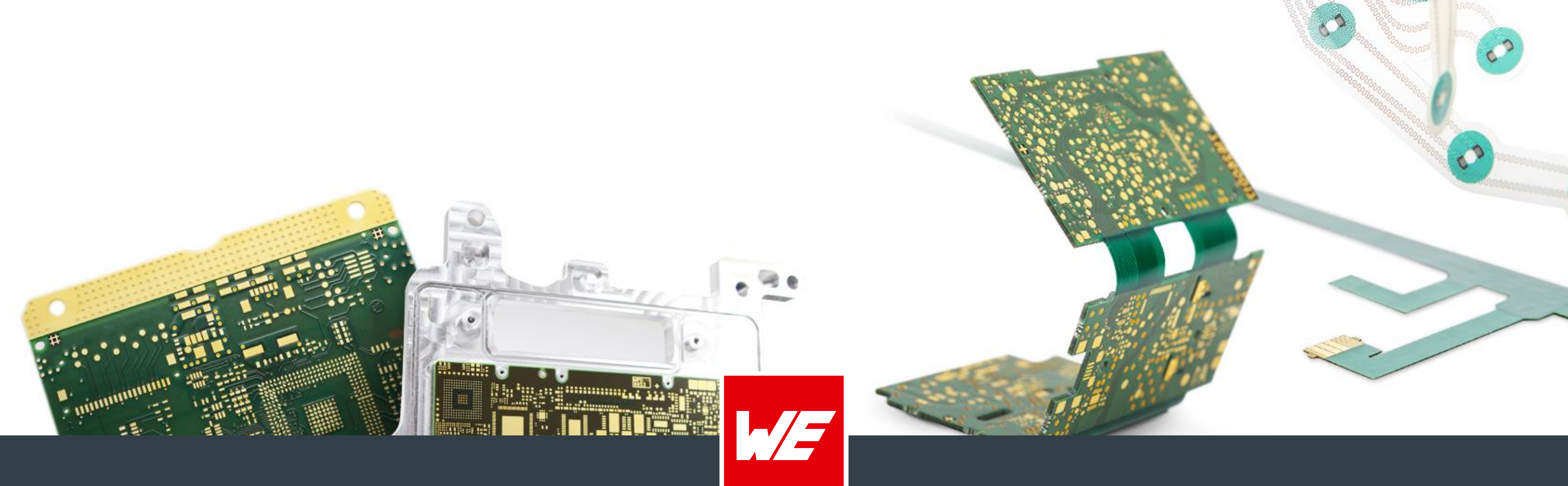


PCB PRODUCTION, PART 3:
RIGID.FLEX PCBS, CONTINUED

Guido Biernat

WEBINAR

WÜRTH ELEKTRONIK MORE THAN YOU EXPECT

AGENDA

1) Review Webinar PCB Production Part 2

2) Application Examples for
   1. 3Ri-2F-3Ri Torque tool
   2. 3Ri-2F-3Ri small industrial camera
   3. 4Ri-4F-4Ri modern High-Speed camera
   4. Standard stackups WE Circuit Board Technology

3) Production Processes of a RIGID.flex 3Ri-2F-3Ri (8 Layers)
   1. Owerview of all production processes
   2. Presentation of the processes of the rigid-flex technology
   3. Design Rules

Guido Biernat
Technical Project Management

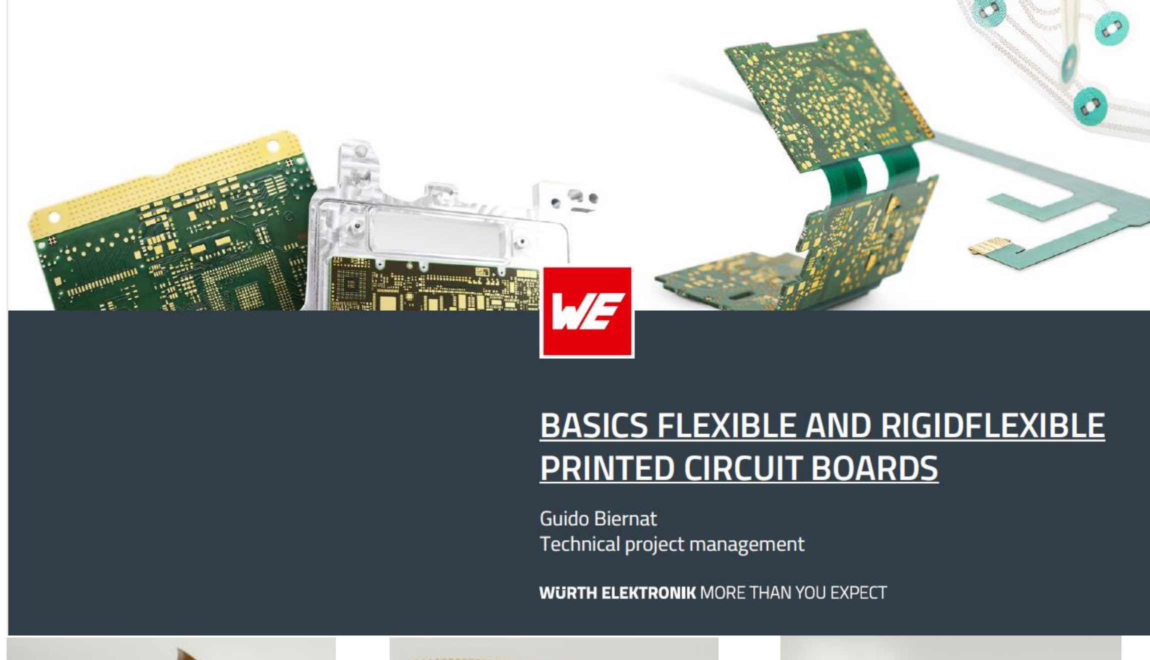
REVIEW

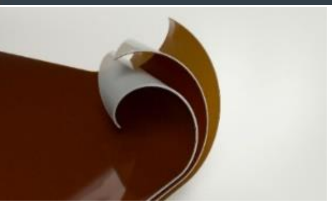
Webinar PCB Production part 2

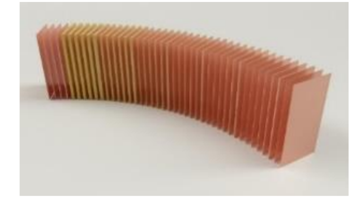
The following points were presented in this webinar:

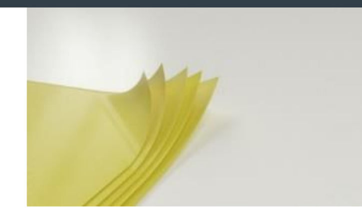
- The difference between flexible and rigid-flex printed circuit boards
- The base materials for RIGID.flex printed circuit boards
- The manufacturing processes of a RIGID.flex 1F-7Ri PCB Flex layer on the outside (Top)

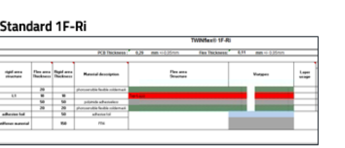
- **Printed circuit board production, part 2: RIGID.flex**
- **Webinar Archive (all)**

