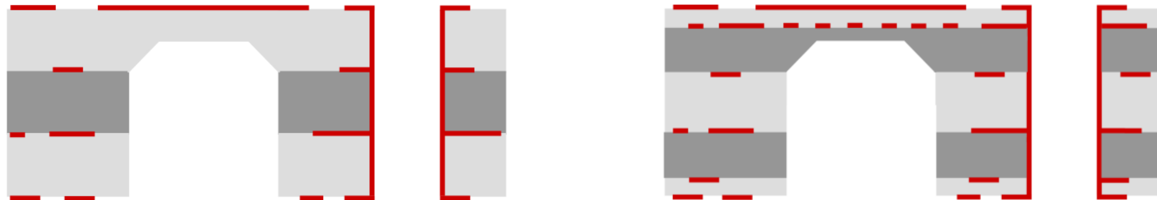
Design Rules
FR4 Semiflex

Application according IPC 2223 Use A: Flex-to-install

These design rules apply to:

bendable, semiflexible circuit boards with 1 or 2 copper layers in the bending area, externally located.



Example: 1layer in the bending area: 1Ri-3Ri Example: 2 layers in the bending area: 2Ri-4Ri

Nomenclature: Ri = Rigid, F = Flexible material is not used at all!

Basic information

- Please comply with general standards, such as IPC or IEC
- Please note 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 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).
- Flex-to-install bending radii for FR4 Tg135: only bending for assemby is allowed!
  - 1 copper layer in a Multilayer stack-up: Bending radius minimum 4mm
  - 2 copper layer and 2-layer circuit boards: Bending radius minimum 5mm
- Bending direction: only allowed in the semiflex area with copper outside the radius (milled area inside bending!)
- Important hint for the assemby: a controlled pre-bendenig using a bending tool assuring the minimum bending radius prior to the final assembly is important to simplify assembly and to prevent damage!
- Drying before assembly and soldering: FR4 Semiflex circuit boards do not contain flexible material Polyimide. So they could be processed like standard rigid FR4 circuit.
- We will be happy to create the optimal delivery panel for you (best price!)

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

Design Rules
FR4 Semiflex

Application according IPC 2223 Use A: Flex-to-install

Material	Standard	Spec. Sheet	Description	Application
Rigid material	IPC4101	24	FR4 Tg 135°C	Standard
Soldermask	IPC-SM840		green, photosensitive	Standard on rigid areas
Flexible soldermask	JIS C 5012/ IPC-SM840		green	Standard: Partially in the bending area or over the entire areas
Coverlay	IPC-4203	1 / 2	Polyimide Coverlay, Acrylic- or Epoxy- glue	Optional coverage of the flexible area (surcharge)

Layer stack-up

Do not use standard Multilayer stack-up plans for Semiflex Technology. For each project there is an individual stack-up design to be created – please ask us!

Standard design

1. PCB total thickness 0,8 mm to 1.55 mm
2. Copper layer thickness inner layers 18µm, outer layers 12µm + electroplating
3. Photosensitive solder resist green in rigid areas, flexible soldermask green in bending area
4. Standard PTHs
5. Minimum milling diameter 1.6 mm
6. Solderable surface ENIG (electroless nickel immersion gold)
7. Packaged in ESD shrink wrap

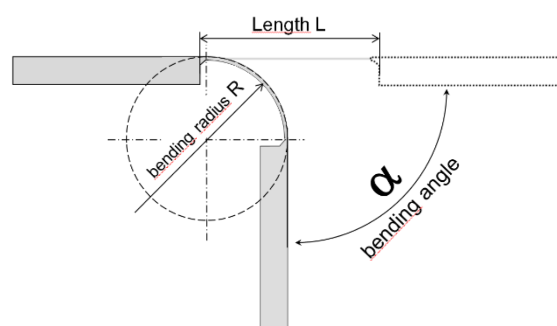
Combination with Microvia- and buried via - Technique possible:

following the WE HDI Design Guide.

