Design Rules

SLIM.**flex** xF and (xF)PTH

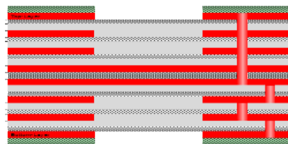
These design rules apply to:

Flexible circuit boards in anylayer microvia technology with 3 to 8 copper layers, stacked and staggered microvias.

- Application in accordance with IPC-2223 Use A: Flex-to-install on flexible polyimide material.
- Optionally with PTH (Plated Through Hole) for extra charge with restricted design rules.
- Optionally with glued mechanical stiffener or solder carrier (for extra charge).
- No UL-marking. All materials are UL-listed.

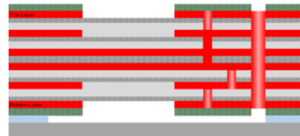
Examples:

SLIM.flex 8F



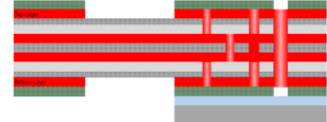
Standard: Only Microvias used

SLIM.flex (6F)PTH-Ri



Option: Solder carrier, PTH

SLIM.flex (4F)PTH-Ri

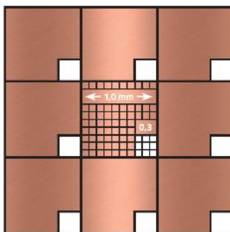


Options: Stiffener, PTH

Nomenclature: F = Flex, Ri = Rigid, i.e. Stiffener or solder carrier out of FR4, ()PTH extra

Basic instructions

- Please comply with general standards, such as IPC or IEC.
- Please take note of the useful information and tips in our WE Flex-Solutions Design Guide at www.we-online.com/flex.
- Flexible circuit boards must be dried before they are assembled. Further information about this is available at www.we-online.com/dryingspecification.
- 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:
  - 1 or 2 copper layers: 10 x total thickness (IPC-2223 section 5.2.3.3)
  - From 3 copper layers: 20 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

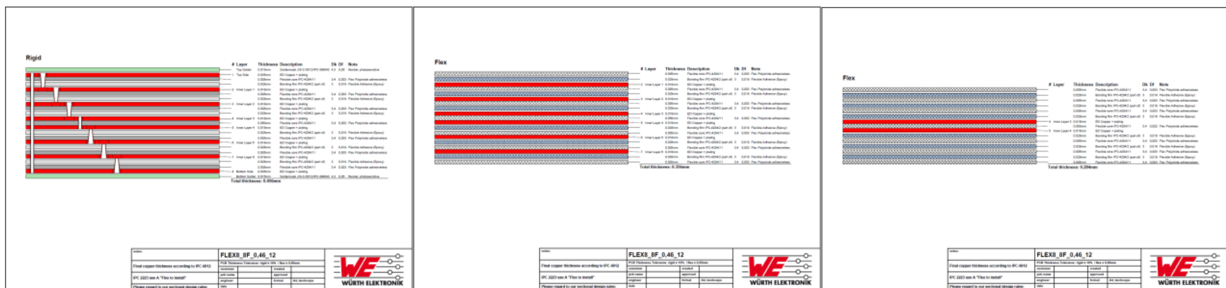
SLIM.**flex** xF and (xF)PTH

Material specifications

Material	Standard	Spec. sheet	Description	Application
Flexible base material	IPC-4204	11	Polyimide adhesiveless	Core of SLIM.flex
	IPC-4204	2	Polyimide with glue	Build-up layers of SLIM.flex
Rigid material	IPC4101	21	FR4.0 Tg135 °C	Standard for stiffener and solder carrier
Flex solder mask	JIS C 5012/ IPC-SM840		green, photosensitive	Standard
Coverlay	IPC-4203	1 / 2	Polyimide covering film 25 µm, acrylic or epoxy glue (multilayer process)	Optional in place of flex solder mask (surcharge)
Transfer adhesive film			Modified acrylic glue, Foil 50 µm thick	Cold bonding process for stiffeners

Standard Stackups

Standard stackups see www.we-online.com/slimflex.