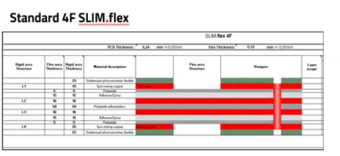


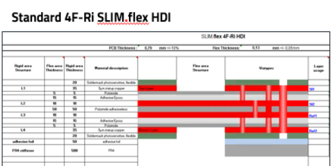


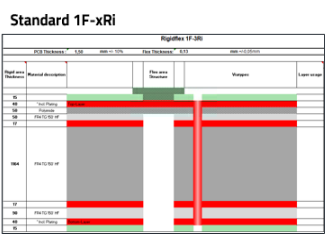


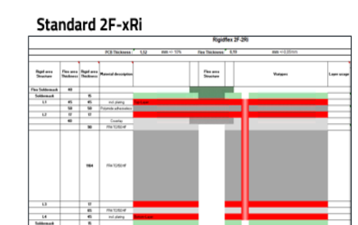


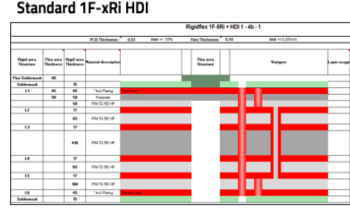








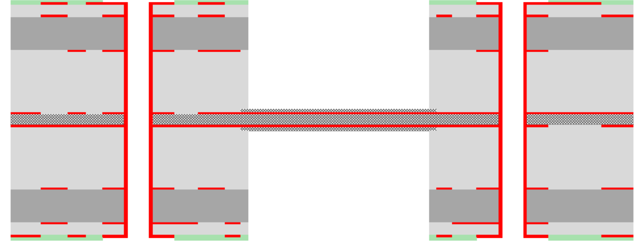




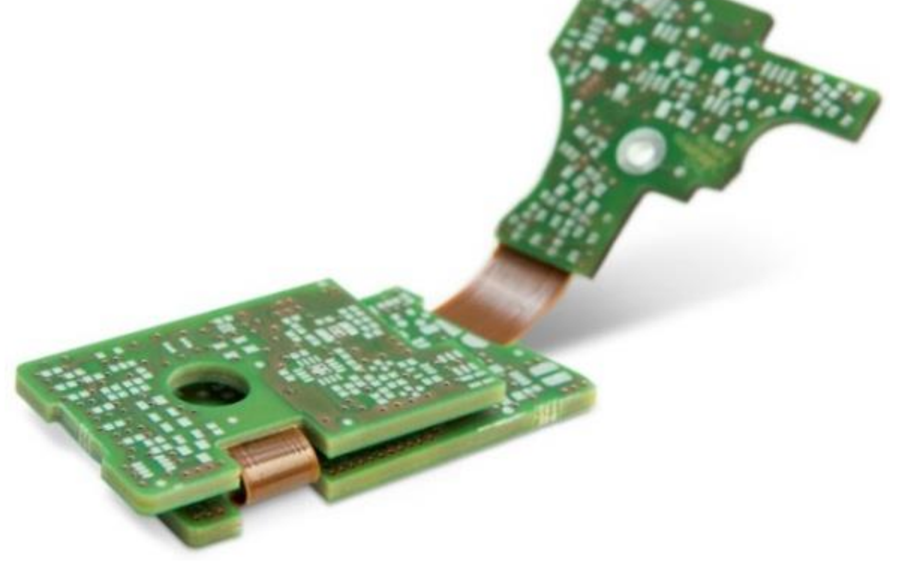
APPLICATION EXAMPLE

Torque Tool Industry

- Stackup RIGID.flex 3Ri-2F-3Ri
- 50µm Polyimide core , FR-4.1 TG 150°
- Microvias double sided



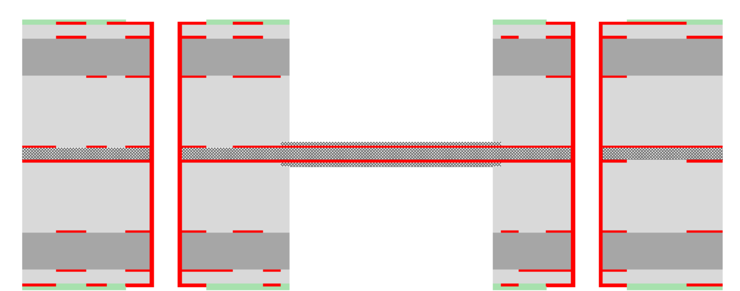
- Miniaturization
- Reliability
- System advantages

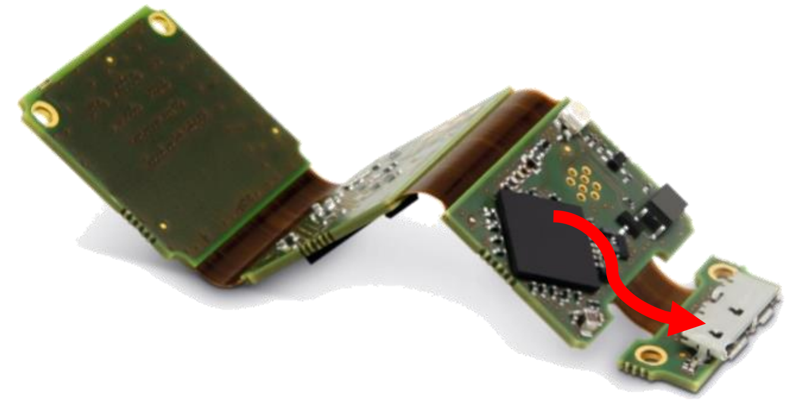


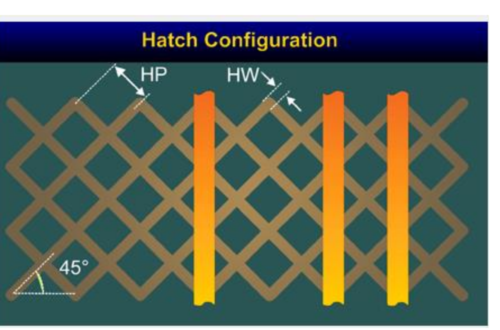
APPLICATION EXAMPLE

Industrial camera with a small construction

- Stackup RIGID.flex 3Ri-2F-3Ri
- Impedance defined signals, Rdiff 90Ω
- Use of Hatch structures (mesh design)
- USB connector on a separate rigid section
- Avoidance of connectors
- Saving of footprint areas





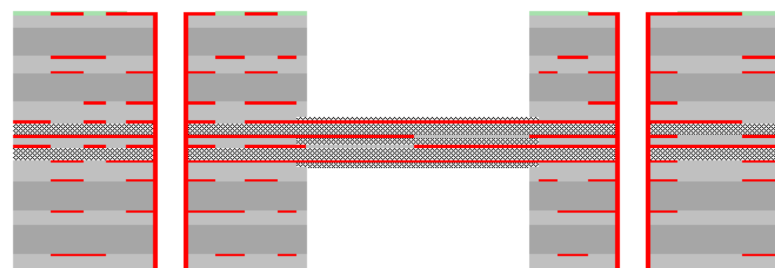


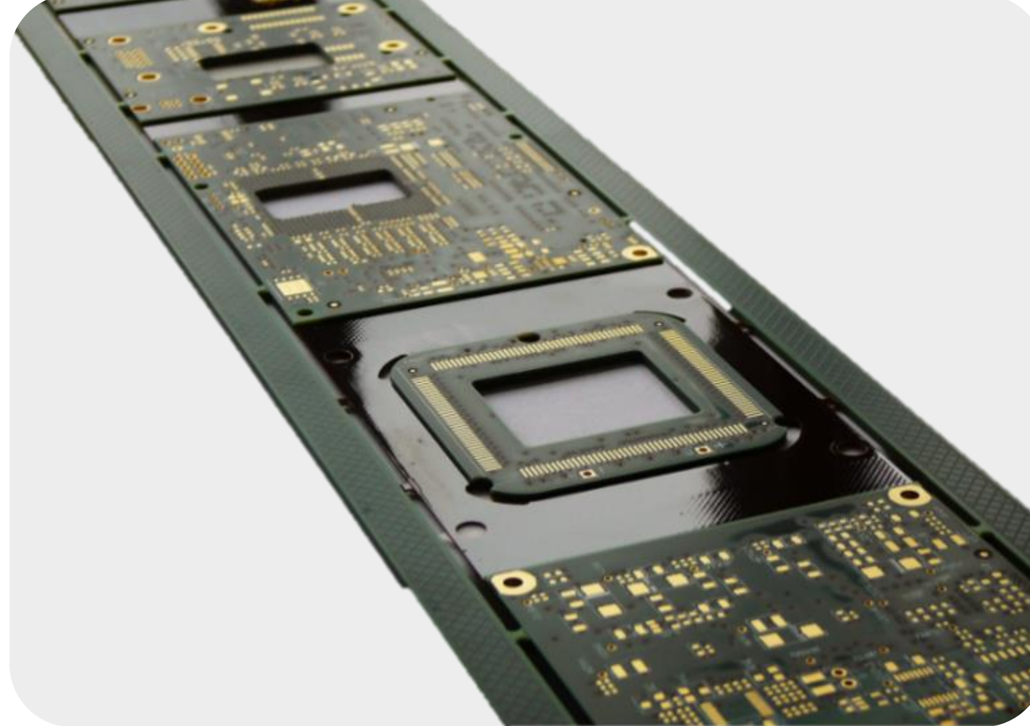
APPLICATION EXAMPLE

High resolution high speed camera

Application:
Astronomy, area scanning and spectroscopy

- Stackup RIGID.flex 4Ri-4F-4Ri (12 layers)
- Impedance defined signals
- 50µm Polyimide core , FR-4.1 TG 150°
- All flex layers are bonded in the flex area.



