

Flex & RigidFlex Technical Guidelines

January, 2021

Gatema - Flex & RigidFlex products

Content	
Title	Page
Flex type code	2
Basic configuration	3
Flex PCB	3
Outer RigidFlex PCB	4
Inner RigidFlex PCB	5
Available materials	6
Basic materials	6
Prepregs	6
Covering materials	6
Adhesive for stiffeners	6
Materials for stiffeners	6
Limitation of production	7
Surfaces	7
Coverlay pad clearance	7
RigidFlex transition zone	7
Other limitations	7
General recommendations	8
Number of bending cycles	8
Choose right material	8
Other recommendation:	8
Data output	9
Rigid Flex PCB data output	9
PCB panelisation	9

Flex type code

Flex and rigidflex types are described using short code which describe number of copper layers. Code shows position of flexible cores at multilayer RigidFlex PCB too.

Code format:

- **xRi-yF-zRi**

- x** ... Number of copper layers above flex core (on rigid area of PCB)
- y** ... Number of copper layers on flex core
- z** ... Number of copper layers below flexible core (on rigid area of PCB)

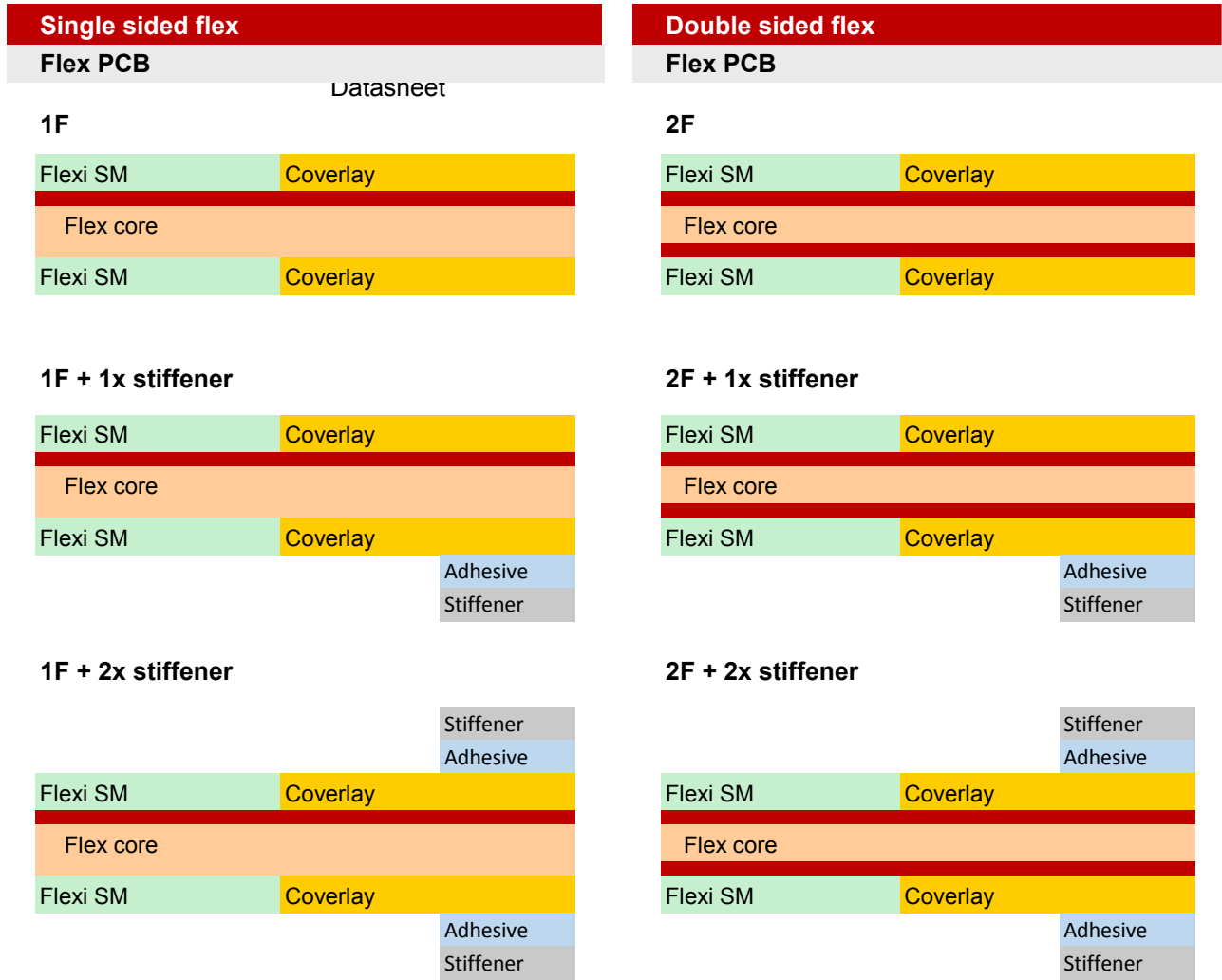
- Ri** ... Rigid core or stiffener
- F** ... Flex core

