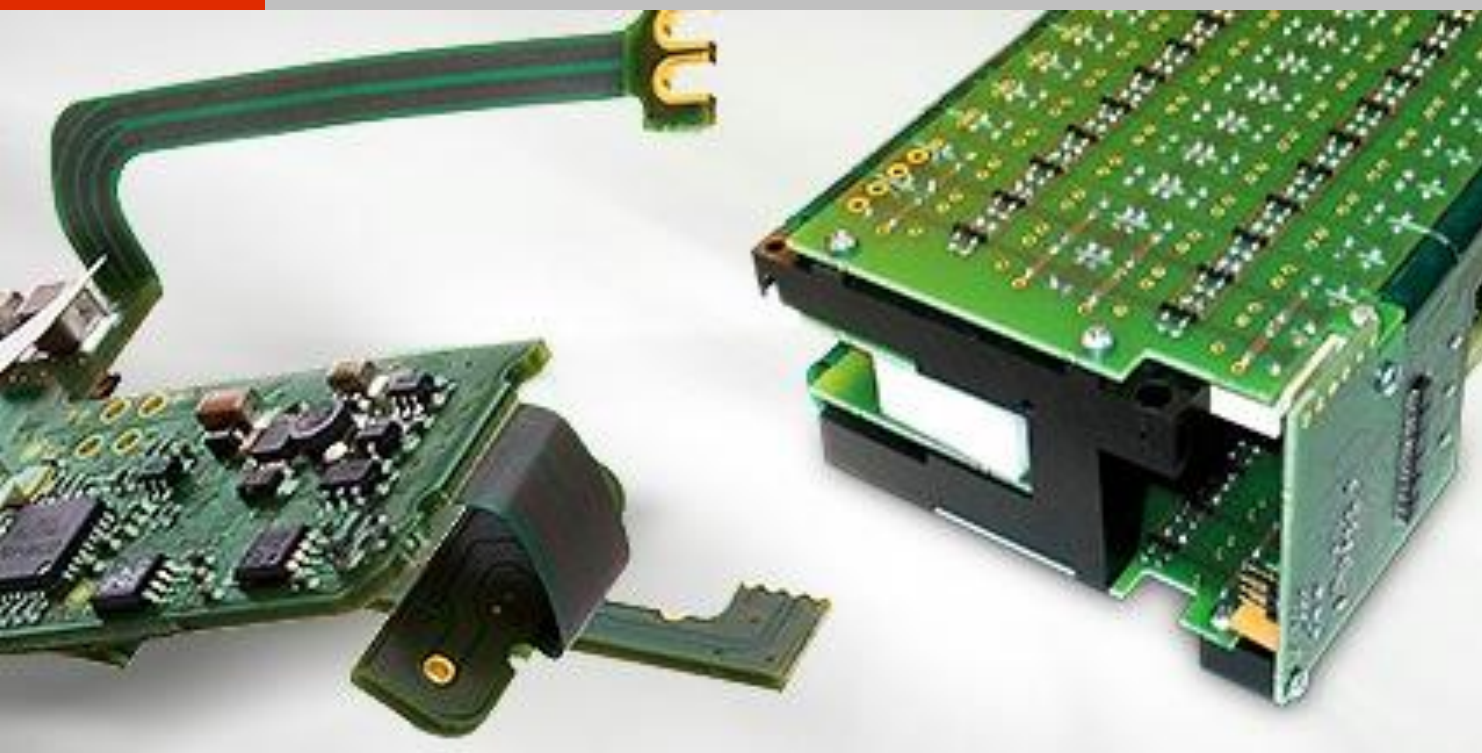


Flex-rigid: Design mistakes and their effects

Würth Elektronik Circuit Board Technology



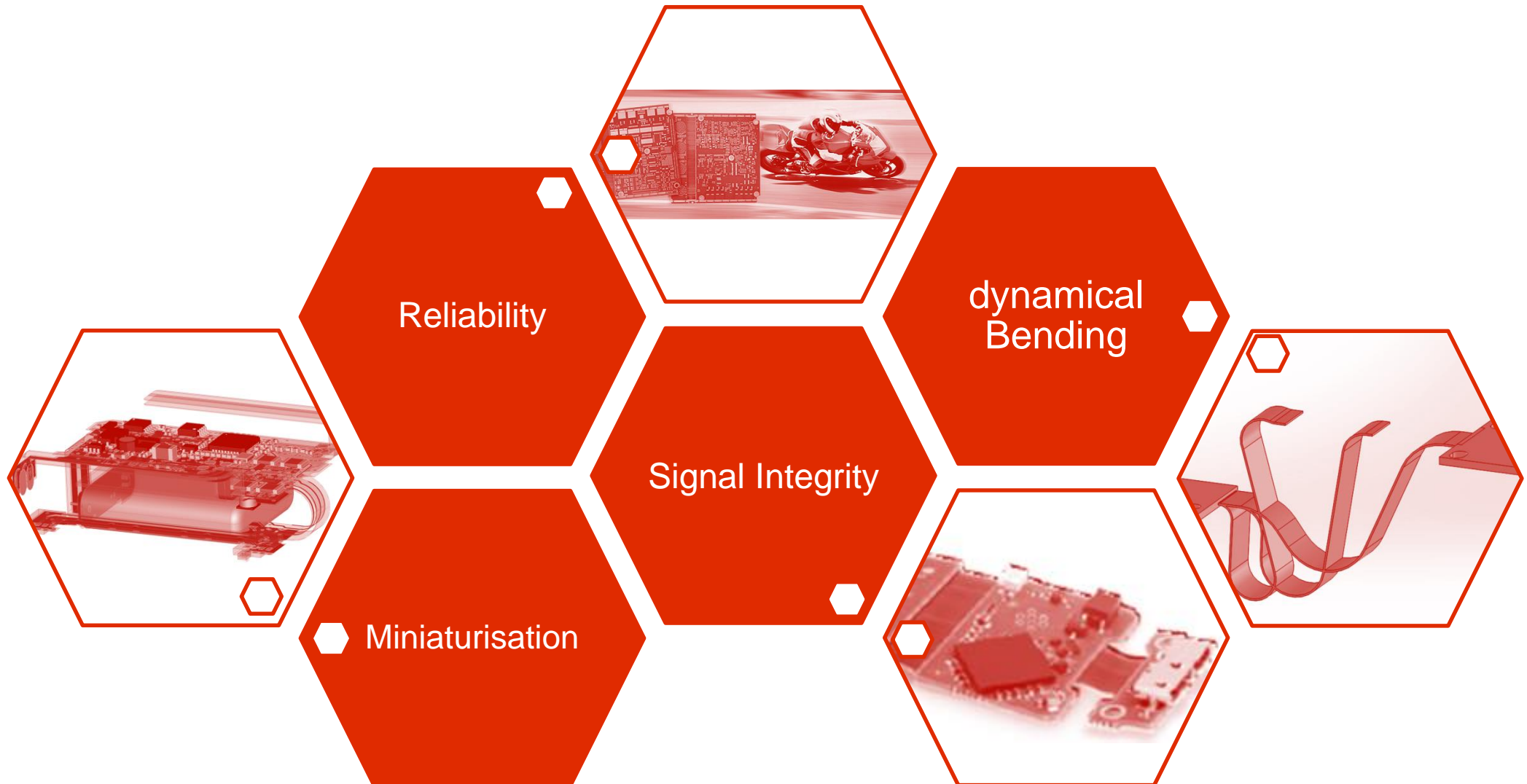
Webinar December 5, 2017
Speaker: Andreas Schilpp

Agenda

- 
- A vertical list of five items, each preceded by a white circle with a red outline. The circles are connected by a thin red line that starts at the top left and ends at the bottom left. The text for each item is white and centered within a red rectangular bar.
- Introduction Flex-Rigid
 - Design Standards, Design Rules
 - Examples, Consequences, Tips
 - Combination of Technologies
 - Summary, Q&A

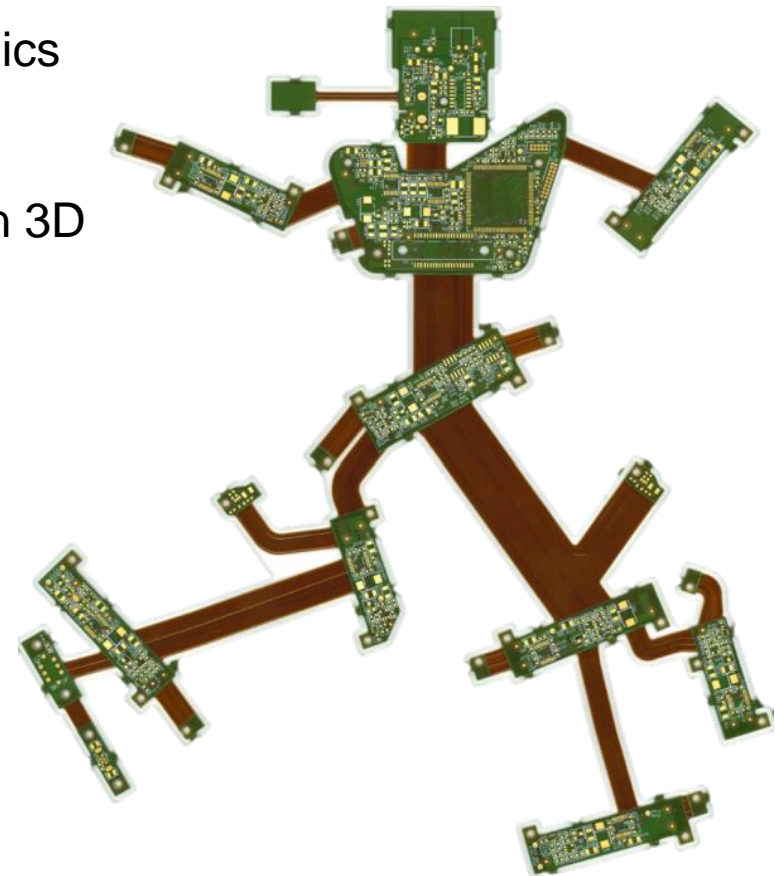
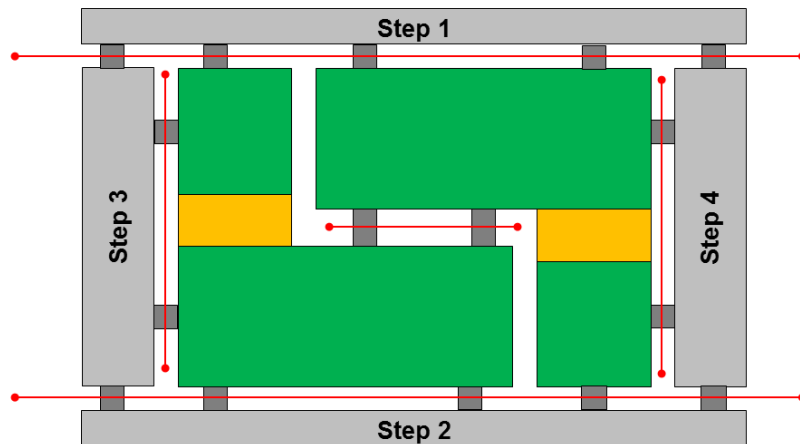
Introduction: Integration of Module Wiring