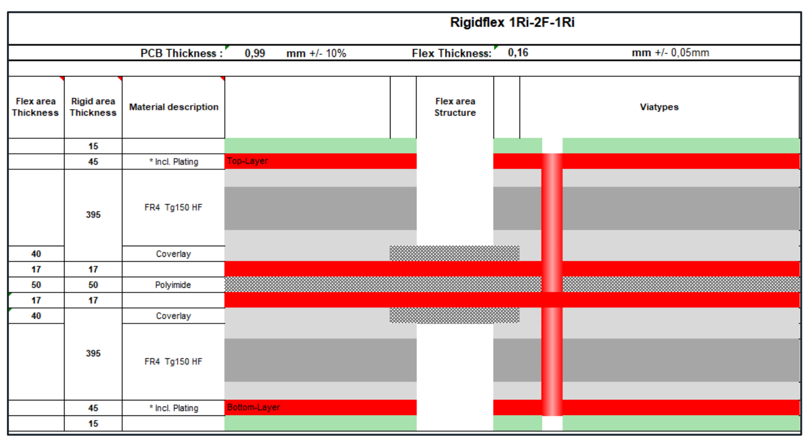


Source: ANDOR

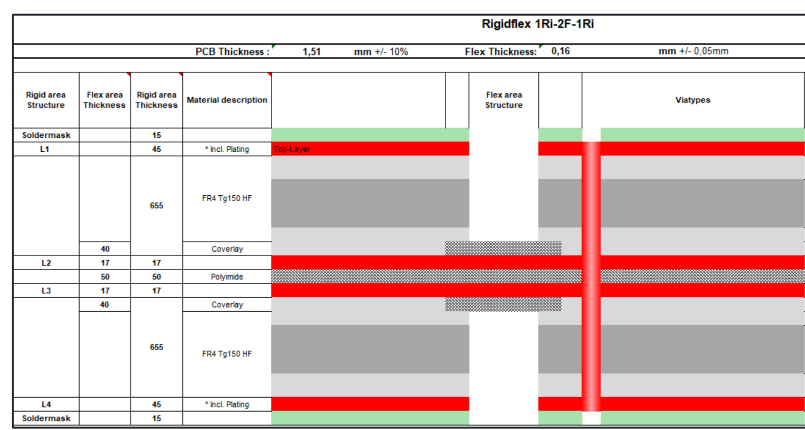
WE CIRCUIT BOARD TECHNOLOGY STANDARD STACKUPS

RIGID.flex xRi-2F(4F)-xRi

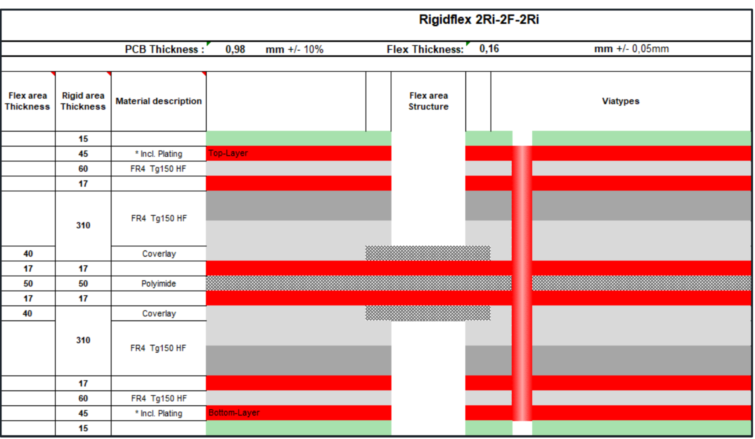
Standard 1Ri-2F-1Ri Thickness 1.00 mm



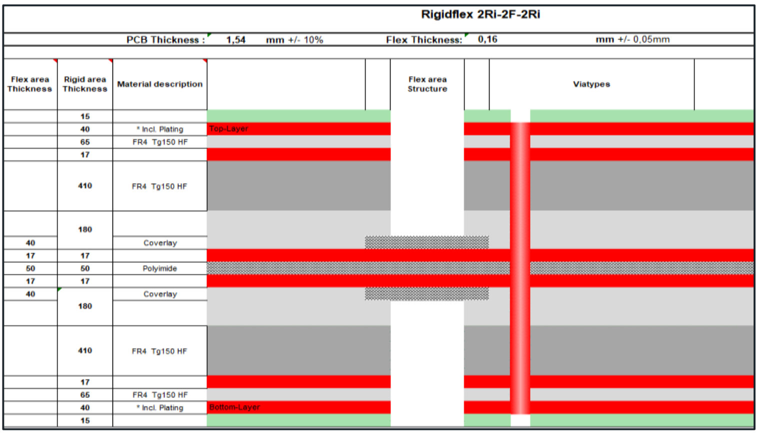
Standard 1Ri-2F-1Ri Thickness 1.55 mm



Standard 2Ri-2F-2Ri Thickness 1.00 mm



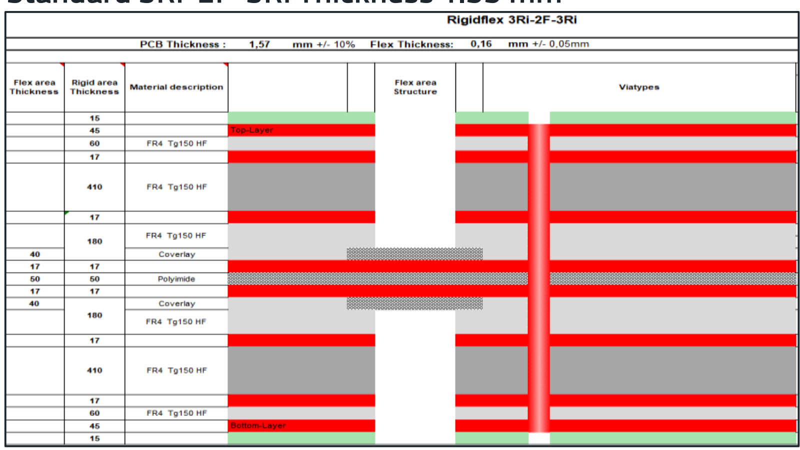
Standard 2Ri-2F-2Ri Thickness 1.55 mm



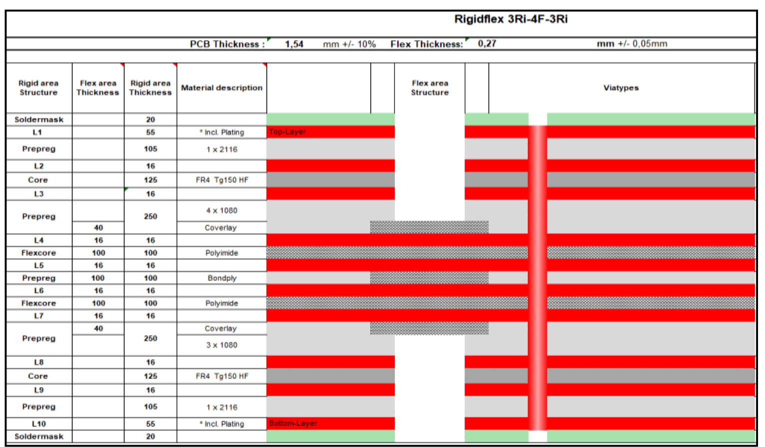
WE CIRCUIT BOARD TECHNOLOGY STANDARD STACKUPS

RIGID.flex xRi-2F(4F)-xRi

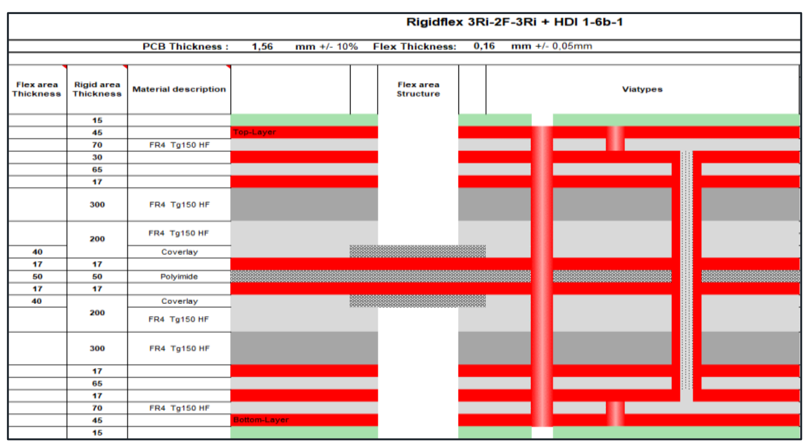
Standard 3Ri-2F-3Ri Thickness 1.55 mm



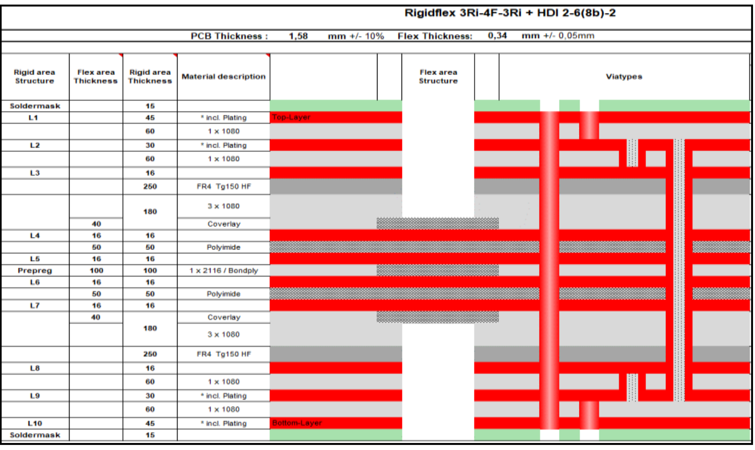
Standard 3Ri-4F-3Ri Thickness 1.55 mm



Standard 3Ri-2F-3Ri Thickness 1.55 mm HDI 1-6b-1

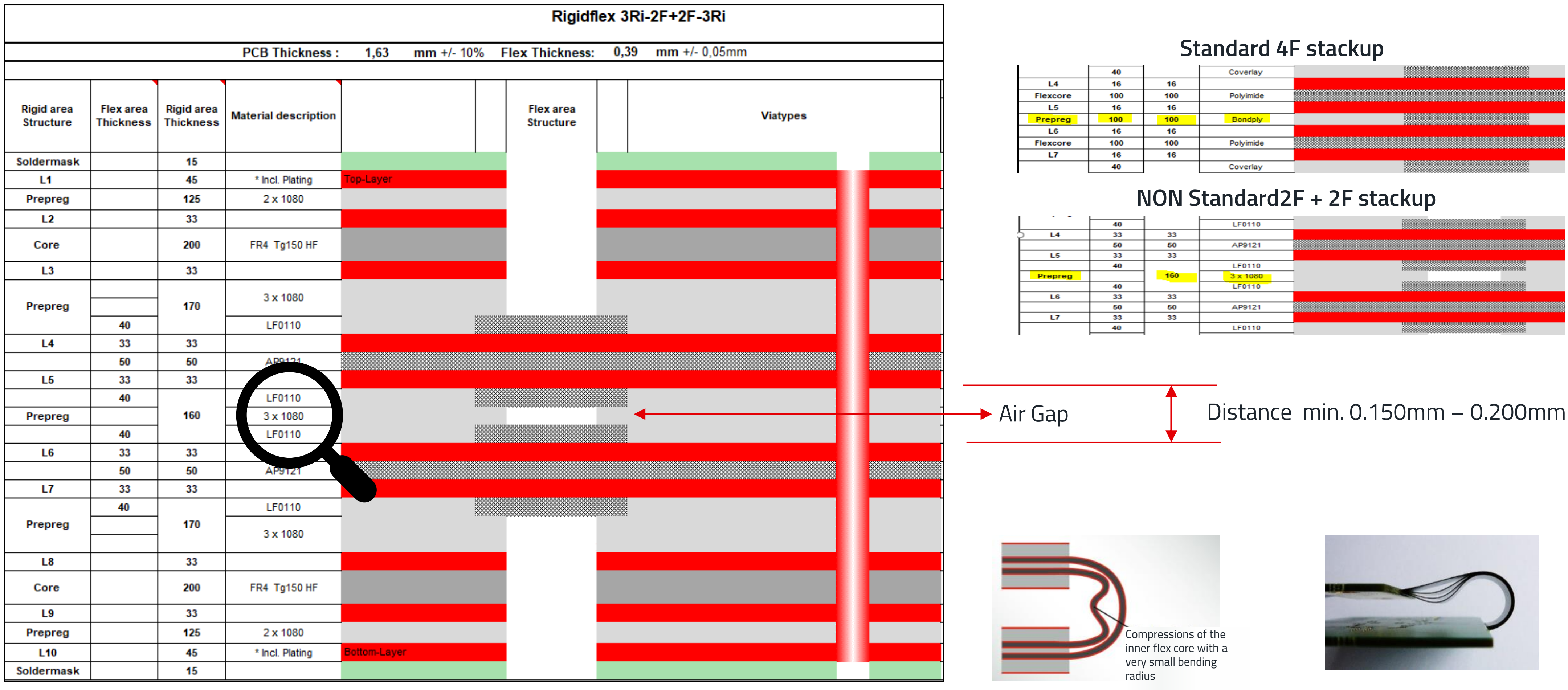


Standard 3Ri-4F-3Ri Thickness 1.55 mm HDI 2-6(8b)-2



WE CIRCUIT BOARD TECHNOLOGY NON STANDARD STACKUPS

