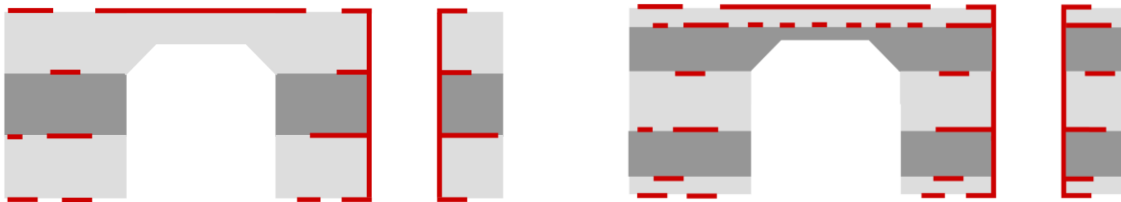
Design Rules

SEMI.**flex**

WÜRTH ELEKTRONIK
MORE THAN YOU EXPECT

These design rules apply to:
Rigid multilayer with partially bendable, semi-flexible bending area. The PCBs have 1 or 2 copper layers in the deep milled area for an application according to IPC-2223 Use A: Flex-to-install.

Examples (shown without solder mask):



1Ri-3Ri: 1 layer in the bending area

2Ri-4Ri: 2 layers in the bending area

Nomenclature: Ri = Rigid, F = Flex material is not used!

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 400µm 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)!
- Flex-to-install bending radii for FR4.0 Tg135: only installation bending load is permissible
  - o 1 copper layer with multilayer: bending radius at least 4mm
  - o 2 copper layers and 2-layer PCB: bending radius at least 5mm
- Bending direction: only allowed in the semi-flexible area with copper outside the radius (milled area inside bending!)
- Important hint for the assembly: a controlled pre-bending using a bending tool assuring the minimum bending radius prior to the final assembly is important to simplify assembly and to prevent damage!
- Note on drying before the soldering process: SEMI.flex PCBs can be processed like standard PCBs. Special drying is not necessary as no polyimide flex material is used.
- We will be happy to create the optimal delivery panel for you (best price!)

Material specifications

Material	Standard	Spec. sheet	Description	Application
Rigid material	IPC-4101	21	FR4.0 Tg 135°C	Standard for SEMI.flex
Soldermask	IPC-SM840		green, photosensitive	Standard in rigid areas
Flexible soldermask	JIS C 5012/ IPC-SM840		green	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 stackup

Do not use standard Multilayer stackup plans for SEMI.flex Technology. For each project, there is an individual stackup design to be created – please ask us!

Standard configuration

1. PCB total thickness 0,8mm to 1.55mm
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 1.6mm
6. Solderable surface ENIG (electroless nickel immersion gold)
7. Packaged in ESD shrink wrap

Combination with HDI Microvia- and Buried via - Technique possible:

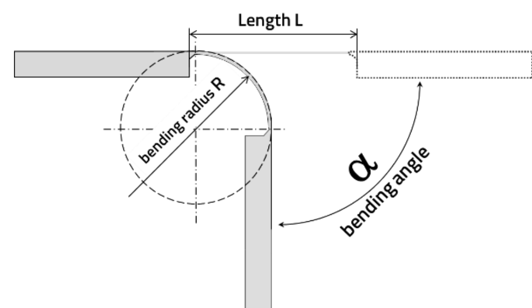
Follow the WE HDI Microvia Design Guide for Microvia and Buried Via design.

Recommendations for SEMI.flex design

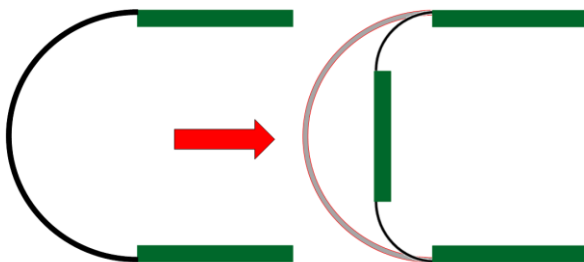
- Calculation of Length "L" for SEMI.flex area

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

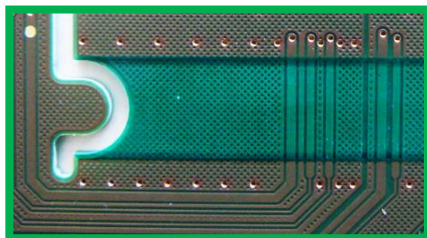
	Length L of the bending area @ bending radius [mm]	
angle [°]	4	5
45	3,9	4,7
90	7,1	8,7
180	13,4	16,5



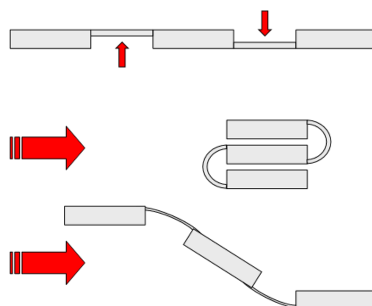
- 180° bending replaced by 2 x 90° with rigid middle part