The Advantages of Flex-Rigid

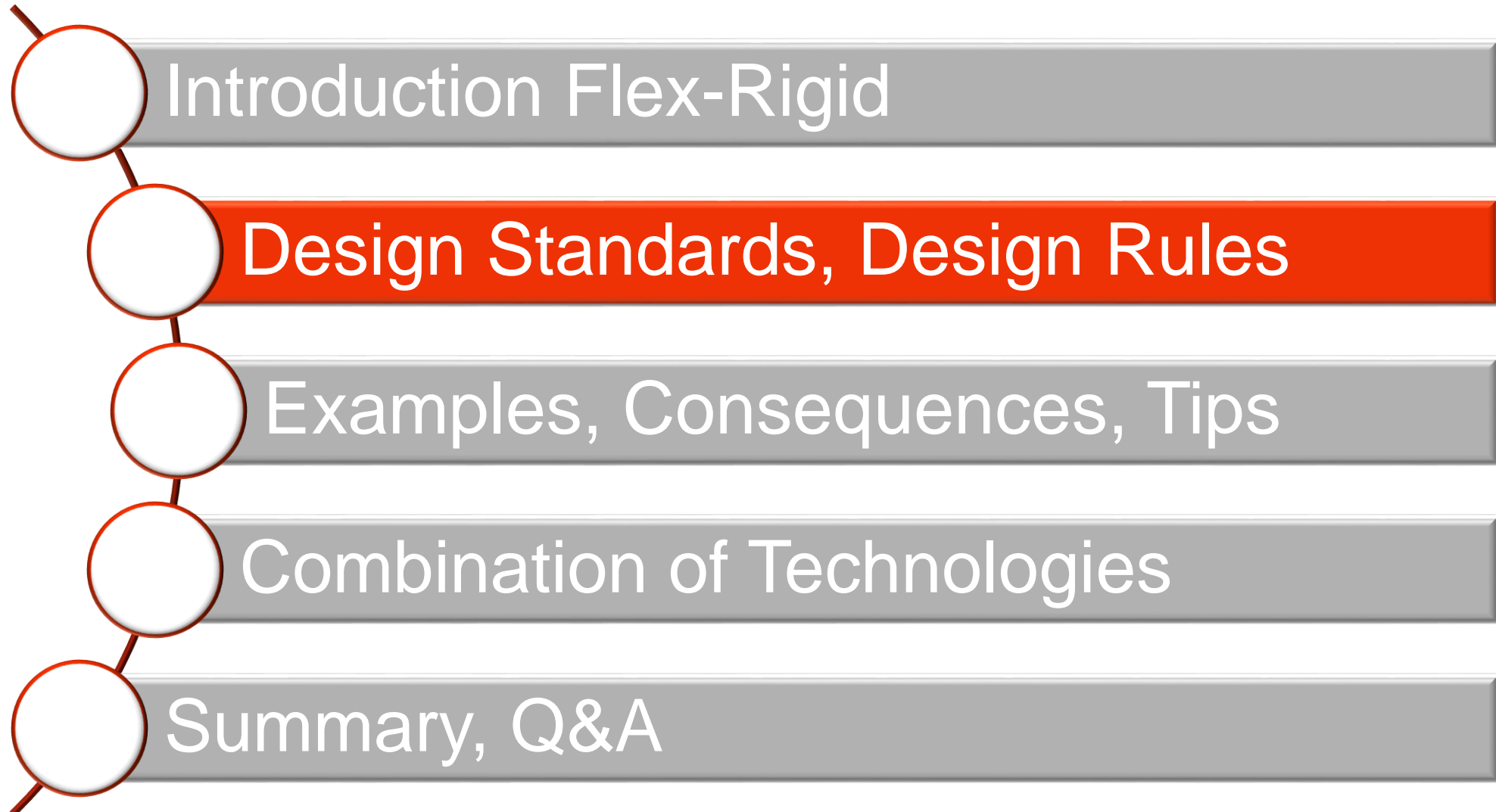


The Concept of Flex-Rigid: Mechatronics

- Rigid PCBs with integrated flexible wiring layers
- Principle:
 - rigid Moduls: Components and Routing, Mechanics
 - Flexible area: 3D – Wiring
 - Production, Test in 2D → Assembly and usage in 3D



Agenda



Design Standards

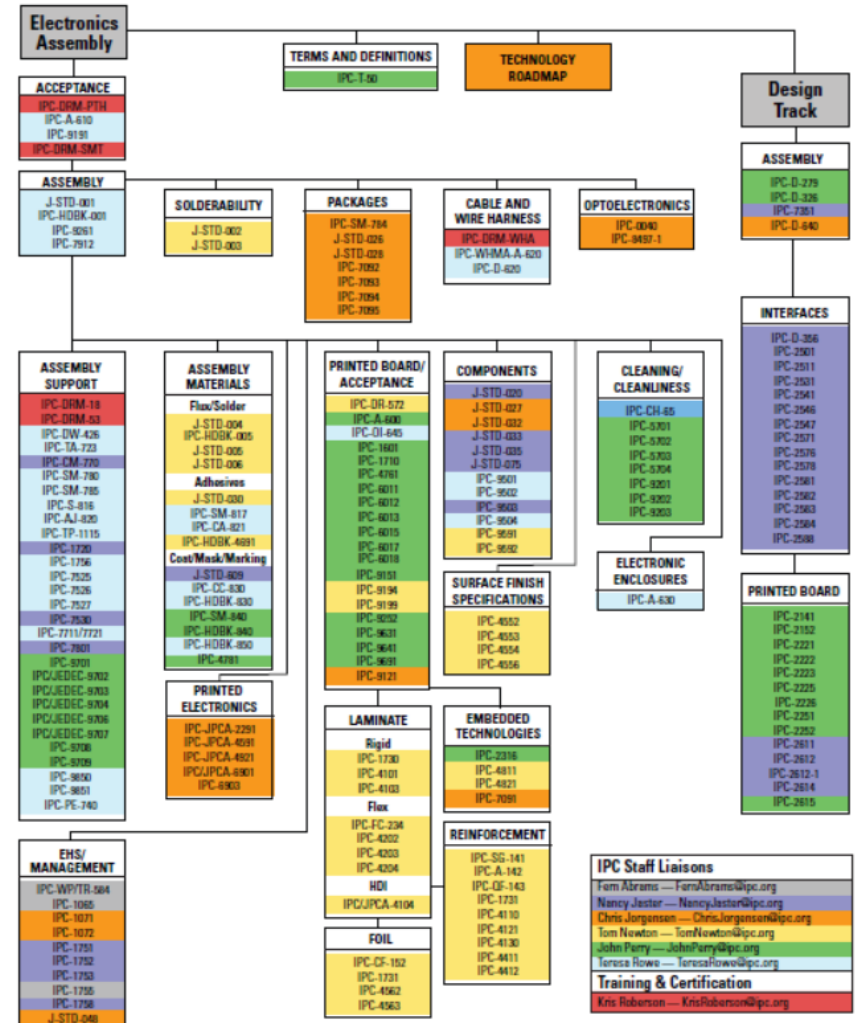
- **International Standards, i.e. IPC, EN**
 - **Technology specific (ML, HDI, Flex...)**
- **Application specific Standards, i.e. ATEX, UL**
 - **Security, i.e. Isolation, copper adhesion after aging, flammability**
- **PCB producer specific Design Rules**



IPC Standards

- Acceptability of Printed Boards: IPC-A-600
- Material
- Design: IPC-2223
- Qualification and Performance Spec: IPC-6013
-

IPC. IPC Standards Tree



IPC-A-600H

- **Description Transition Zone: +/- 1,5mm**

4.1 FLEXIBLE AND RIGID-FLEX PRINTED BOARDS

4.1.6 Transition Zone, Rigid Area to Flexible Area

The transition zone is the area centered at the edge of the rigid portion from which the flexible portion extends. The inspection range is limited to 3.0 mm [0.12 in], about the center of the transition, which is the edge of the rigid portion.

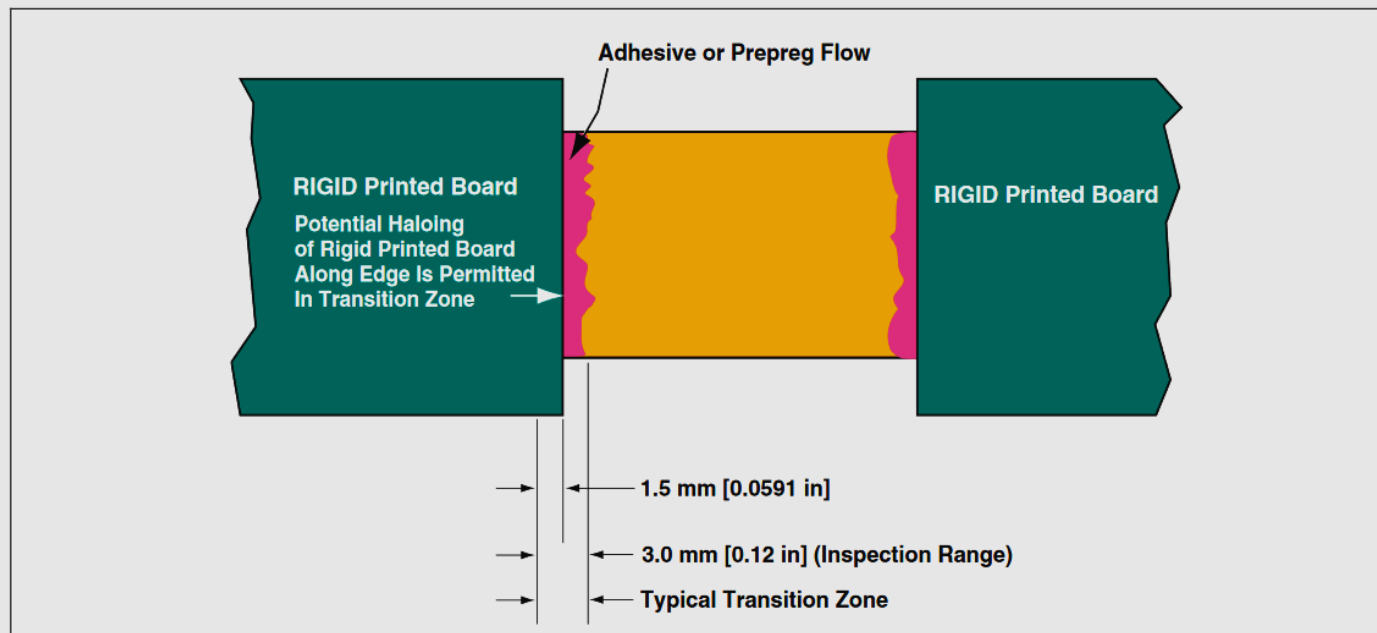


Figure 416a

IPC-6013C - Definition

December 2013

IPC-6013C

3.3.1.3 Transition Zone, Rigid Area to Flexible Area The transition zone is the area centered on the edge of the rigid portion from which the flex portion extends. The inspection range is limited to 3.0 mm [0.118 in], centered on the transition, which is the edge of the rigid portion (see Figure 3-1). Visual imperfections inherent to the fabrication technique (i.e., adhesive squeeze-out, localized deformation of dielectric and conductors, protruding dielectric materials, crazing, or haloing) **shall not** be cause for rejection. Imperfections in excess of that allowed **shall** be AABUS, or as so stated on the procurement documentation.