RIGID.flex 3Ri-2F+2F-3Ri



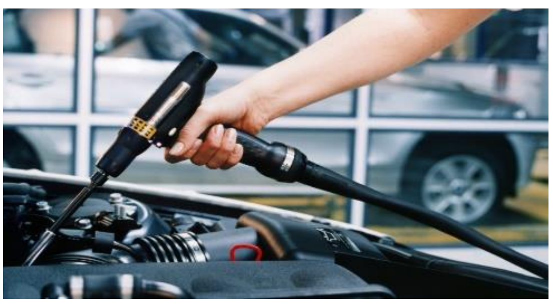
Rigidflex 3Ri-2F+2F-3Ri

PCB Thickness : 1,63 mm +/- 10% Flex Thickness: 0,39 mm +/- 0,05mm

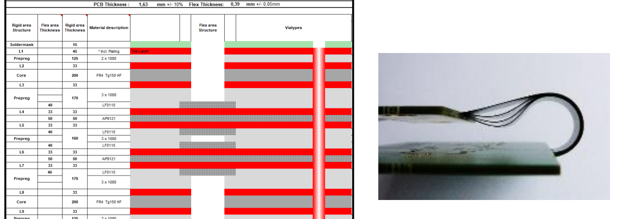
Rigid area Structure	Flex area Thickness	Rigid area Thickness	Material description
Soldermask		15	
L1		45	* Incl. Plating
Prepreg		125	2 x 1080
L2		33	
Core		200	FR4 Tg150 HF
L3		33	
Prepreg		170	3 x 1080
	40		LF0110
L4	33	33	
	50	50	AP9121
L5	33	33	
	40		LF0110
Prepreg		160	3 x 1080
	40		LF0110
L6	33	33	
	50	50	AP9121
L7	33	33	
	40		LF0110
Prepreg		170	3 x 1080
L8		33	
Core		200	FR4 Tg150 HF
L9		33	
Prepreg		125	2 x 1080
L10		45	* Incl. Plating
Soldermask		15	

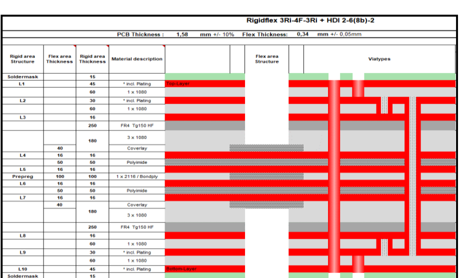
SUMMARY STACKUPS

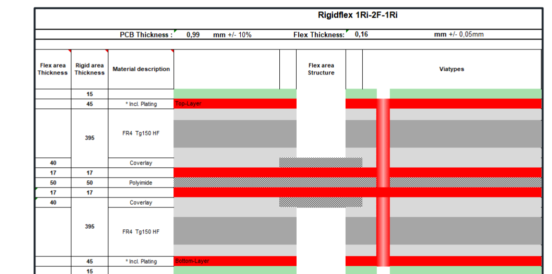
- In the webinar **PCB production part 2** you will find important information about asymmetrical stack-ups as well as a description of the basic materials required.