Standard design

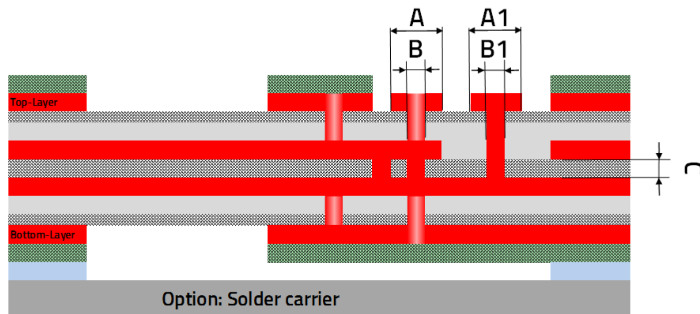
1. Core Polyimide 50 µm adhesiveless, sequential lamination of 5 µm Polyimide foil + Epoxy glue
   Minimum dielectric strength 500 VDC at minimum dielectric thickness of 20 µm
2. Base Copper thickness inner layers 17 µm, exterior layers 9 µm + electroplating
3. Flexible photosensitive solder resist green
4. Standard vias are laser drilled microvias, plating thickness according to IPC-6013
5. Outline lasered or milled, smallest milling diameter 1.6 mm. V-scoring not permitted!
6. Solderable surface ENIG (electroless Nickel – immersion Gold)
7. Packaged in ESD shrink wrap

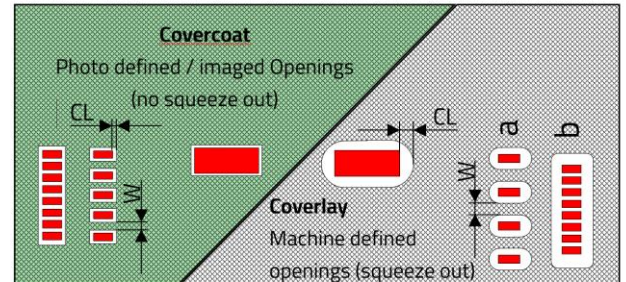
Design Rules

SLIM.**flex** xF and (xF)PTH

Stackup SLIM.**flex 4F-Ri**

Standard: Microvias only





a: single Coverlay opening larger than pad
b: window opening for all pads

Symbol	Desicription	Technical Standard	Advanced requirements
	Line widths and spacing → microvias only	75 µm/75 µm	
A	Minimum pad diameter for microvia	225 µm	200 µm
B	Finished hole diameter of lasered microvia	70 µm	70 µm
A1	Minimum pad diameter for microvia 1-3 stackup 1-2-1	-	250 µm
B1	Finished hole diameter of lasered microvia 1-3 stackup 1-2-1	-	100 µm
Important:	**Avoid vias in bending areas! Use Teardrops**		
-	Distance copper to outline	≥300 µm	≥100 µm (Laser)
-	Number x of copper layers (xF)	3 up to 8	
C	Thickness of flexible core (polyimide)	50 µm	75 µm/100 µm
-	Thickness of cold-bonded stiffener made of FR4 material	0.1 – 0.5 mm	0.5 – 0.8 mm
	Thickness of cold-bonded solder carrier made of FR4	0.8 mm	0.8 mm
-	Thickness of glue for stiffener or solder carrier	50 µm	
CL (soldermask)	Minimum clearance of copper pad with flex solder mask	40 µm circumferential	
CL (Coverlay)	Minimum clearance of copper pad with coverlay	450 µm circumferential	
W (soldermask)	Minimum bridge width photosensitive flex solder mask	70 µm	
W (Coverlay)	Minimum bridge width coverlay (milled, lasered)	500 µm	
„ZIF"	**ZIF contact thickness tolerance**	± 0.05 mm	

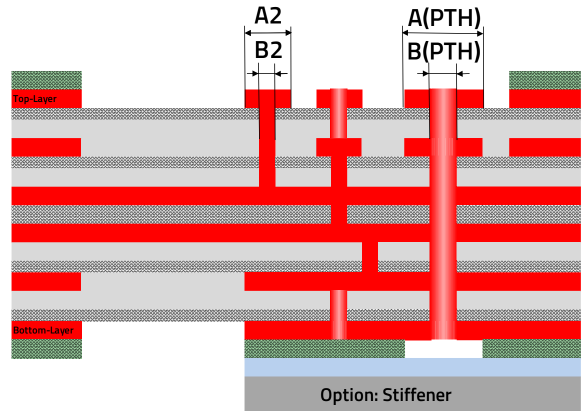
Further specifications available on request, please contact us: flex@we-online.com

Design Rules

SLIM.**flex** xF and (xF)PTH

Stackup SLIM.**flex (6F)PTH-Ri**

Option: Microvias and PTH (surcharge)
only deviating parameters



Symbol	Desicription	Technical Standard	Advanced requirements
	Line widths and spacing → PTH and microvias	75 µm/100 µm	
A(PTH)	Minimum pad diameter for microvia	450 µm	400 µm
B(PTH)	Finished hole diameter of PTH, typical	200 µm	150 µm
A2	Minimum pad diameter for microvia 1-3 stackup 2-2-2	-	225 µm
B2	Finished hole diameter of lasered microvia 1-3 stackup 2-2-2	-	85 µm
Important:	**Avoid vias in bending areas! Use Teardrops**		
	Do NOT remove non functional / non- used pads!		

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