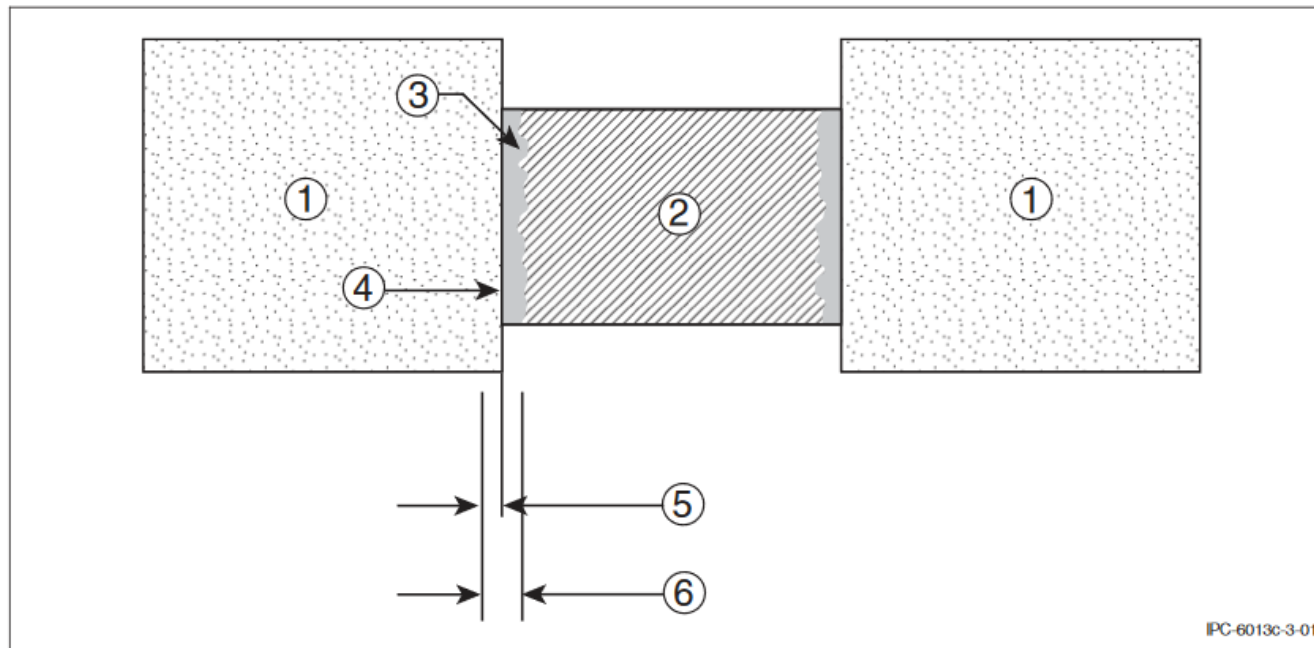


Figure 3-1 Transition Zone

Note 1. Rigid Printed Board

Note 2. Flex Circuit

Note 3. Adhesive or Prepreg Flow

Note 4. Potential Haloing of Rigid Printed Board along Edge is permitted in Transition Zone

Note 5. 1.5 mm [0.060 in]

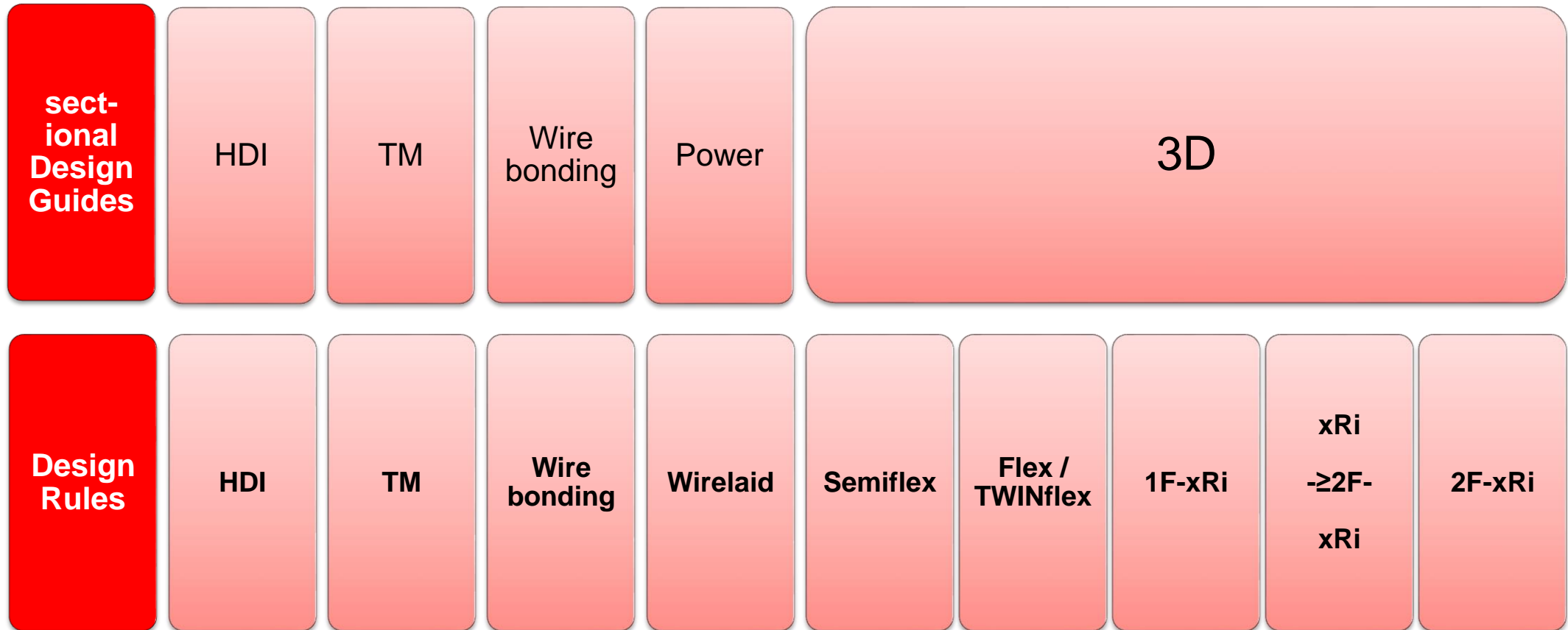
Note 6. Limit of Inspection Range (3.0 mm [0.118 in]), centered on the Transition Zone.

Grey Area



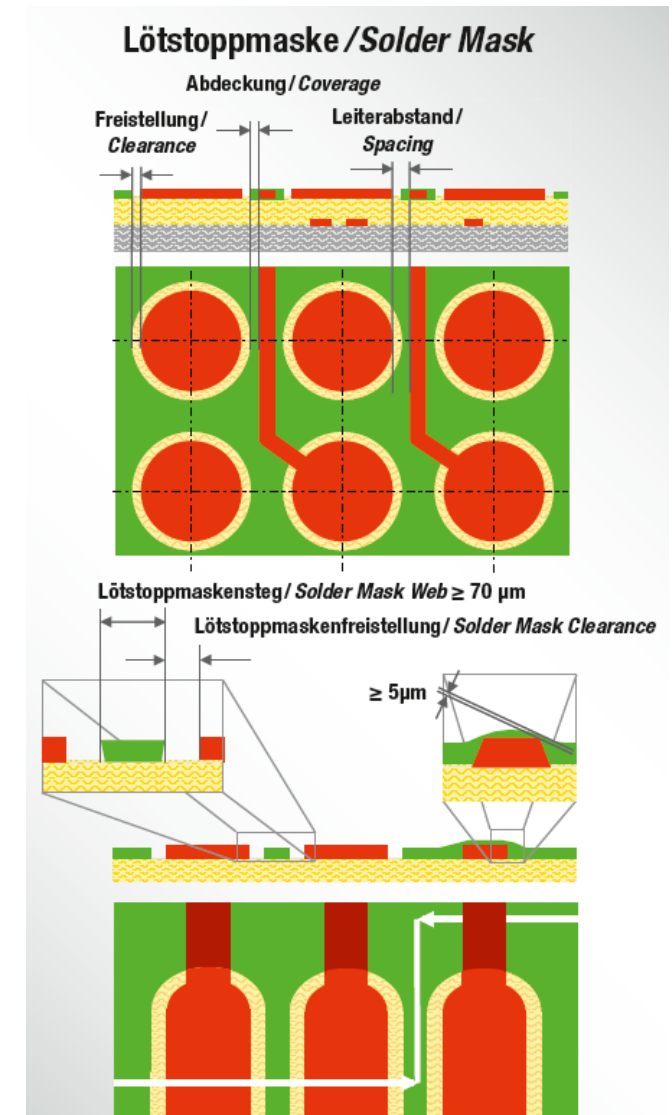
Hierarchy WE Design Guides / Design Rules

Basic Design Guide



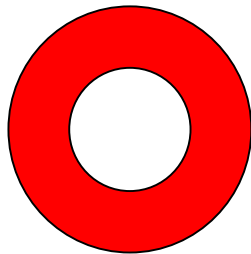
Basic Design Guide

- **basic = also valid for HDI, 3D ...**
- **partially extended in „sectional Design Guides“, i.e. HDI**
- **covers Basic issues as**
 - PTH and PTH pad design
 - Lines and space values
 - Soldermask Web and Clearance
 - Legend Print
 - Copper Clearance to
 - milled / V-scored contour
 - NPTH

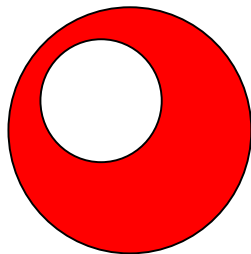


Sense of Annular Rings

Layout / Screen:



Real life

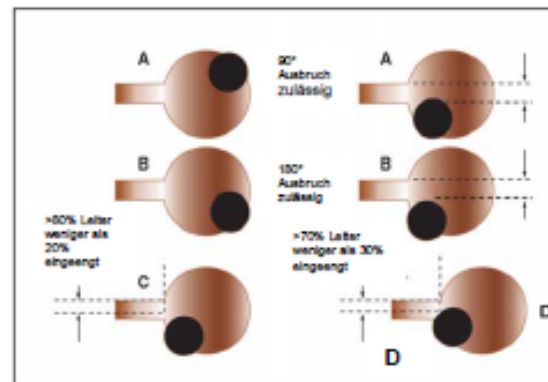


IPC-A-600H:

2.10.3 Äußerer Restring – metallisierte Löcher

2.10 Definition der Leiterbilder - Abmessungen

2.10.4 Äußerer Restring – metallisierte Löcher (Fortsetzung)



Zulässig – Klasse 2

- Ausbruch von maximal 90° (A).
- Wenn der Ausbruch an der Verbindungsstelle der Leiterbahn zur Anschlussstelle auftritt, darf die Leiterbahn um maximal 20% der Mindestleiterbreite reduziert sein, die in der technischen Zeichnung oder dem Nennwert des Fertigungsmasters festgelegt ist. Die Verbindung der Leiterbahn sollte nicht weniger als 0,05 mm [0,0020 in] oder die Mindestleiterbreite sein (der kleinere Wert gilt.). (C)
- Minimaler Seitenabstand zwischen den Leitern ist eingehalten.

Zulässig – Klasse 1

- Ausbruch von maximal 180° (B).
- Wenn der Ausbruch an der Verbindungsstelle der Leiterbahn zur Anschlussstelle auftritt, darf die Leiterbahn um maximal 30% der Mindestleiterbreite reduziert sein, die in dem Fertigungsmaster festgelegt ist (D).
- Form, Sitz und Funktion sind nicht beeinträchtigt.
- Minimaler Seitenabstand zwischen den Leitern ist eingehalten.

Fehler – Klasse 1, 2, 3

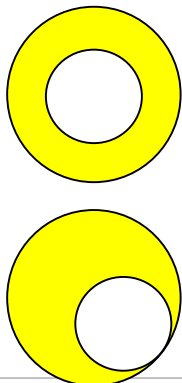
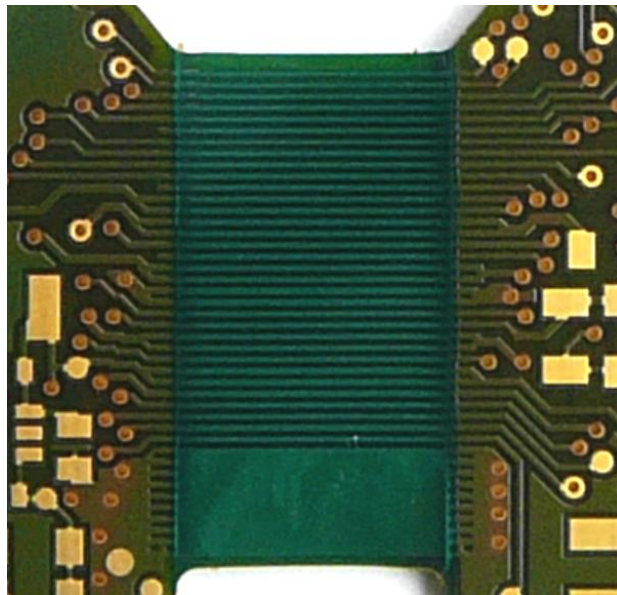
- Abweichungen, die gegen obige Kriterien verstoßen.