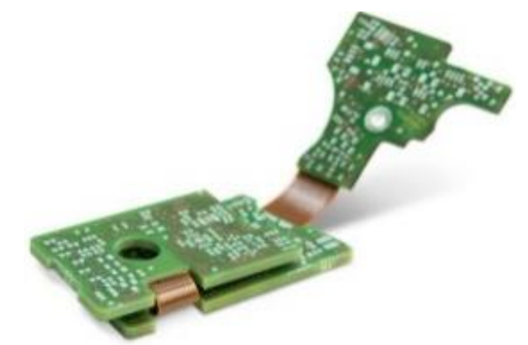


- Symmetrical **RIGID.flex** PCBs are used for industrial tools up to ultra-modern high-speed cameras.
- Standard stackups of **xRi-2F(4F)-xRi** variants were presented
- NON Standard using the example of a **3Ri-2F+2F-3Ri stackup** with air gap









PCB PRODUCTION, PART 3: RIGID.FLEX PCBS, CONTINUED

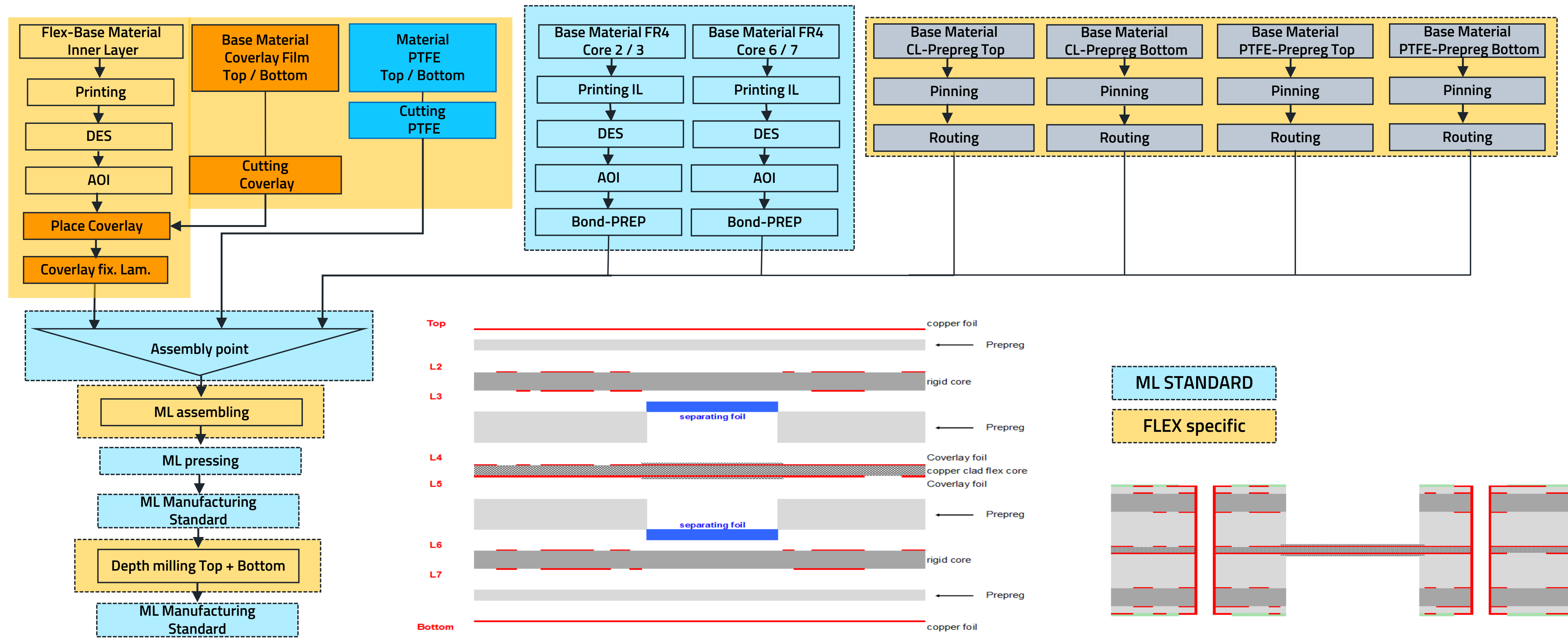
POLL: Multiple-Choice with one correct answer

What is the typical feature of a symmetrical layer stackup xRi-yF-xRi of a rigid-flex circuit board?

- Impedance defined signals are not possible
- They only consist of flexible base materials
- The flexible layers are always located outside
- The flexible area can only consist of 2 flex layers
- The flexible layers are located inside (center) of the PCB