Bending area: Calculation of Length „L“:

Necessary length L of the bending area is:
L = angle α x radius R x Pi / 180° + 2 x 0.4mm (bevel Y)

angle [°]	Length L of the bending area 1-layer @ bending radius [mm] 4	5
45	**3,9**	**4,7**
90	**7,1**	**8,7**
180	**13,4**	**16,5**

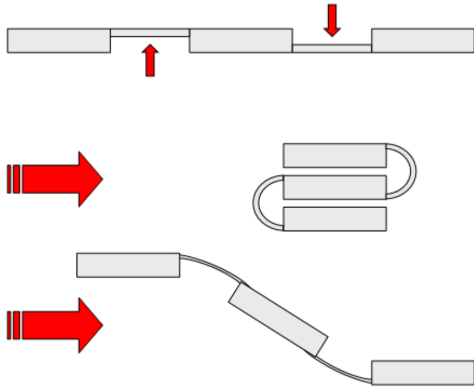


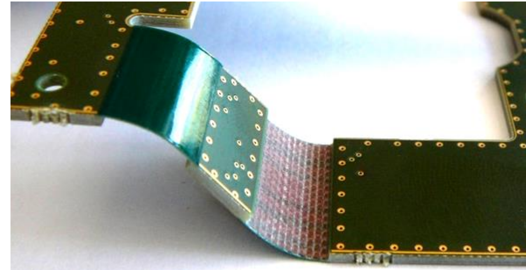
Design Rules
FR4 Semiflex

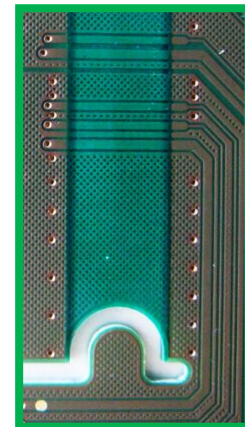
Application according IPC 2223 Use A: Flex-to-install

➔**Attention:**

Bending is only allowed with bending area / copper outside in the bending area!🡐

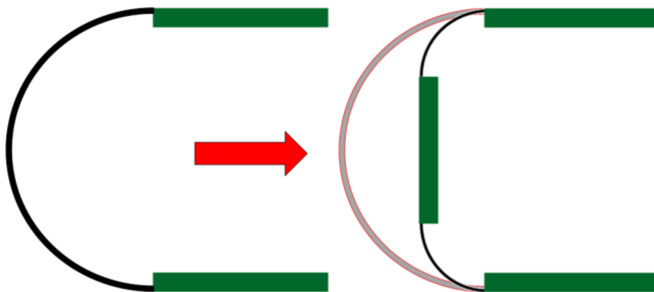




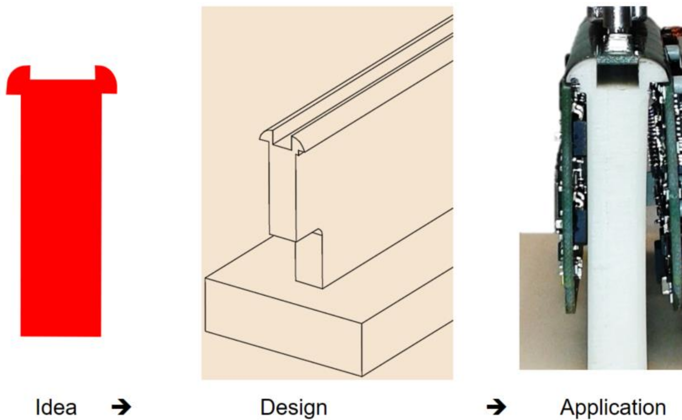


Recommendations:

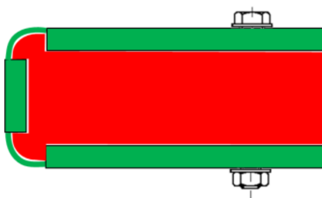
- 180° Bending replaced by 2 x 90°



- First preforming with Bending Tool before final Box Assembly!