Design Rules Flex-Rigid – Spacing Vias / Pads

- Most common Mistake: Standard Rules Item „G“ not regarded

IPC-2223:

– Spacing Vias to Flex-Rigid Transition



Distance to the Via Pad!

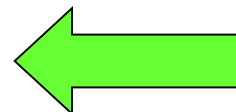


Figure 5-6 Spacing of Plated-Through Hole from Rigid to Flex Interface
Note 1. Minimum recommended spacing of 3.18 mm [0.125 in] plus one-half PTH pad diameter from the PTH center to the edge of the rigid material.



Structure of 3D Design Rules

- Application definition, stack-up examples
- Basic Information
- *Bending Radii*
- Processing Conditions
- Material Specificatons
- Link to Standard Stack-up's
- Definition Standard
- specific Layout Parameters

- additionally for Semiflex:
 - Tips for Bending and Fastening

more than you expect

Designregeln
Starrflex xRI – ≥ 2F – xRI
 Anwendung nach IPC 2223 Use A: Flex-to-install
 UL-Kennzeichnung nach UL94 oder UL796F möglich

Diese Designregeln gelten für:
starrflexible Leiterplatten mit ≥ 2 Kupferlagen auf Flexmaterial Polyimid Innen legend.

Beispiel 8-lagig: 3F-2F-3F Beispiel 8-lagig: 2F-4F-2F Beispiel 12-lagig: 3F-3x2F-3F

Nomenklatur: F = Flex, RI = Rigid (starr)

Grundlegende Hinweise

- Bitte beachten Sie allgemeine Standards wie IPC oder IEC
- Bitte beachten Sie die wertvollen Hinweise und Tipps im WE Starrflex Design Basic Design Guide!
- **Füllen von Bohrungen:**
Verwenden Sie keine offenen Bohrungen in Lötflächen! Halten Sie beidseitig von Lötflächen zu Bohrungen, die gesplaggt werden sollen (Durchschlagzucht nach IPC Typ VII (filled and capped)) bitte Rücksprache wegen erlaubter Distanz in unserem Internet *
- flexible Leiterplatten müssen vor dem Bestücken getrocknet werden. Warten Sie in unserem Internet *
- Für das Trocknen sind Kupferöffnungen in Masse- bzw. Referenzlagen nicht empfohlen! Kupferöffnungen: 0,2mm pro 1mm Kupferlänge

- Flex-to-install Diegrößen: Einbaubeanspruchung nach IPC-2223 bis 6,0mm
 - o 2 Kupferlagen: 10 x Gesamtdicke (IPC 2223 Punkt 5.2.4.2)
 - o Mehr als 2 Kupferlagen: 20 x Gesamtdicke (IPC-2223 Punkt 5.2.4.2)
 - o bei anspruchsvolleren Einsatzbedingungen bitten wir um Rücksprache
- Gerne erstellen wir für Sie einen optimalen Liefernutzen (best price)!

* sämtliche Unterlagen finden Sie online unter: www.we-online.de

Stand 27.01.2017 AS 1 / 3

more than you expect

Designregeln
Starrflex xRI – ≥ 2F – xRI
 Anwendung nach IPC 2223 Use A: Flex-to-install
 UL-Kennzeichnung nach UL94 oder UL796F möglich

Materialspezifikationen

Material	Standard	Spez. Blatt	Beschreibung	Anwendung
Isolierblech	IPC-4204	11	Polyimid Kleberlos	Standard
Basismaterial	IPC-4204	2	Polyimid Kleberhaftig	Standard
Stammmaterial (Kerne und Prepreg)	IPC4101	128 (92,94,127)	FR4 Tg150, getübt, halogenfrei, low CTE(z)	Standard
Lötstopplack	IPC-6160		grün, photosensitiv	Standard
Coverlay / Bondply	IPC-4303	1 / 2	Polyimid Deck- bzw. Verbundfolien, Acryl- oder Epoxy-Kleber	Standard (partiell genannt)

Lagenaufbau
 Standard Lagenaufbau siehe www.we-online.de/flex

Standardausführung

1. Polyimid 50µm kleberlos, ED-Kupfer, LP Gesamtdicke 1,0mm bis 1,55mm
2. Kupferschichtdicke Innenlagen 18µm, Außenlagen 12µm + galvanische Verstärkung
3. Partielle Coverlaytechnik (auch „Blinker“ genannt)
4. photosensitiver Lötstopplack grün
5. Standard Durchkontaktierungen
6. Kleinstster Fräseerdurchmesser 1,6mm
7. Lötberfläche chem. Ni/Au
8. Verpackung in ESD-Schrumpfolie

Kombination mit Microvia: (ab 6. Layer) und buried via - Technik möglich; siehe WE HDI

Stand 27.01.2017 AS 2 / 3

more than you expect

Designregeln
Starrflex xRI – ≥ 2F – xRI
 Anwendung nach IPC 2223 Use A: Flex-to-install
 UL-Kennzeichnung nach UL94 oder UL796F möglich

Querschnitt: 2F-2F-2F
 F = Flex, RI = rigid (starr)

Draufleisch xRI-2F-xRI

wenn möglich - Einzelfschutz in Form von breiten Kupferbahnen anbringen
 Fluorimlagen mit PI Coverlay partiell

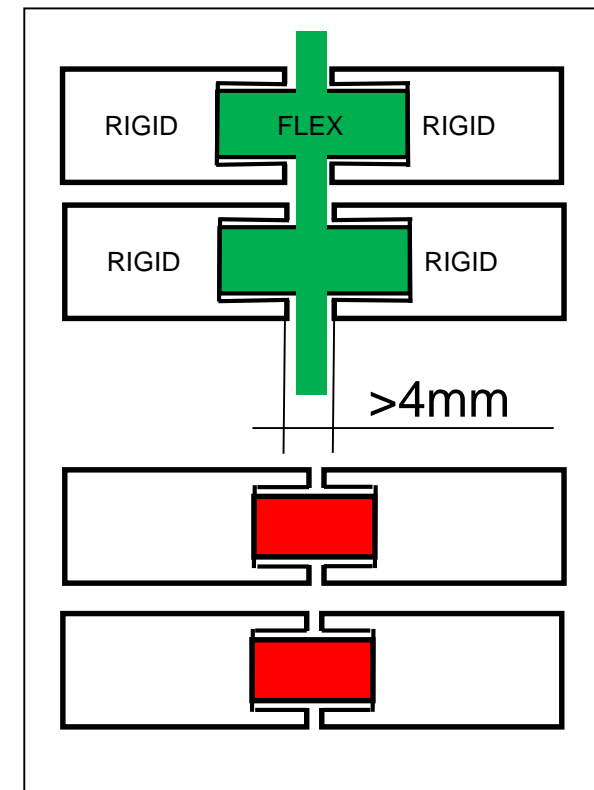
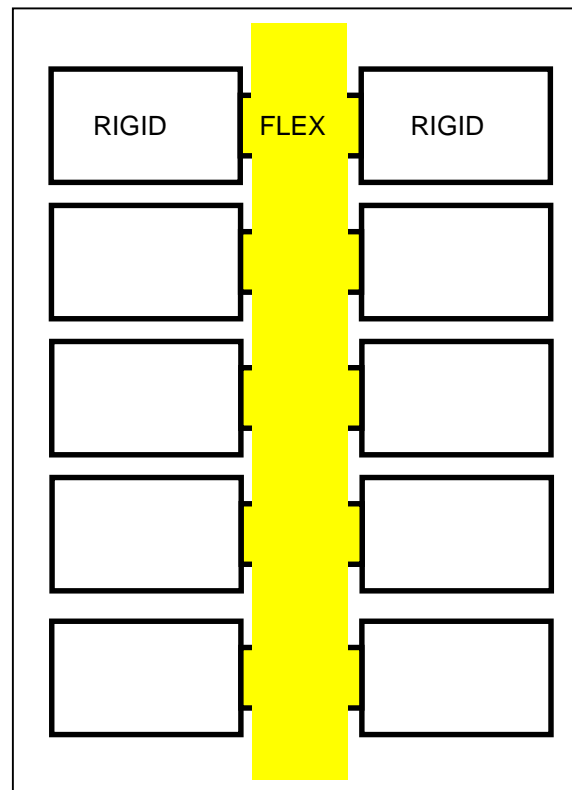
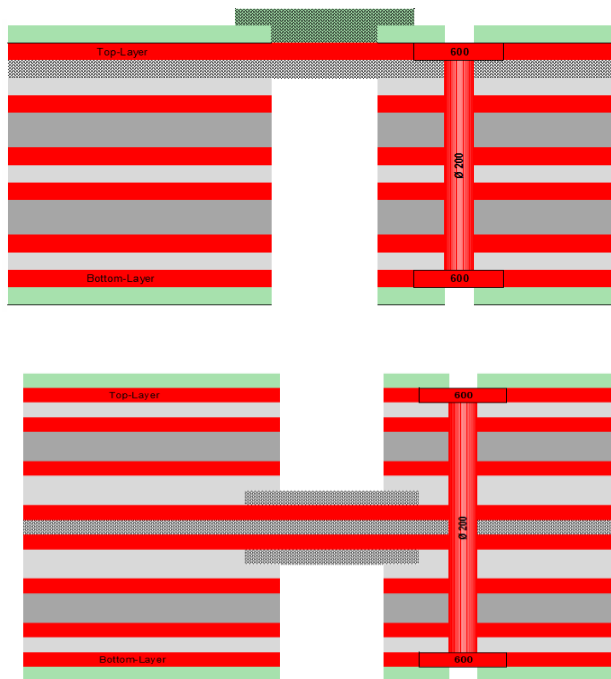
Symbol	Beschreibung	Technischer Standard	Erhöhte Anforderung
L	Lötflächen und -abstände		siehe WE Basic Design Guide
A	Minimale Viaquerdurchmesser → Tardrops empfohlen ←		siehe WE Basic Design Guide
B	Erdschleifen durchgehende Vias		siehe WE Basic Design Guide
R	→ NFP: Non functional pads nicht anbringen ←		siehe WE Basic Design Guide
C	Abstand Cu - Ankerlage zu Starflex Übergang (Blickem)	≥ 300 µm	
D	Abstand Cu - Innenlage zu Starflex Übergang	≥ 500 µm	
E	Abstand Lötfläche zu Referenz	≥ 300 µm	
F	Abstand Induktives Cu - außerhalb des Starflex Übergangs	≥ 300 µm	
G	2F: Abstand Vias zu Starflex Übergang	≥ 1500 µm	1600 µm
H	2F: Abstand Vias zu Starflex Übergang	≥ 2000 µm	1500 µm
I	Engstellung IPC2223: 3,18mm - V-Pad Durchmesser		
J	Länge des Induktives bis 2F (2F Starflex Rückstrom)	≥ 5mm	≥ 2,5mm
K	Minimale Einlötlötlänge direkt am Boardcover	1,6mm	1,0mm
L	Kernbearbeitung Handreich: kein Kern zulässig!		