PRODUCTION PROCESSES

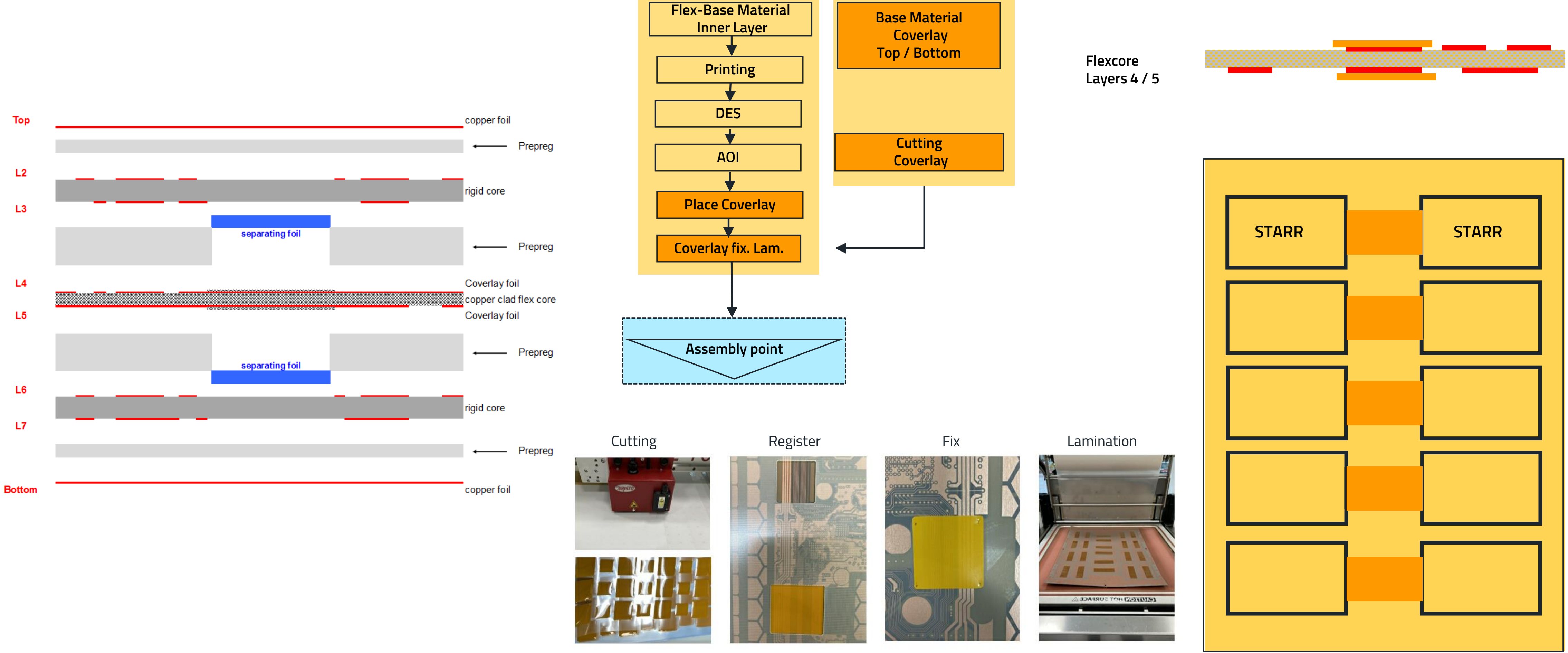
RIGID.flex 3Ri-2F-3Ri



WE

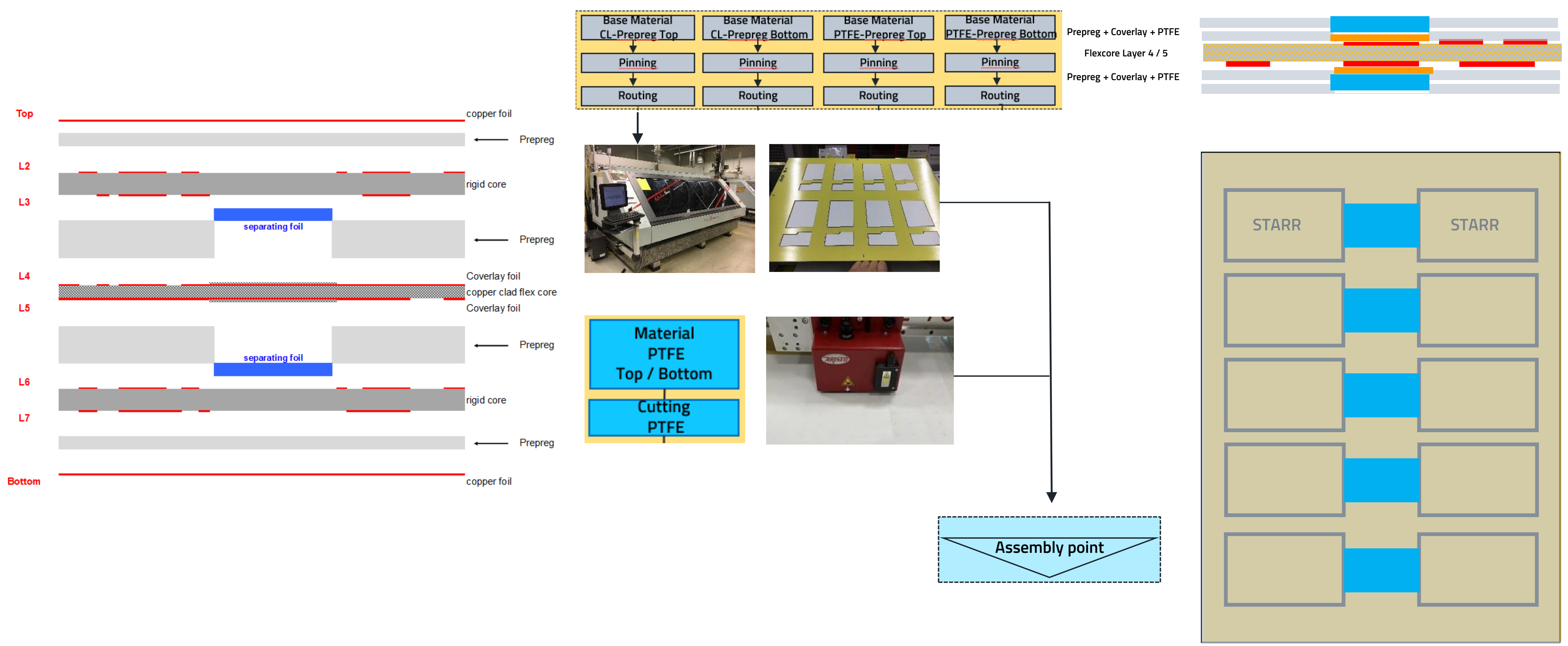
PRODUCTION PROCESSES

RIGID.flex 3Ri-2F-3Ri (flex inner layers 4 / 5)



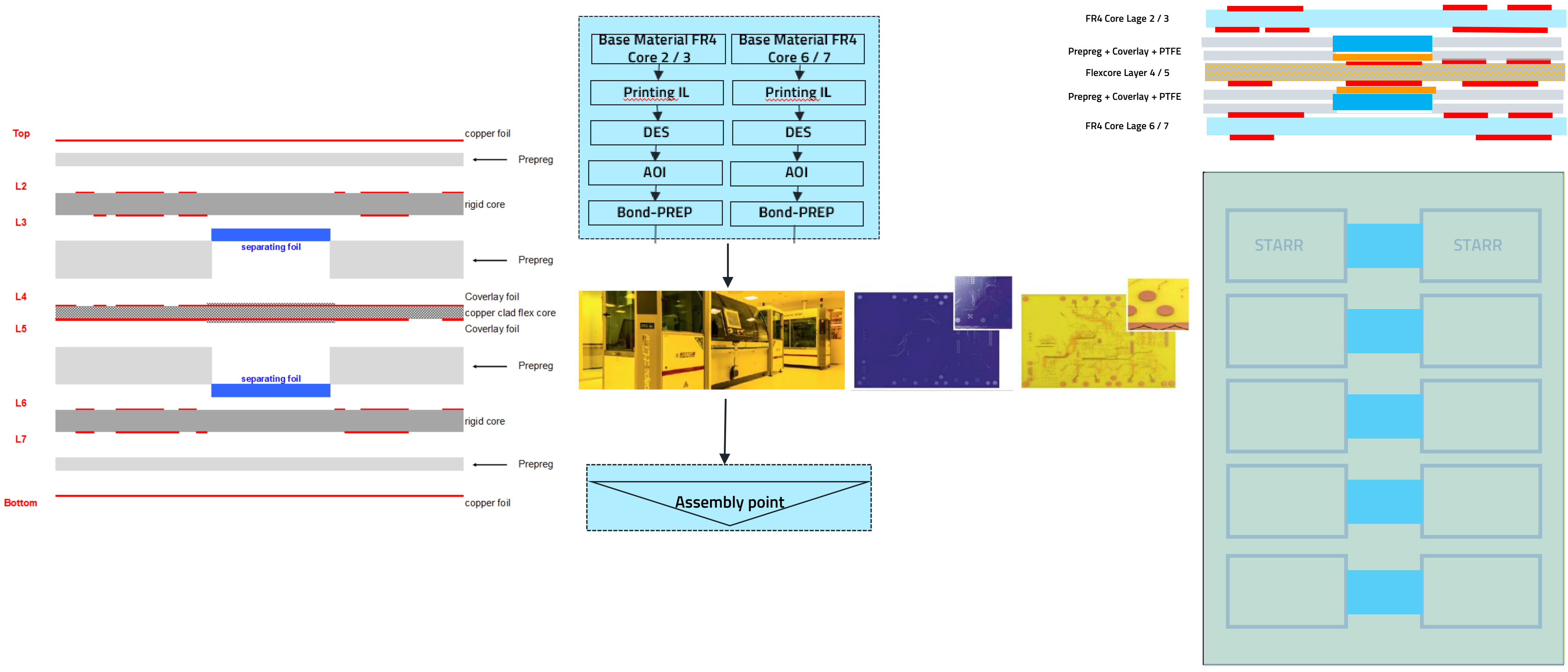
PRODUCTION PROCESSES

RIGID.flex 3Ri-2F-3Ri (Prepreg, PTFE foil)



PRODUCTION PROCESSES

RIGID.flex 3Ri-2F-3Ri (rigid inner layers 2 & 3 / 6 & 7)



PRODUCTION PROCESSES

RIGID.flex 3Ri-2F-3Ri(Prepreg und Copper foil)

Top
copper foil
Prepreg
L2
rigid core
L3
separating foil
Prepreg
L4
Coverlay foil
copper clad flex core
L5
Coverlay foil
Prepreg
separating foil
L6
rigid core
L7
Prepreg
Bottom
copper foil