- Total number of layers is equal to: $x + y + z$

- Example of our flexi PCB configurations:

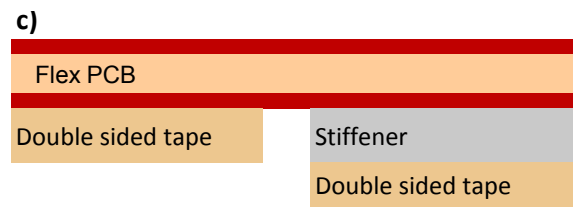
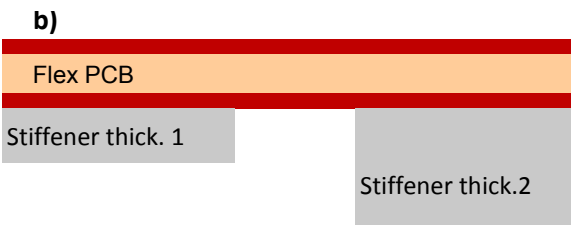
1F	...	Single copper layer flexible PCB
2F	...	Two copper layer flexible PCB
1Ri-1F	...	Total of two copper layers with one flex on the outer layer
3Ri-2F	...	Total of five copper layers with two flex on the outer layer
1Ri-2F-1Ri	...	Total of five copper layers with 2 flex on the inner layers

Basic configurations of Flexible PCB



Special possibilities

- a) Different stiffener thickness from each side are allowed
- b) Also more stiffener thickness from one side is allowed
- c) Double sided tape on flex or stiffener for sticking PCB to final product



Solder mask

For Flexibe PCB we use only green solder mask suitable for SMT components:

- Elpemer SD 2463 FLEX-HF [| Datasheet](#)