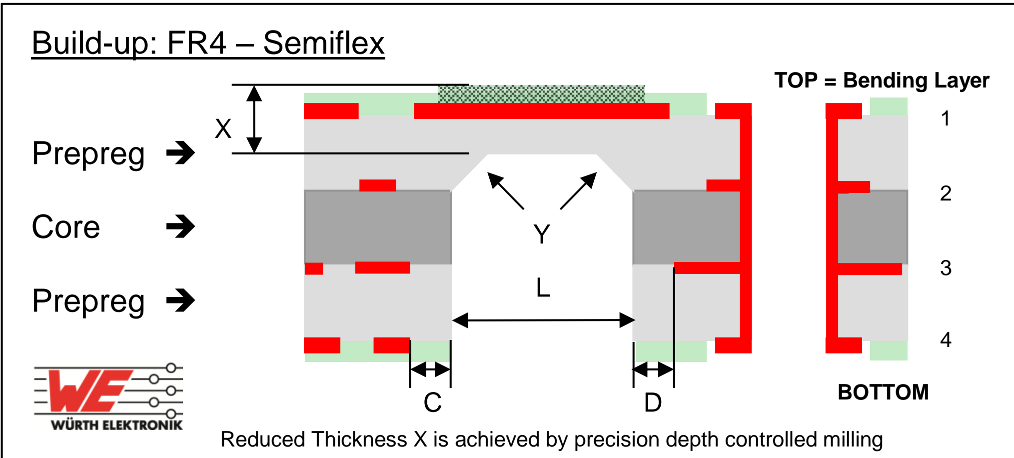


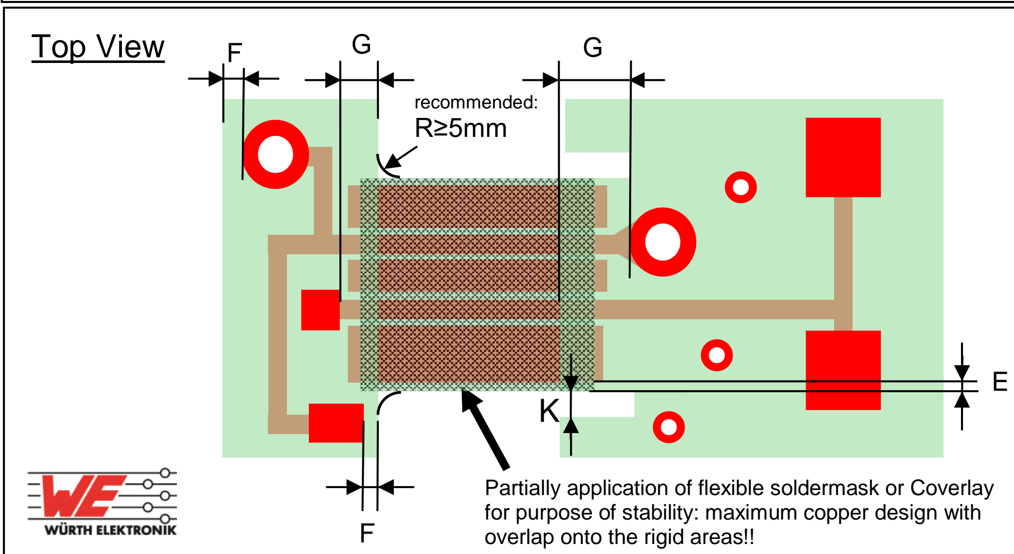
- „Mooring" radii in Housing or on Carrier to stabilize bending area.



Design Rules
FR4 Semiflex

Application according IPC 2223 Use A: Flex-to-install





Symbol	description	standard requirements	advanced requirements
	Only use of rigid material FR4	Tg135	On request
X	thickness of bending area incl. flexible soldermask/coverlay 1-layer: (depending on layer stack and copper thickness) 2-layers:	typ. 200 µm ± 50 µm ➔ project specific “WE stack-up plan”	
Y	bevel	0,4 x 45°	
C	distance Copper on outer layer – outline of bending area (Bottom)	≥ 230 µm	
D	distance Cu on inner layers – outline of bending area	≥ 230 µm	
E	distance track – outline of bending area	≥ 300 µm	
F	distance of exposed copper – outside of bending area	≥ 230 µm	
G	distance exposed Copper – bending area (top) flexible soldermask Coverlay PI	≥ 1000 µm ≥ 1500 µm	≥ 800 µm ≥ 1000 µm
K	Minimum width for „cut-in“	1,6mm	1,1mm
„K“	Outline manufacturing: **No scoring allowed!**		
L	length of bending area: depending on bending angle	See also page -2-	
„L“	bending area: for purpose of stability: maximum copper design	max. 200µm Isolation between lines	
„L“	Bending direction: copper on outerlayer to tension	See also page -3-	
„L“	Minimum bending radius, 1 copper layer in bending area: **2 Layer pcb**	5 mm	Individ. qualification
„L“	Minimum bending radius, 1 copper layer in bending area: **Multilayer cb**	4 mm	Individ. qualification
„L“	Minimum bending radius, 2 copper layers in bending area	5 mm	Individ. qualification
„L“	max. quantity of bending cycles (@ minimum bending radius)	Typ. 10	Individ. qualification
-	Solder Finishes	ENIG, imm. Sn	
	Rules for line width/space, Via- and Pad-sizes, Soldermask design	see Basic Design Guide!	
	combination with microvia- und buried via - Technology possible	see HDI Design Guide!	

➔ Enhanced specifications on request – please ask us ! ➔