Assembly point

ML assembling

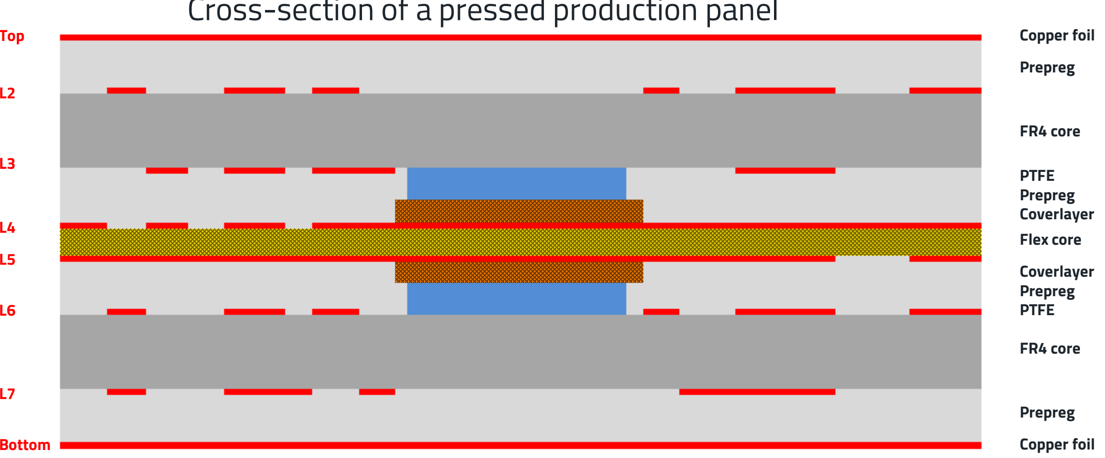
ML pressing

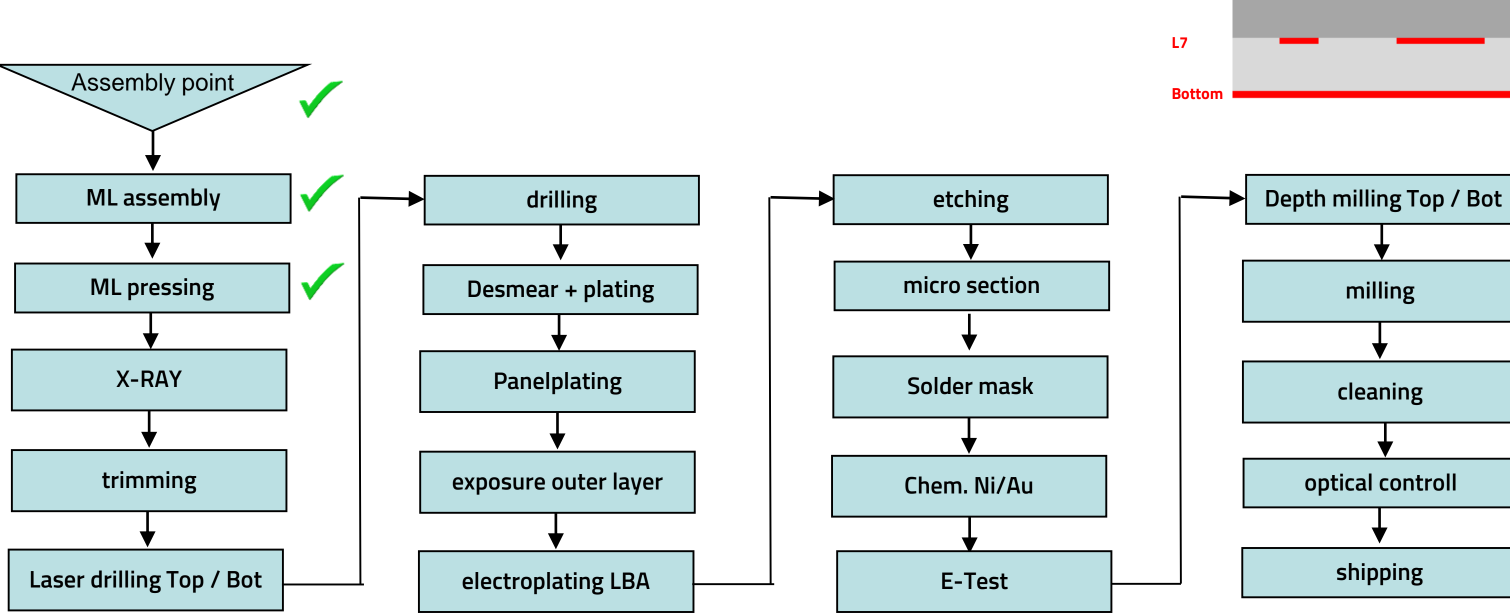
Prepreg + Copper-Foil
FR4 Core Layer 2 / 3
Prepreg + Coverlay + PTFE
Flexcore Layer 4 / 5
Prepreg + Coverlay + PTFE
FR4 Core Layer 6 / 7
Prepreg + Copper-Foil

PRODUCTION PROCESSES

RIGID.flex 3Ri-2F-3Ri

- After pressing the rigid-flexible PCB, all the usual processes of a BASIC multilayer PCB are applied.





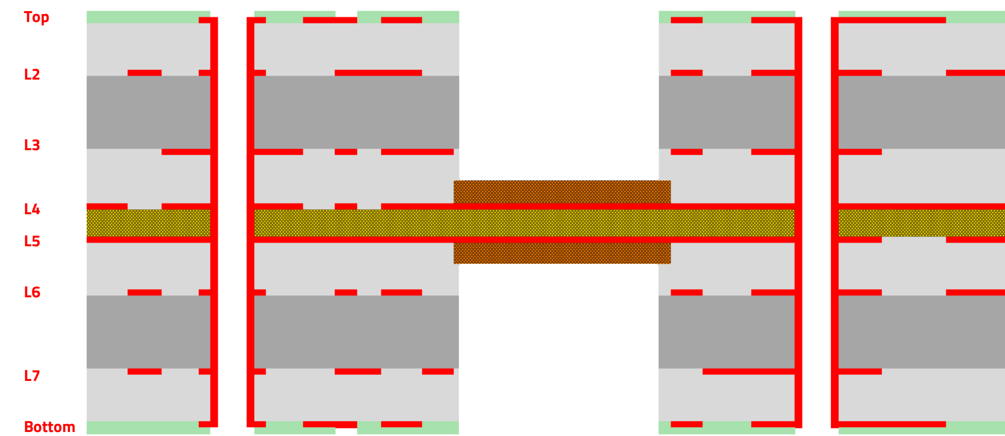
PRODUCTION PROCESSES

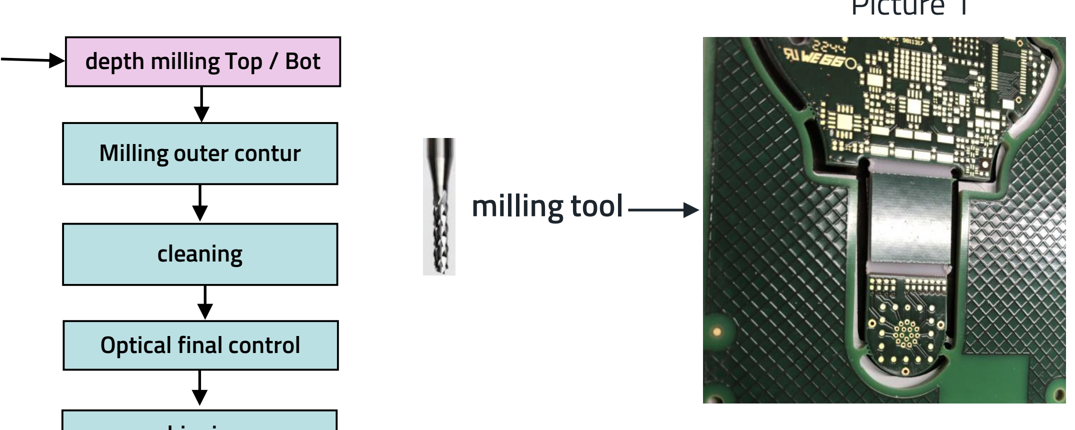
RIGID.flex 3Ri-2F-3Ri (depth milling)

Two process steps are necessary for the **"depth milling"**

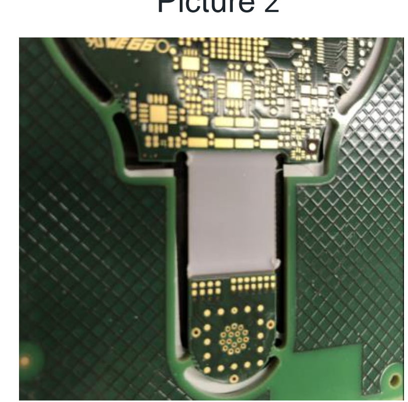
- Depth milling from the top side
- Depth milling from the bottom side

The depth of the milling tool reaches the PTFE foil (separating foil). The PTFE foil is removed in the process.