Basic configurations of Outer RigidFlex PCB

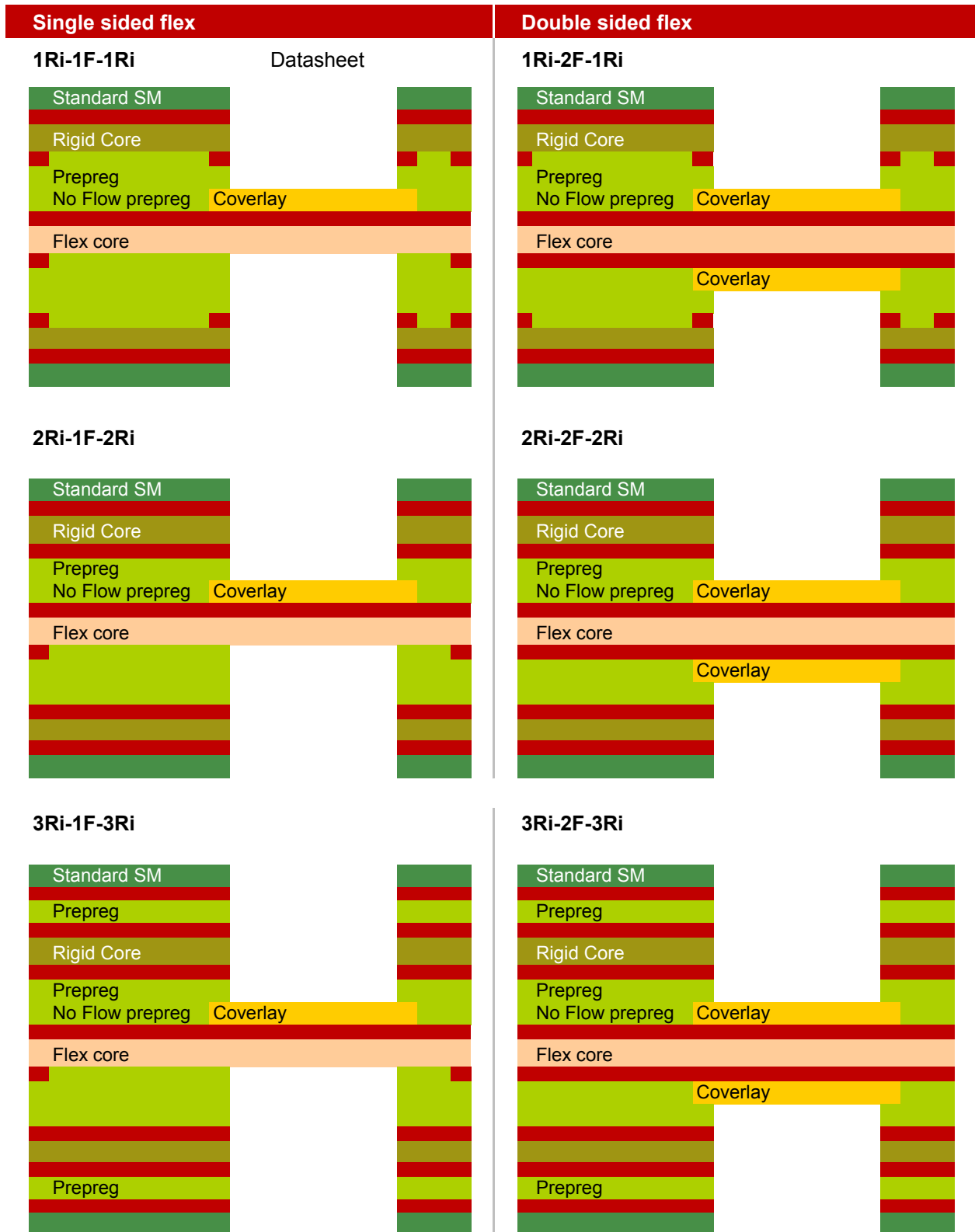


Solder mask

For Outer-RigidFlex PCB we use only green soldermask suitable for flexible parts of RigidFlex PCB:

- Elpemer SD 2460 UV-FLEX-HF | [Datasheet](#)

Basic configurations of Inner RigidFlex PCB



Materials

Basic materials					
Adhesiveless Flex cores	Core	Cu	Cu type	Tg	Datasheet
Pyralux AP	50 µm	35 µm	RA	220°C	Datasheet
Pyralux AG	50 µm	18 µm	RA	220°C	Datasheet
Rigid cores					
Isola IS400	100-1500 µm	18-35 µm		150°C	Datasheet
PCL370HR	200-1200 µm	18-35 µm		170°C	Datasheet
Prepregs					
Standard prepregs	Tg				Datasheet
Isola IS400	150°C				Datasheet
PCL370HR	170°C				Datasheet
No Flow Prepreg					
Arlon 49NP	170°C				
Covering materials					
Adhesive coverlays	Thickness	Adhesive		Datasheet	
Pyralux LF	25 µm	25-50 µm		Datasheet	
Standard Soldermask	Color			Datasheet	
Elpemer GL2467 SM-DG	green			Datasheet	
Other color on request	white, red, blue				
Flexi Soldermask	Color	Note		Datasheet	
Peters SD 2463 FLEX-HF	green			Datasheet	
Peters SD 2460 UV-FLEX-HF	green	only screen printing only flexible parts of Outer RigidFlex		Datasheet	
Adhesive for stiffeners					
Double coated tape	Adhesive	Max. temp.		Datasheet	
3M 467MP	50 µm	204°C		Datasheet	
3M 468MP	130 µm	204°C		Datasheet	
Adhesive sheets					
Pyralux LF	25 µm	288°C		Datasheet	
Stiffener basic material					
FR4	Thickness	Tg		Datasheet	
Isola DE104	100-1500 µm	135°C		Datasheet	
Isola IS400	100-1500 µm	150°C		Datasheet	

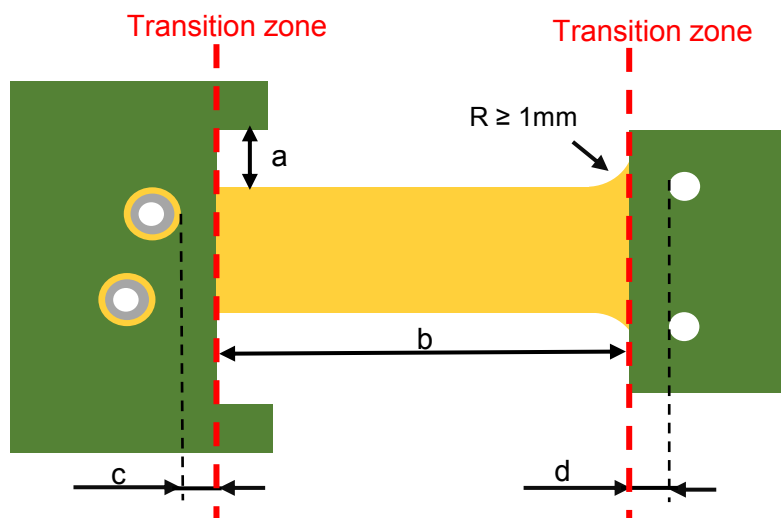
*RA Rolled copper; *ED Elektrodeposited copper

Limitation of production

Surfaces		
	Flexi PCB	RigidFlex PCB
HAL/HAL PbFree	No	Yes
Imersion Ni/Au	Yes	Yes
Galvanic Ni/Au	Yes	Yes
Imersion Tin	Yes	Yes

