid transition, pad / exposed Cu (top)	≥1500 µm	≥1000 µm
G1	PI-Coverlay **with UL marking**: Spacing for pad / exposed Cu (top)	≥2000 µm	≥1500 µm
G2	Spacing drillpad to flex-rigid transition: **Cu inner layers 18µm**	≥1500 µm	≥1000 µm
G2	Spacing drillpad to flex-rigid transition: **Cu inner layers 35µm or with UL marking**	≥2000 µm	≥1500 µm
G2	*For your information: Recommendation in IPC-2223D 5.2.2.3:*	*3.18 mm+ ½ pad diameter*	
H	Length of the flex area	≥5 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 and buried via technique is possible	see HDI Microvia Design Guide!	

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