→ weitergehende Spezifikationen auf Anfrage möglich, sprechen Sie mit uns: fox@we-online.de

Stand 27.01.2017 AS 3 / 3 www.we-online.de

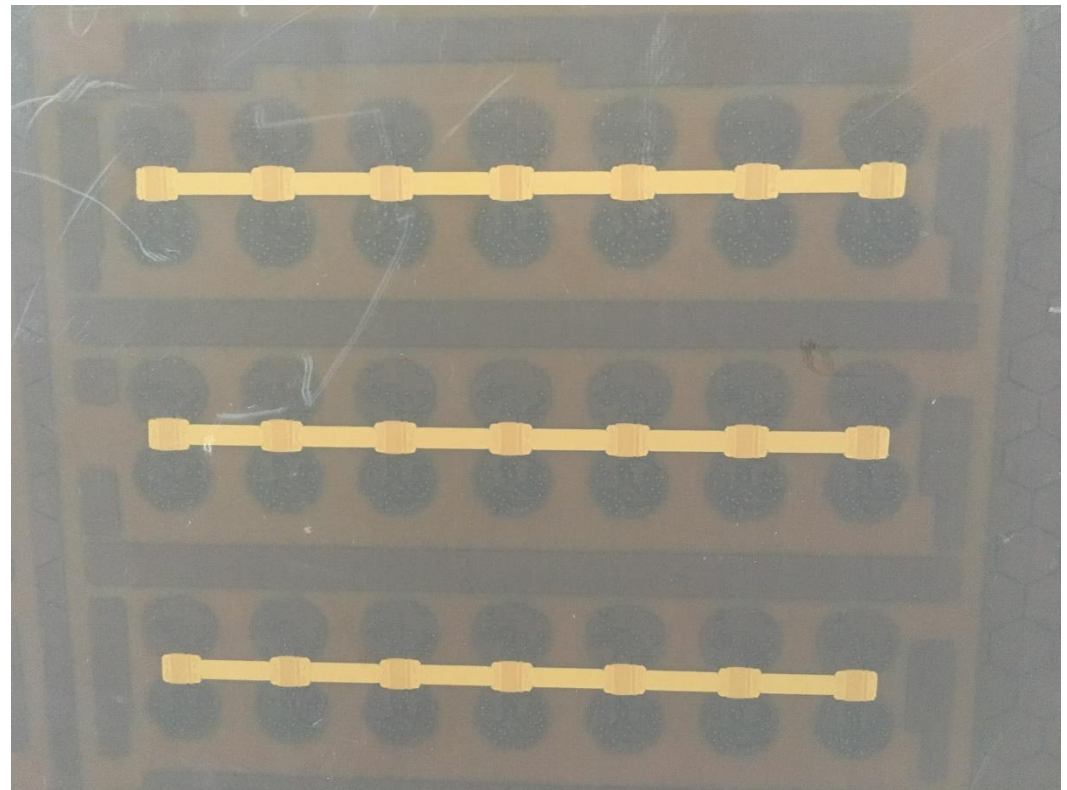
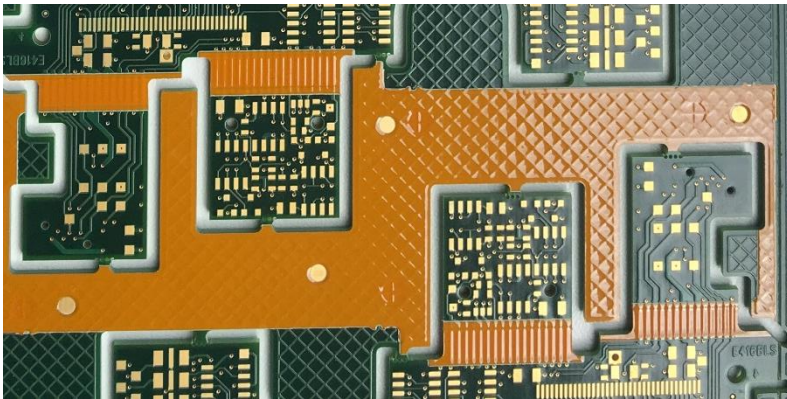
Design Flex Area: Application Coverlay Folie

- **Mechanical Design influences expense for PCB production: Array Creation**
 - reduces expense
 - improves Registration Accuracy



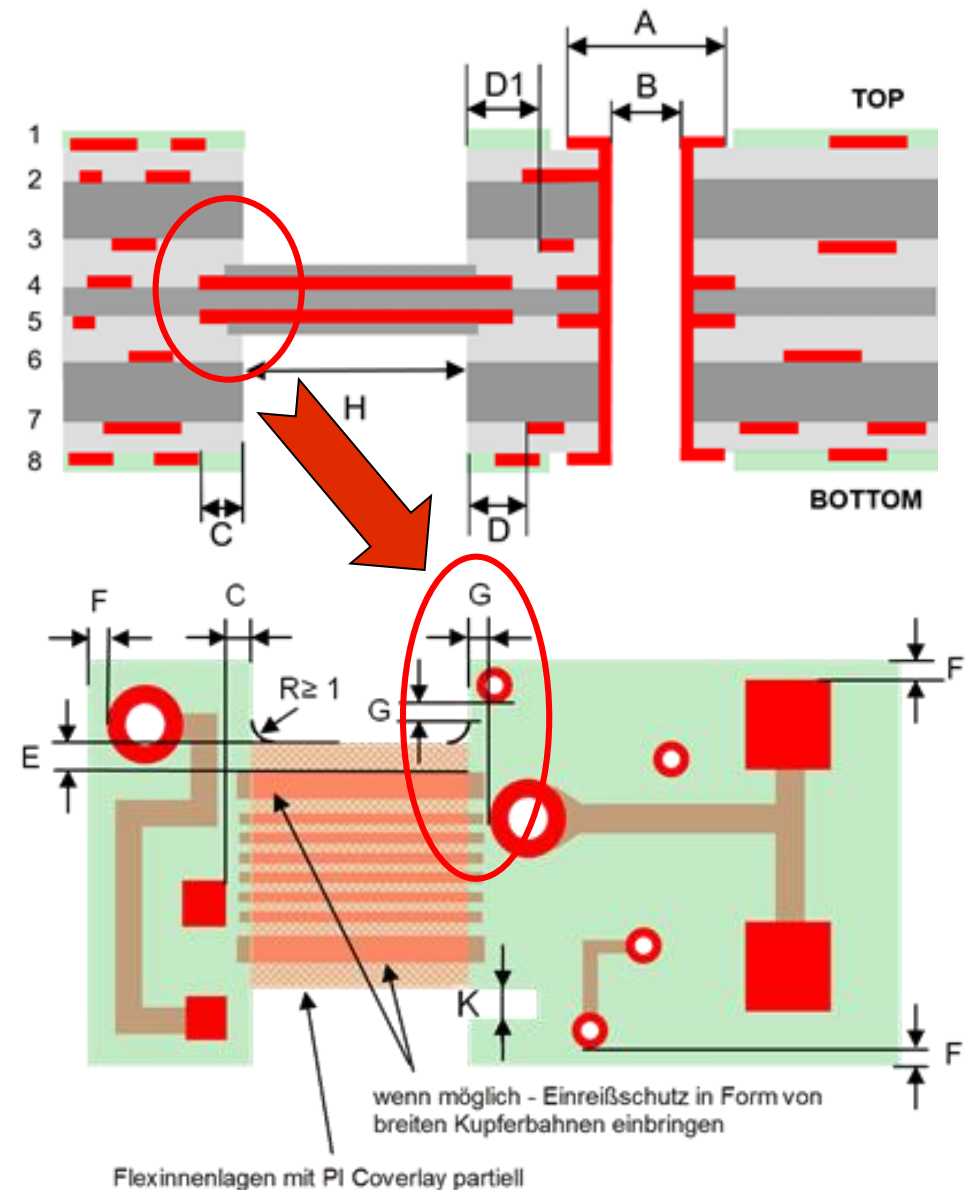
Design Flex Area: Application Coverlay Folie

- Examples:

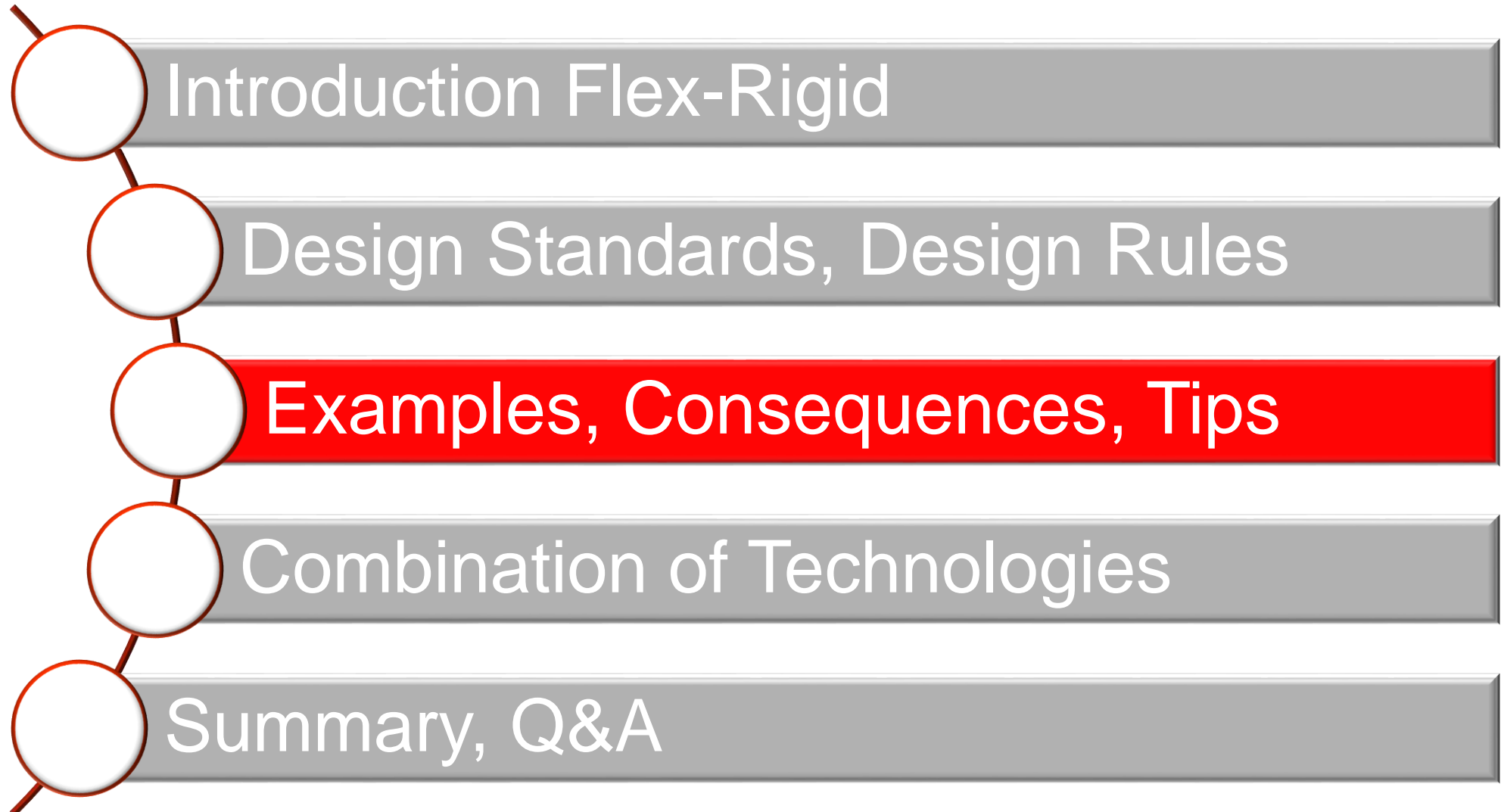


Design Rules

- **Overlap Coverlay foil**
 - Recommendation: 1,0mm
 - Securing Isolation in spite of registration tolerances and material shrinking
- → limits Spacing „G“
- **Standard values for „G“:**
 - 1F-xRi (flex soldermask): 1,0mm
 - 1F-xRi (coverlay): 1,5mm
 - 1F-xRi (coverlay UL): 2,0mm
 - xRi-2F-xRi: 1,5mm
 - xRi>2F-xRi: 2,0mm
- IPC-2223C:
3,18mm+ $\frac{1}{2}$ Pad Diameter



Agenda

- 
- A vertical line of five white circles with red outlines, connected by a thin red line. The circles are positioned to the left of the agenda items, which are contained within horizontal grey bars. The third bar, 'Examples, Consequences, Tips', is highlighted in red.
- Introduction Flex-Rigid
 - Design Standards, Design Rules
 - Examples, Consequences, Tips**
 - Combination of Technologies
 - Summary, Q&A