Coverlay pad clearance	
Min. Bridge	350 μm
Min. Pad clearance	250 μm

Rigid Flex Transition zone		
Legend	Description	Value
Transition zone	Is outline at which the layer structure changes from a rigid area to a flex only area and vice versa	
a	Countering of flex area	Min. 1.6 mm
b	Length of flex area	Min. 5 mm
c	Distance of via pad to Transition zone	Min. 1.5 mm
d	Distance of NPTH pad to Transition zone	Min. 0.5 mm



Other	
Blind	Yes
Burried via	Yes (only Rigid cores)
Max. PCB dimension: 1-2 Layer Flex	275 mm x 428 mm
Max. PCB dimension: RigidFlex	267 mm x 382 mm
Minimal plated via	0,1 mm
Scoring of flexible area in RigidFlex	No
Min. track/isolation Flex	100 μm
Min. track/isolation RigidFlex	100 μm

General recommendations

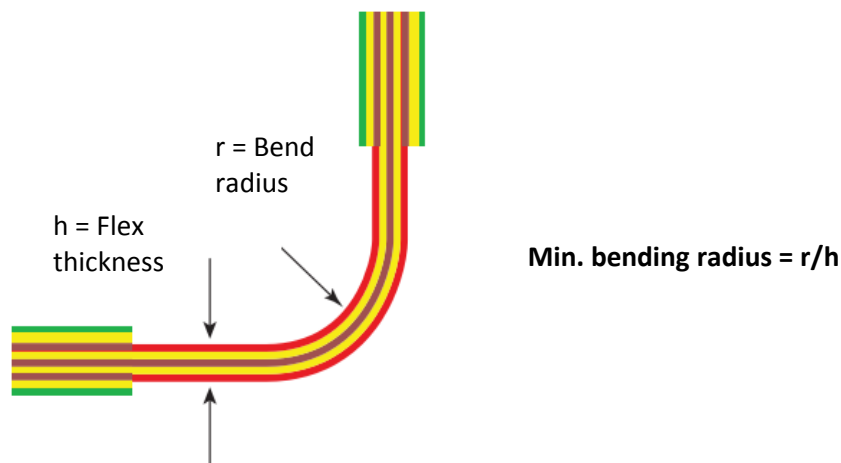
We recommend following the design recommendations listed in **IPC-2223 Sectional Design Standard for Flexible/Rigid-Flexible Printed Boards** when designing a Flex or RigidFlex PCB.

IPC standard is available in online store:

shop.ipc.org

Datasheet

Flexi PCB types according to number of bending cycles		
Types	Number of bending	Min. bending radius
Dynamic	Frequent	100-150 x flex layer thickness
Semi-Dynamic	Max. 20x	> 20 x flex layer thickness
Stable	Bend to install	> 10 x flex layer thickness



Many factors play role in PCB bend area reliability such as:

- Number of copper flex layer => Less layers better bending
- Copper thickness => Thinner copper better bending
- Copper type => RA (Rolled copper) suitable for dynamic application or ED (Electrolytically deposited copper) suited for Stable and Semi-Dynamic application
- Covering of flex area => Coverlay for dynamic application vs Flexi solder mask for Semi-dynamic and Stable

How to select the right material			
Flex type	Dynamic	Semi-Dynamic	Stable
Flex covering			
Covering type:	Coverlay	Coverlay or flexi SM	Coverlay or flexi SM
Material:	Pyralux LF	Pyralux LF Elp. SD 2463 FLEX-HF* Elp. SD 2460 UV-FLEX-HF**	Pyralux LF Elp. SD 2463 FLEX-HF* Elp. SD 2460 UV-FLEX-HF**
Flex core			
Copper type:	RA copper	RA or ED copper	RA or ED copper
Material:	Pyralux AP	Pyralux AP or Pyralux CG	Pyralux AP or Pyralux CG

* minimal bend radius (90°) 1.5 mm; suitable for SMD pad clearance

** minimal bend radius (90°) 1 mm (only for flexible parts of Outer RigidFlex)

Other recommendation:

- Track width and spacing in flexible areas should be as large as possible
- All soldering pads should be as large as possible
- Use tear drop for connection between tracks and solder pads in flexible areas
- If tracks are doublesided, they should be shifted to avoid placing below each other
- Use round milling transitions.
- Use hatched copper for GND in flexible areas.
- Use a stiffener to achieve higher thickness.
- If possible insert wide copper track on outer of flexible part for tear protection

For PCB designers

Rigid Flex PCB data output

- Create special layer "bend area", which contains border of flexible PCB area
- If you use Eagle, create layer: BendArea
- Example of BendArea borders (red color)



PCB panelisation

- V-scoring is not allowed (Flexible PCB)
- Use gap between pieces of at least 10 mm (Flexible PCB)