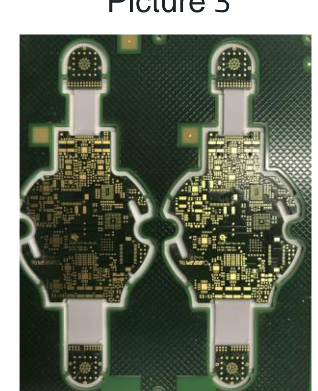




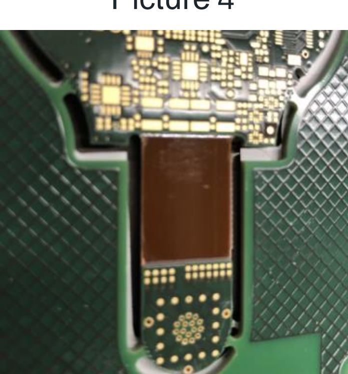
Picture 2



Picture 3



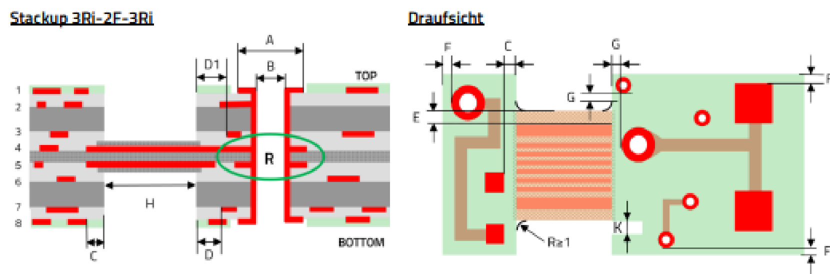
Picture 4



DESIGN RULES

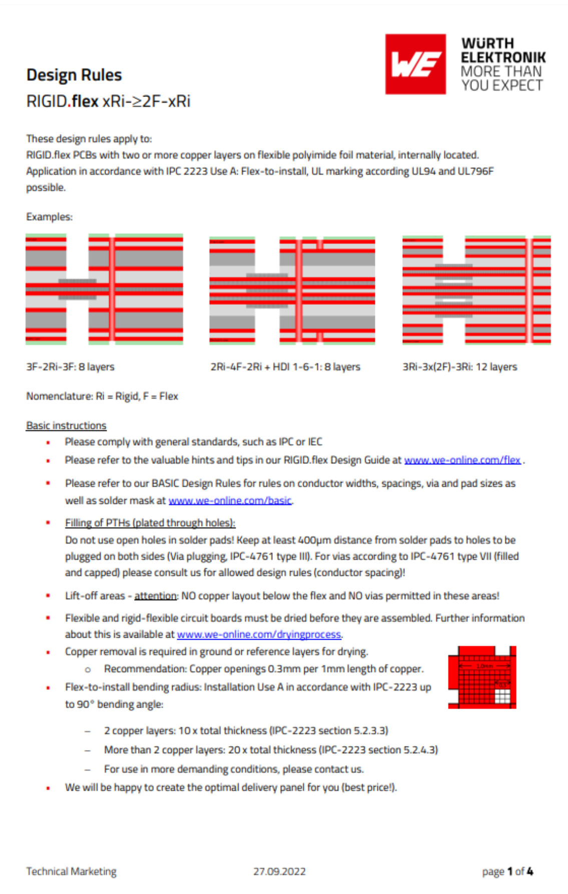
RIGID.flex xRi-≥2F-xRi

Now we have learned that the production of a rigid-flex circuit board differs fundamentally from a rigid BASIC PCB.
Especially the mechanical processing of the prepreg and the cover layer require **elementary design rules**.



Especially points **D** and **G** need a closer look.
For this, please visit our web page.

- Design Rules RIGID.flex xRi-2F-xRi
- Design Guide RIGID.flex



WÜRTH ELEKTRONIK MORE THAN YOU EXPECT

Design Rules

RIGID.**flex** xRi-≥2F-xRi

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

Examples:

3F-2Ri-3F: 8 layers

2Ri-4F-2Ri + HDI 1-6-1: 8 layers

3Ri-3x(2F)-3Ri: 12 layers

Nomenclature: Ri = Rigid, F = Flex

Basic instructions

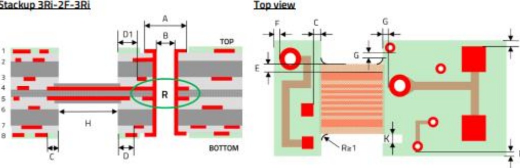
- Please comply with general standards, such as IPC or IEC
- Please refer to the valuable hints and tips in our RIGID.flex Design Guide at www.we-online.com/flex .
- Please refer to our BASIC Design Rules for rules on conductor widths, spacings, via and pad sizes as well as solder mask at www.we-online.com/basic.
- Filling of PTHs (plated through holes):
  Do not use open holes in solder pads! Keep at least 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)!
- 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 0.3mm per 1mm 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)
  - More than 2 copper layers: 20 x total thickness (IPC-2223 section 5.2.4.3)
  - For use in more demanding conditions, please contact us.
- We will be happy to create the optimal delivery panel for you (best price!).

Technical Marketing 27.09.2022 page **1** of **4**

WÜRTH ELEKTRONIK MORE THAN YOU EXPECT

Design Rules

RIGID.**flex** xRi-≥2F-xRi

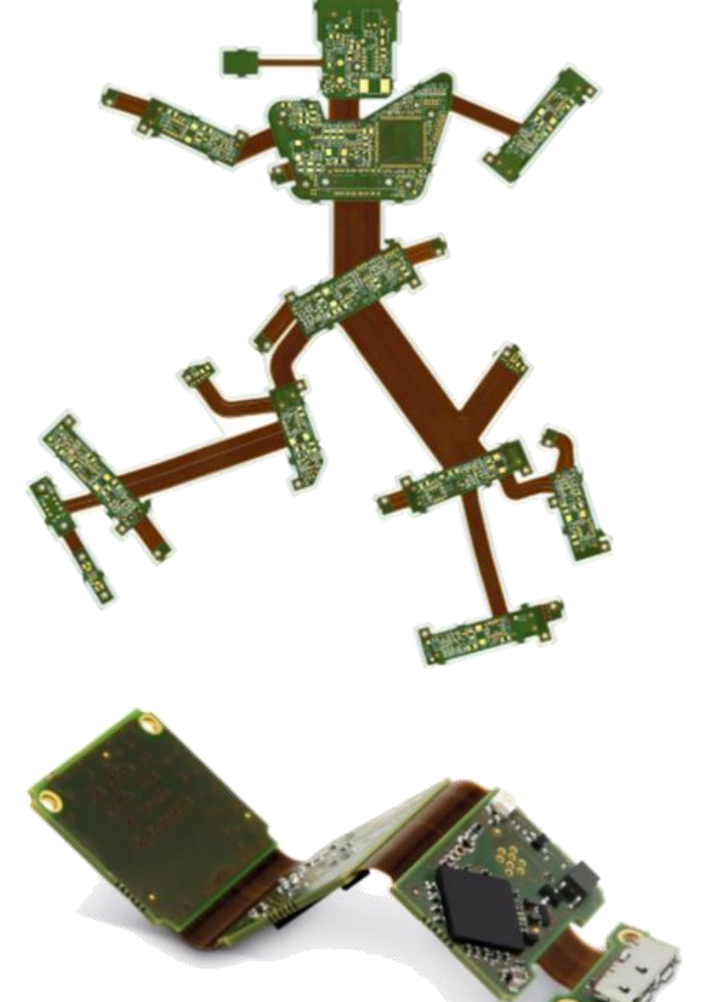


Symbol	Description	Technical Standard	Advanced requirements
	Line widths and spacings	see WE BASIC Design Rules!	
A	Minimum via pad diameter **→For all Pad-connections Teardrops are recommended!**	see WE BASIC Design Rules!	
B	Final diameter of PTH	see WE BASIC Design Rules!	
R	**→ NFP: Non functional / non-used pads do NOT remove!!**		
C	Spacing, Cu – outer layer to flex-rigid transition (bottom)	≥300µm	
D	Spacing, Cu – inner layer to flex-rigid transition	≥500µm	
D1	Spacing, Cu – to flex-rigid transition: inner layers next to flex	≥1000µm	
E	Distance of conductor to the flexible contour	≥300µm	
F	Spacing, exposed Cu – outside of flex-rigid transition	≥300µm	
G	**Flexible area 2F**: Distance of via pad to flex-rigid transition	≥1500µm	≥1000µm
G	**Flexible area >2F**: Distance of via pad to flex-rigid transition	≥2000µm	≥1500µm
G	*For your information: Recommendation in IPC-2223D 5.2.2.3:*	*3.18 mm+ ½ pad diameter*	
H	Length of the flex area with 2F (for > 2F please contact us)	≥5mm	≥2.5mm
K	Minimum recess width directly at the flex area	1.6mm	1.0mm
„K"	Outline manufacturing of flex area: **No scoring permitted!**		
„ZIF"	**ZIF contacts thickness tolerance** (material of stiffener)	±0.05mm (FR4)	±0.03mm (PI) special stackup
-	Combination with microvia (from 2Ri-yF-2Ri) and buried via technique is possible	see HDI Microvia Design Guide!	

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

Technical Marketing 27.09.2022 page **4** of **4**

PCB PRODUCTION, PART 3: RIGID.FLEX PCBS, CONTINUED



Thank You!