Flex-Rigid Stack-up: Mistake 1

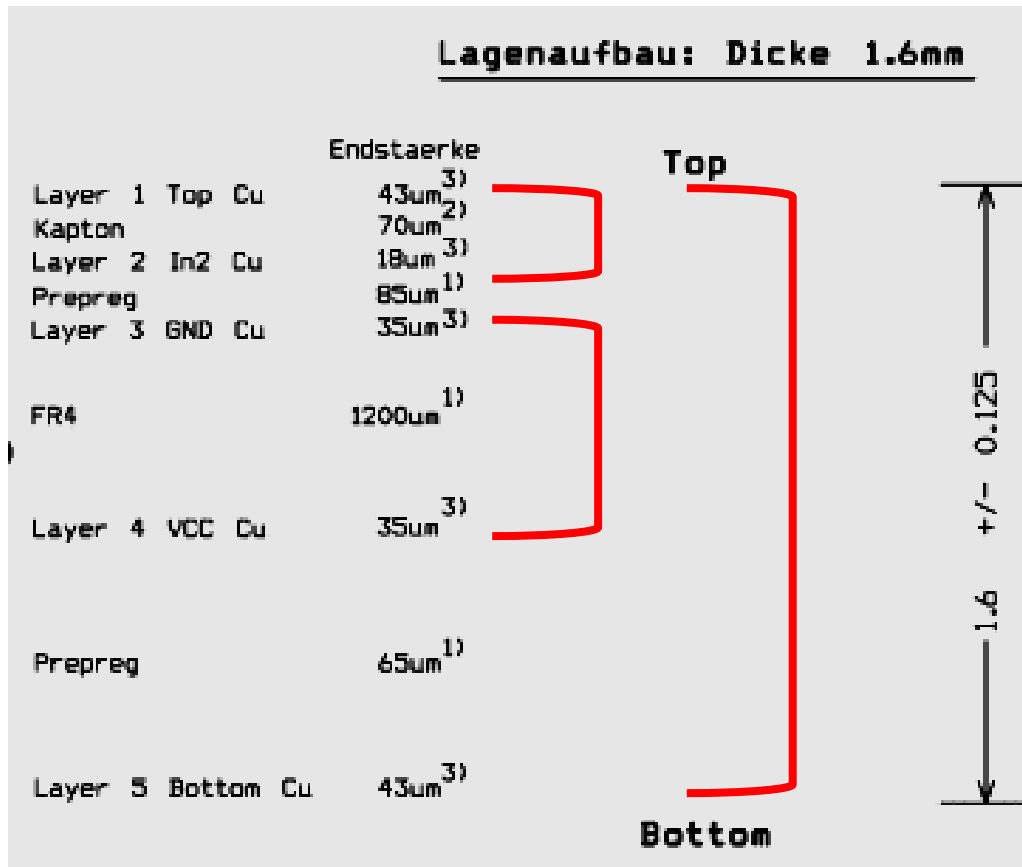
- Excerpt of PCB drawing:

Lagenaufbau: Dicke 1.6mm

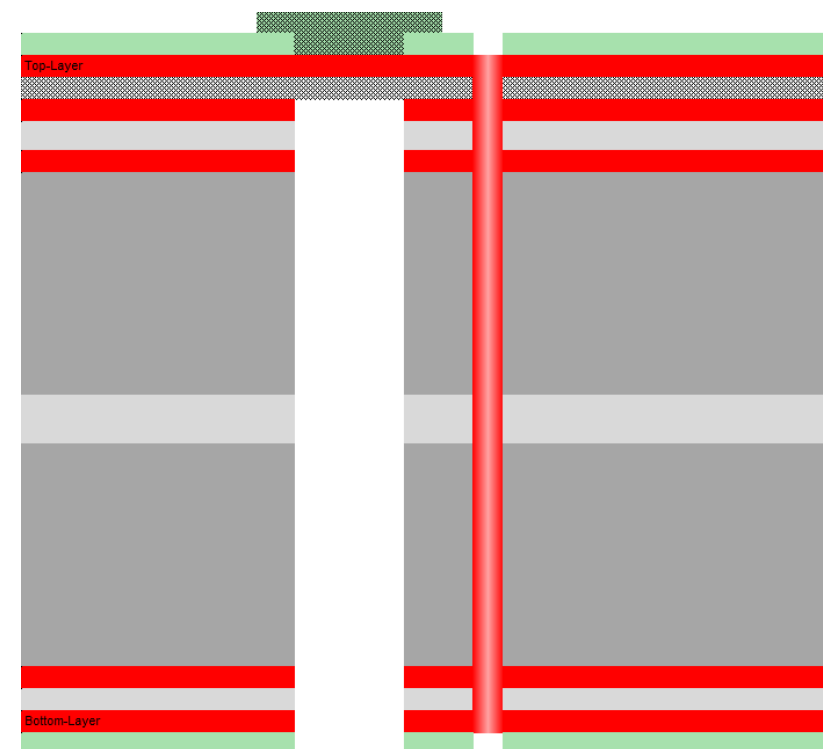
	Endstaerke	
Layer 1 Top Cu	43um ³⁾	} Top
Kapton	70um ²⁾	
Layer 2 In2 Cu	18um ³⁾	} }
Prepreg	85um ¹⁾	
Layer 3 GND Cu	35um ³⁾	} Bottom
FR4	1200um ¹⁾	
Layer 4 VCC Cu	35um ³⁾	} }
Prepreg	65um ¹⁾	
Layer 5 Bottom Cu	43um ³⁾	

+/- 0.125

1.6



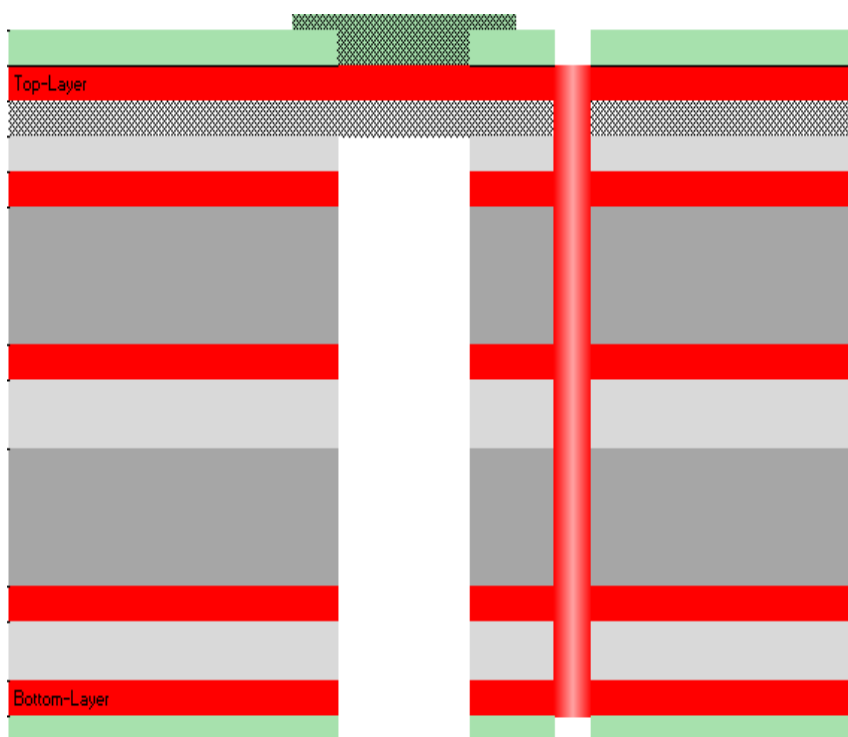
- possible Layer stack-up: 2F-3Ri



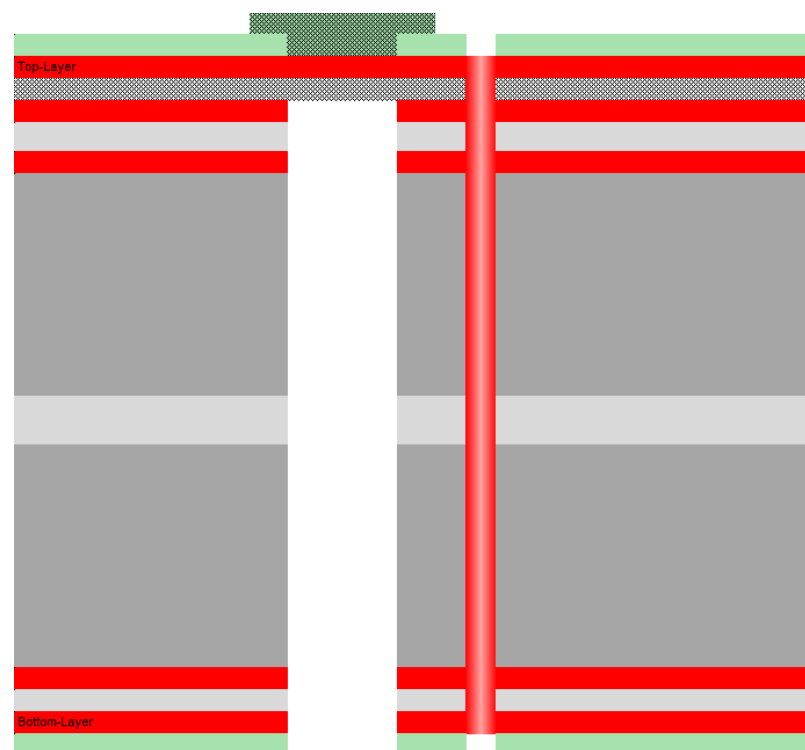
- BUT: no copper Layout on Layer 2 of flex core!**

Flex-Rigid Stack-up: Mistake 1

- **much better Stack-up:
1F-4Ri**



- **possible Layer stack-up :
2F-2Ri**

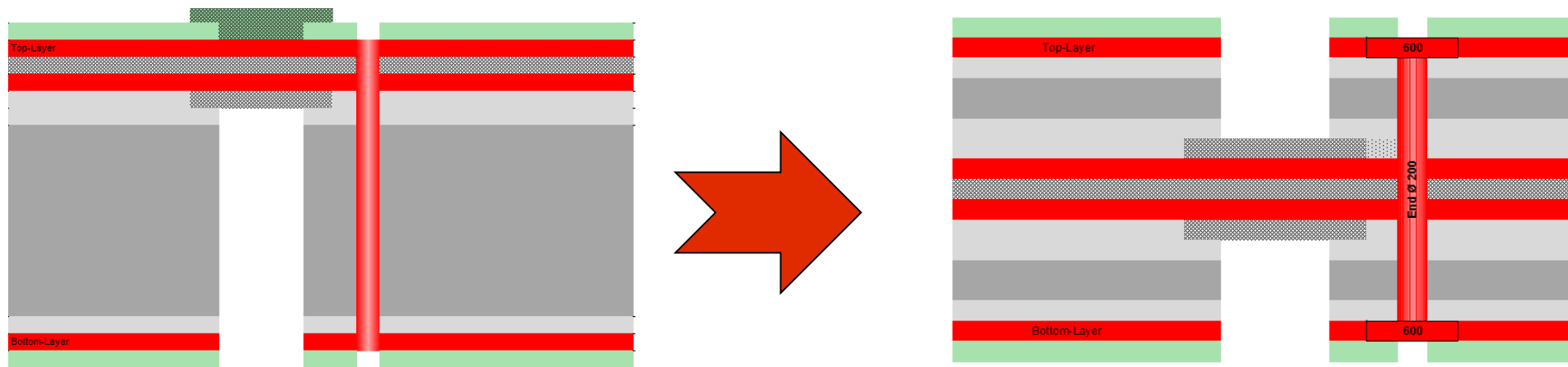


- **Cost Savings: about 30% with same Performance!**
- **at the same time reduced Risk of Warpage!**

Flex-Rigid Stack-up: Mistake 2

- Customer Stack-up:
2F-1Ri with RA-copper on Flex

- better Stack-up :
1Ri-2F-1Ri



- Problem: Plating of copper / ED on RA-Base Copper
 - reduced Flex-Performance!
 - or high expense to avoid Plating on Flex!