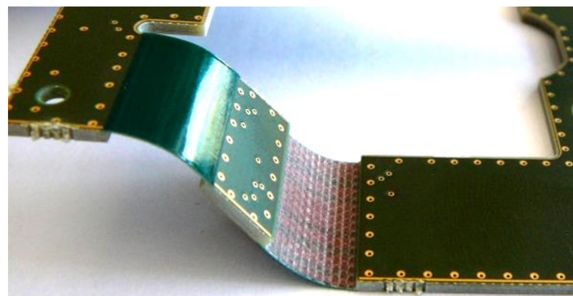


- Fill up free areas in the bending area with hatched copper



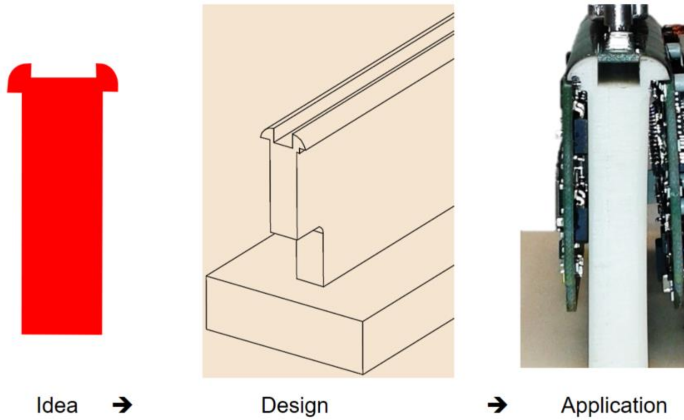
- Design S-shaped bend accordingly with milling from two side!



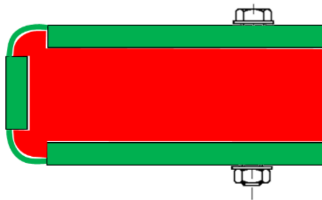


Recommendations for SEMI.flex assembly

- A defined pre-bending with the aid of a bending tool that ensures the minimum permissible bending radius facilitates housing assembly and prevents damage!



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

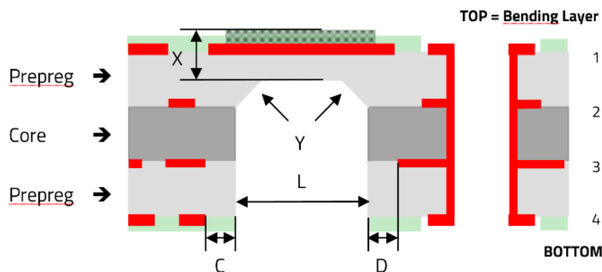


➔**Attention:**
Bending is only allowed with bending area / copper outside in the bending area!🡐

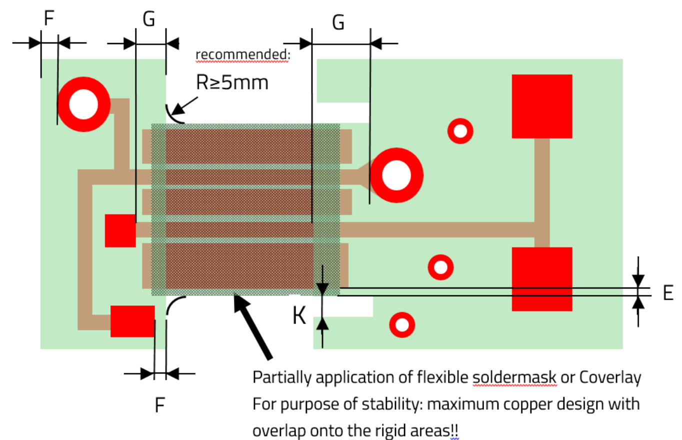
Design Rules

SEMI.**flex**

SEMI.flex stackup



Top view



Symbol	Description		Standard requirements	Advanced requirements
	Only use of rigid material FR4.0, no flexible foils		Tg135°C	see BEND.flex
X	thickness of bending area incl. flexible soldermask/coverlay (depending on layer stack and copper thickness)	1-layer: 2-layers:	typical 200µm ± 50µm ➔ project specific layerstack	
Y	Bevel		0,4 x 45°	
C	Distance Copper on outer layer – outline of bending area (Bottom)		≥230µm	
D	Distance Cu on inner layers – outline of bending area		≥230µm	
E	Distance copper feature – outline of bending area		≥300µm	
F	Distance of exposed copper – outside of bending area		≥230µm	
G	Distance exposed Copper – bending area (top)	flexible soldermask Coverlay PI	≥800µm ≥1500µm	≥400µm ≥1000µm
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 -3-	
„L"	Bending area: for purpose of stability: maximum copper design		max. 200µm Insolation between Cu-structures	
„L"	Bending direction: copper on outer layer to tension		See also page -3-	
„L"	Minimum bending radius, 1 copper layer in bending area: **2 Layer PCB**		5mm	Individual qualification
„L"	Minimum bending radius, 1 copper layer in bending area**: Multilayer PCB**		4mm	Individual qualification
„L"	Minimum bending radius, 2 copper layers in bending area		5mm	Individual qualification
„L"	Maximum quantity of bending cycles (@ minimum bending radius)		Typical 10	Individual qualification
-	Solder Finishes		ENIG, immersion Sn	
	Rules for line width/space, Via- and Pad-sizes, soldermask design		See BASIC Design Rules!	
-	Combination with HDI microvia- und buried via - technology possible		See HDI Microvia Design Guide!	

Enhanced specifications are possible on request – please ask us: flex@we-online.com