Flex-Rigid Stack-up: Mistake 3

- **Asymmetric Stack-up**
 - different milling depths
 - higher Risk of Warpage
 - unnecessary Complexity and Risk in Production

Rigidflex 9RI-2F-5RI

PCB Thickness : 2,81 mm +/- 10% Flex Thickness: 0,20 mm +/- 0,05mm

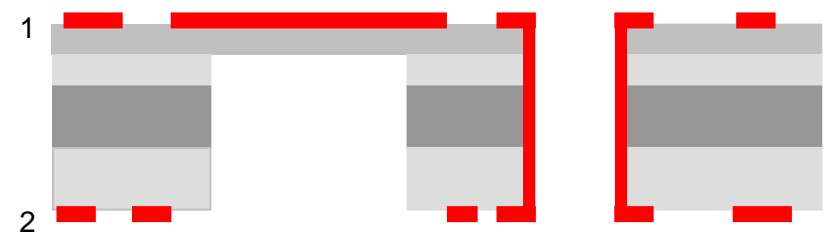
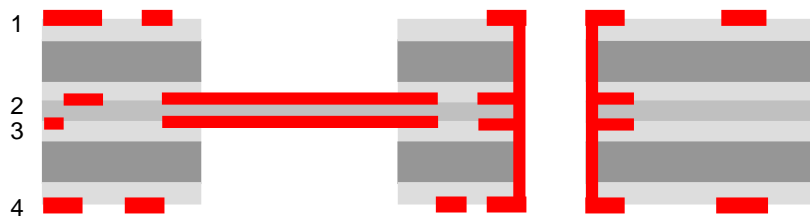
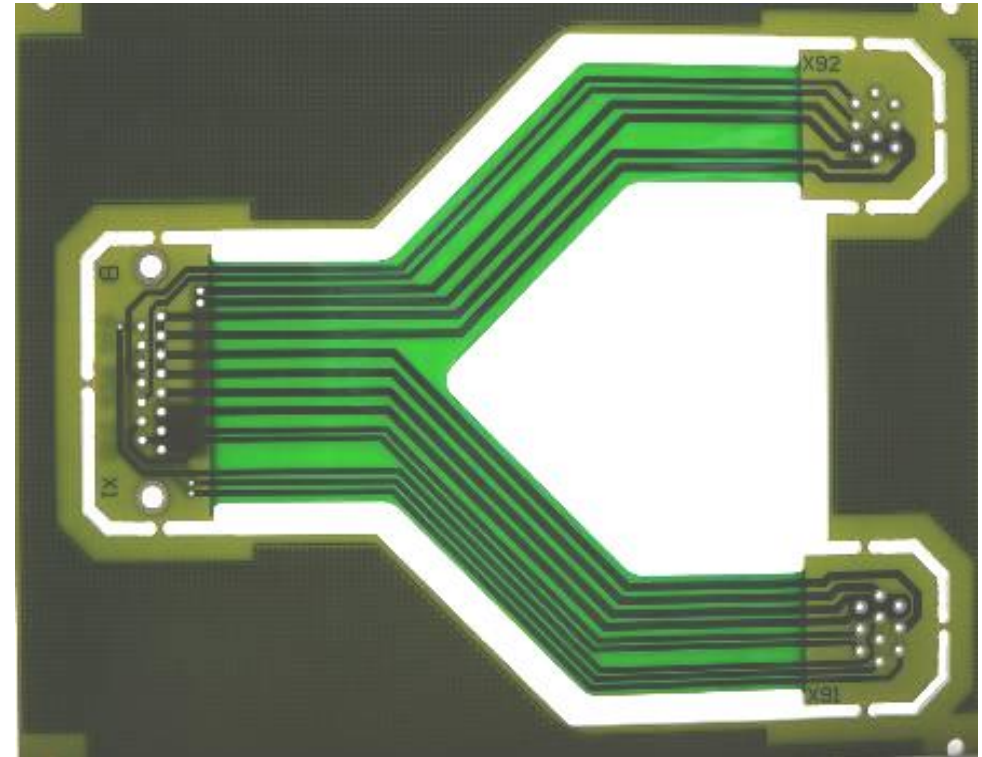
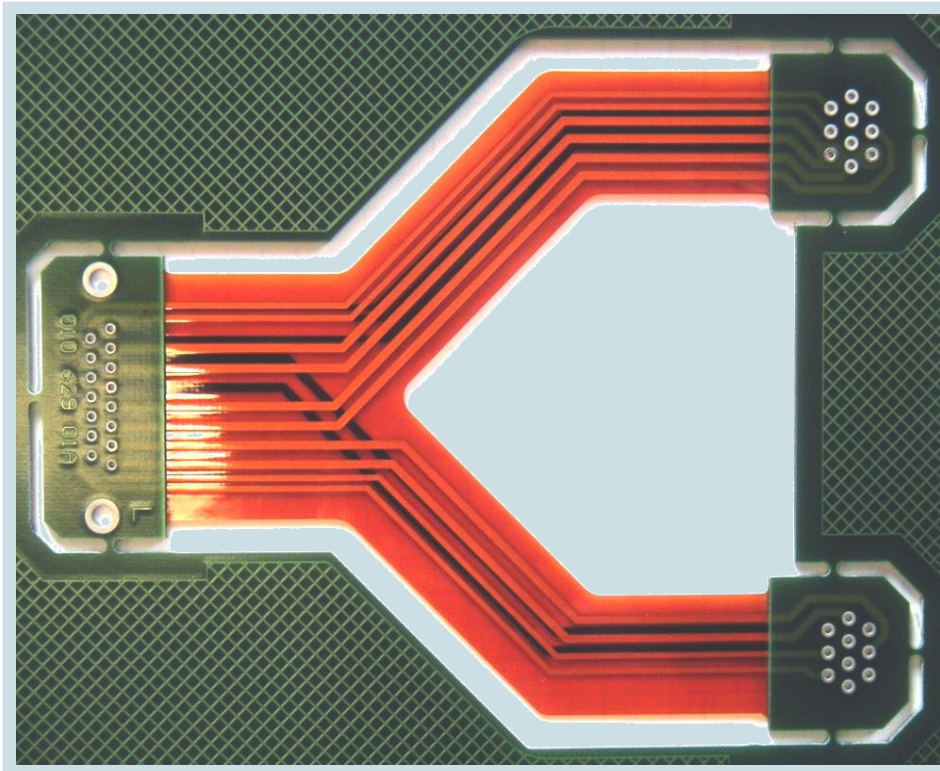
Rigid area Structure	Flex area Thickness	Rigid area Thickness	Material describing		Flex area Structure	Viatypes
Soldermask		12				
L1		35		Top-Layer		
Prepreg		360	2 x 7628			
L2		35				
Core		100	HTG 170'			
L3		35				
Prepreg		125	2 x 1080			
L4		35				
Core		100	HTG 170'			
L5		35				
Prepreg		125	2 x 1080			
L6		35				
Core		100	HTG 170'			
L7		35				
Prepreg		125	2 x 1080			
L8		35				
Core		100	HTG 170'			
L9		35				
Prepreg		180	3 x 1080			
	40		Coverlay			
L10	35	35				
Flex Core	50	50	Polyimid AP9121			
L11	35	35				
Prepreg		180	Coverlay			
	40		3 x 1080			
L12		35				
Core		100	HTG 170'			
L13		35				
Prepreg		125	2 x 1080			
L14		35				
Core		100	HTG 170'			
L15		35				
Prepreg		360	2 x 7628			
L16		35		Bottom-Layer		
Soldermask		12				

Flex-Rigid Stack-up: Mistake 4

2 flexible layers inside



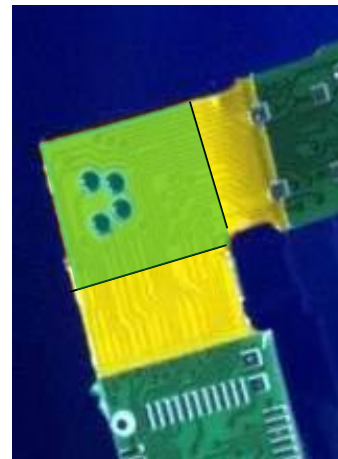
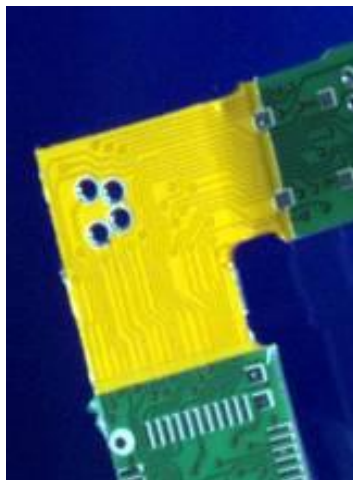
1 flexible layer outside



Flex-Rigid Layout: PTH



- **Mistake 1: contacts and plated through holes in flex areas**
 - additional Processes and Costs
 - Risk in Reliability



→ *Avoid Vias in Flex
in general !*



Flex-Rigid Layout: PTH

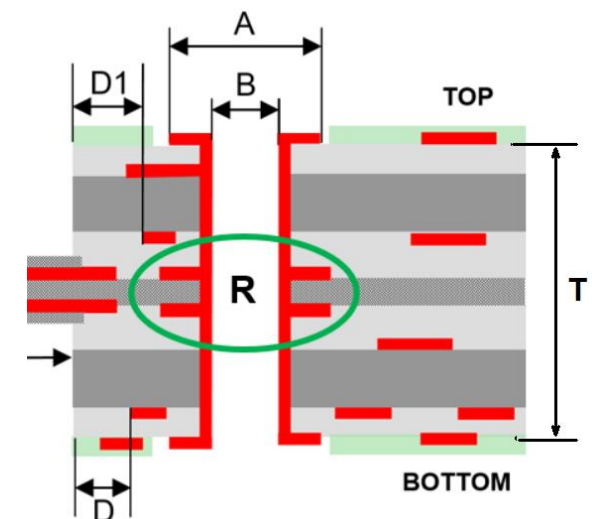
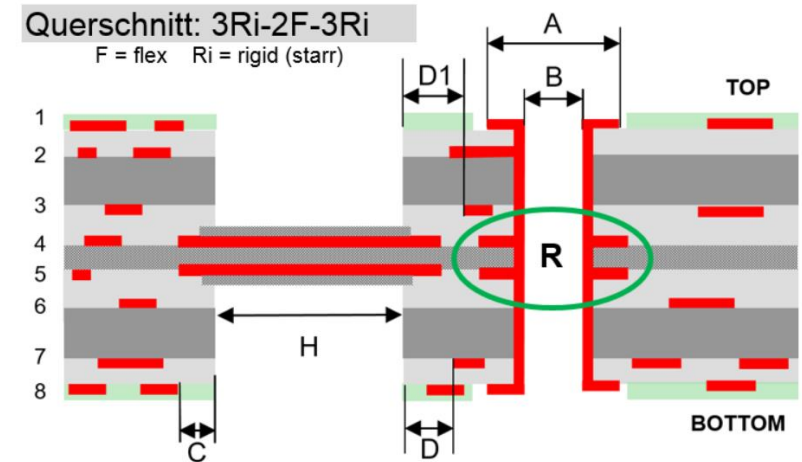
■ Mistake 2 : NFP Removal

- Drilling quality in flexible core could become dramatically worse!
- Consequence: Plating problems and thus problems in Reliability

→ Don't perform NFP Removal on Flex Layers! „R“

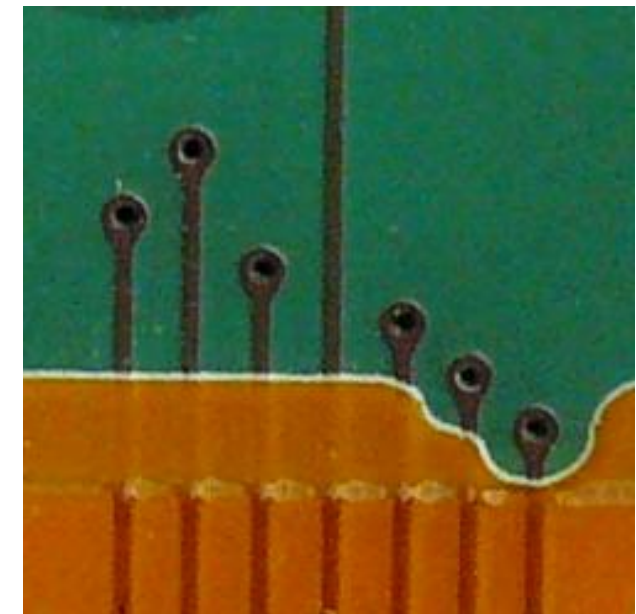
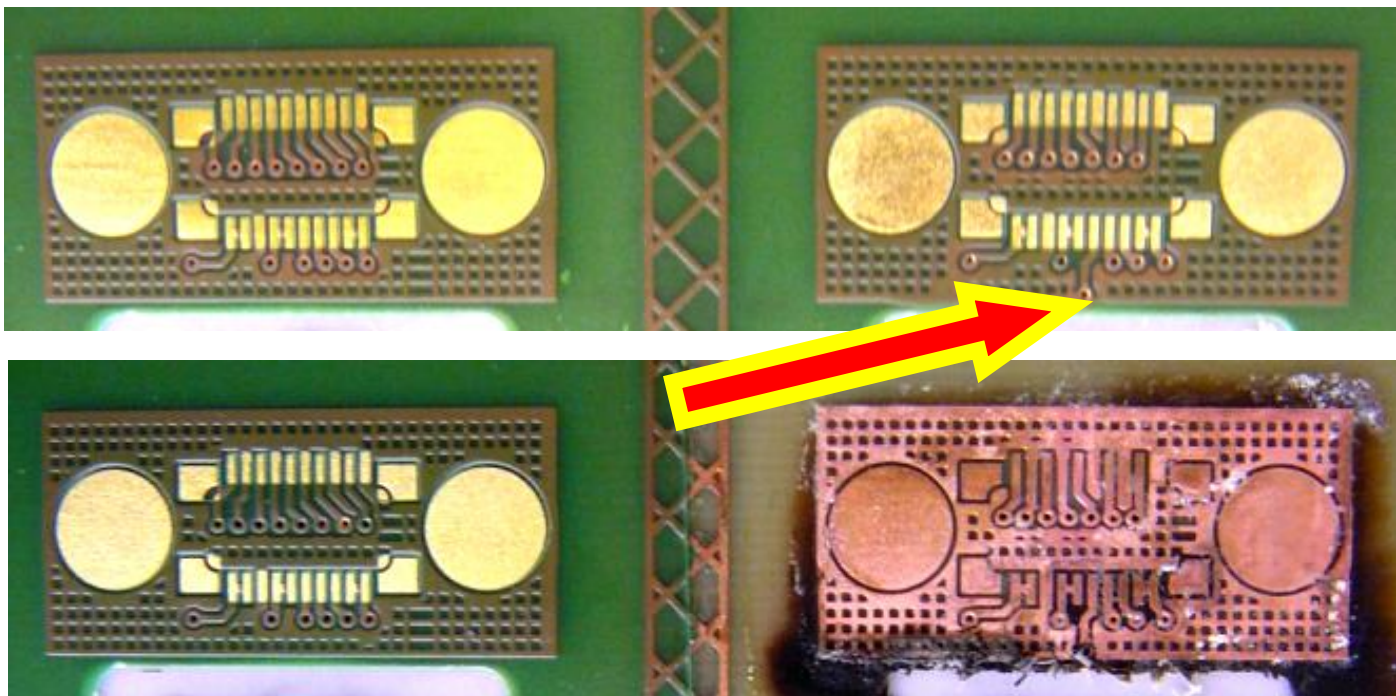
■ Mistake 3: AR Aspect Ratio too large (PCB Thickness T / PTH-Diameter B)

- reduced copper distribution in barrel
- reduced barrel stability, current capacity and heat transfer
- Risk of barrel cracking in case of thermal load
- for Flex-Rigid with Flex core inside even more problematic



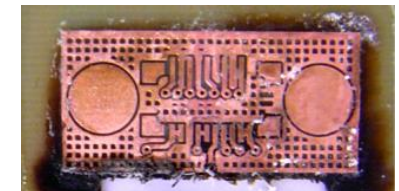
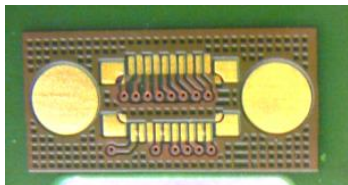
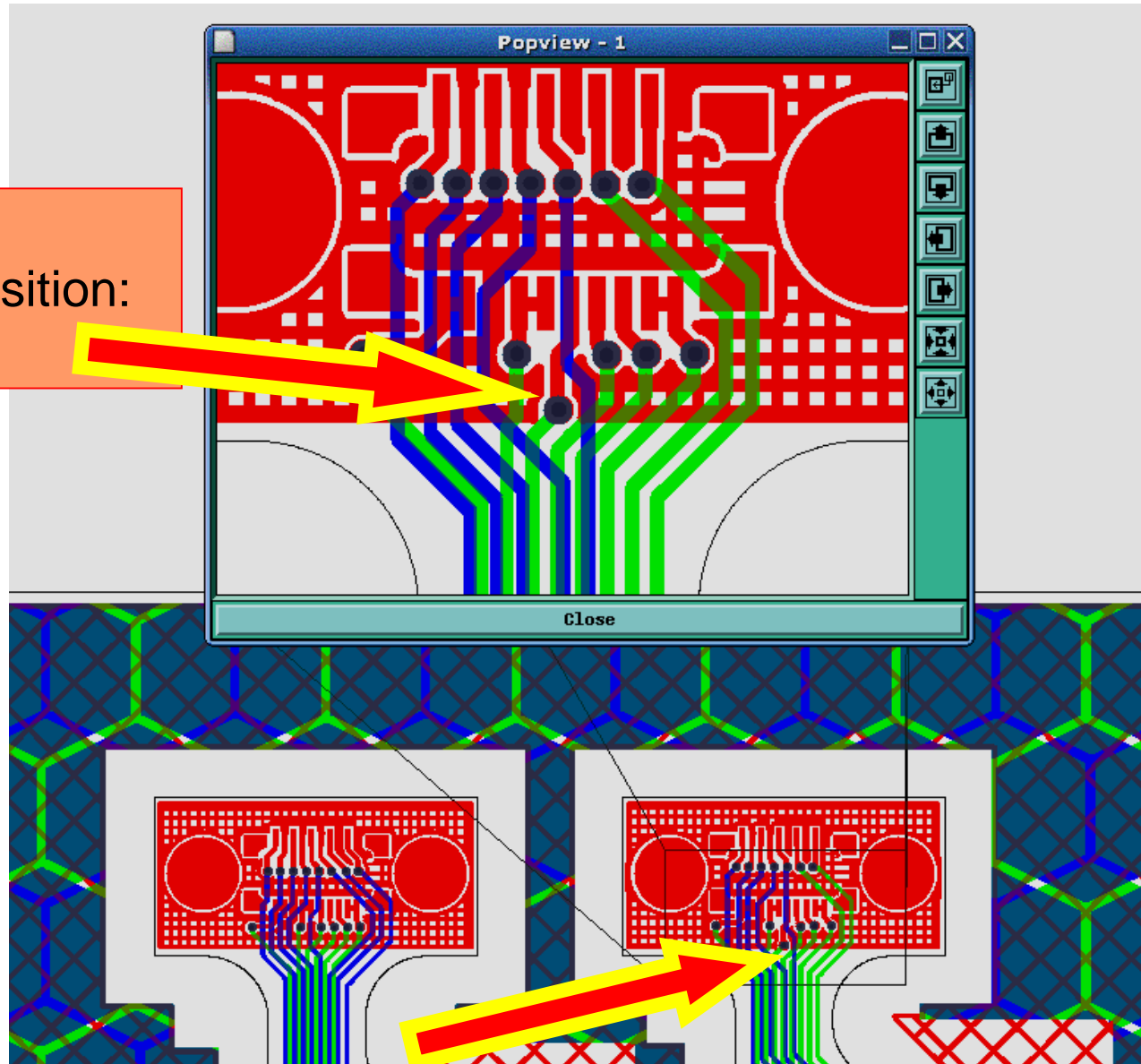
Flex-Rigid Layout: PTH

- **Mistake 4: Spacing PTH to Flex-Rigid Transition**
 - Risk of Scrap during Production and
 - Risk of Failure in the Field in worst case



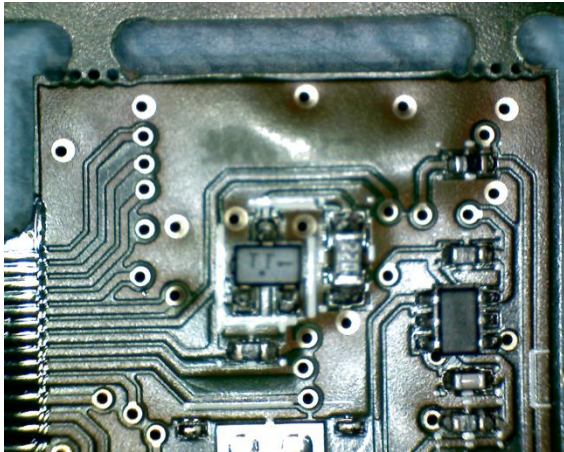
Flex-Rigid Layout: PTH

Spacing
PTH wall – Transition:
0,45mm

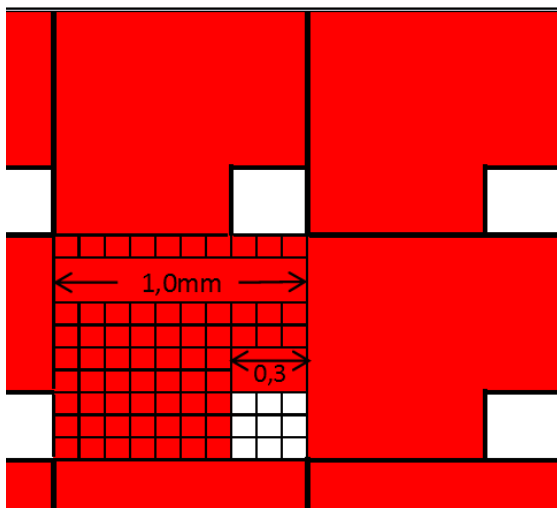


Flex-Rigid Design: Copper Pattern Mistake 1

- **Problem: large unpierced copper areas**



- **Correction: shield opening**



- **Other possible reasons for Delamination:**



- **Do not dry in a stack!**
- **use Oven with outgoing air!**

Flex-Rigid Design: Copper Pattern

- **Mistake 2 : copper lines on same place on all flexible layers**
 - partially big differences in Thickness
 - bad pressure distribution during Lamination process
 - Reduction in Flexibility resp. Bendability



- **Solution: Copper Offset**



Flex-Rigid Mechanics: Mistake 1

- Airgap-Stack-up
 - Flex Length „H“ too small
 - „Buckling“ makes flexible area very stiff

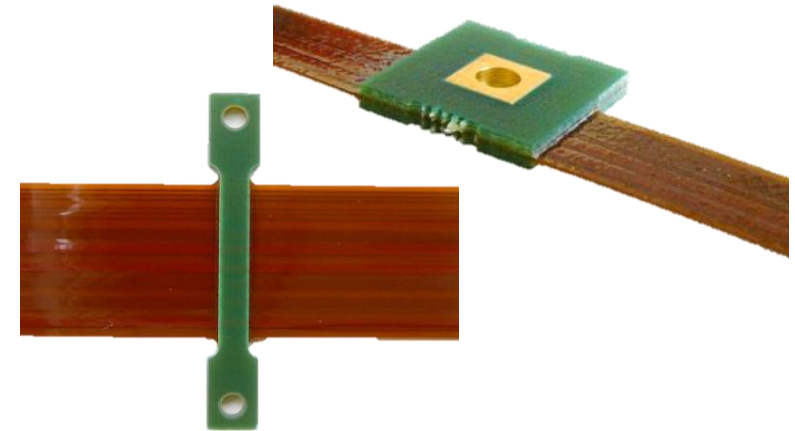
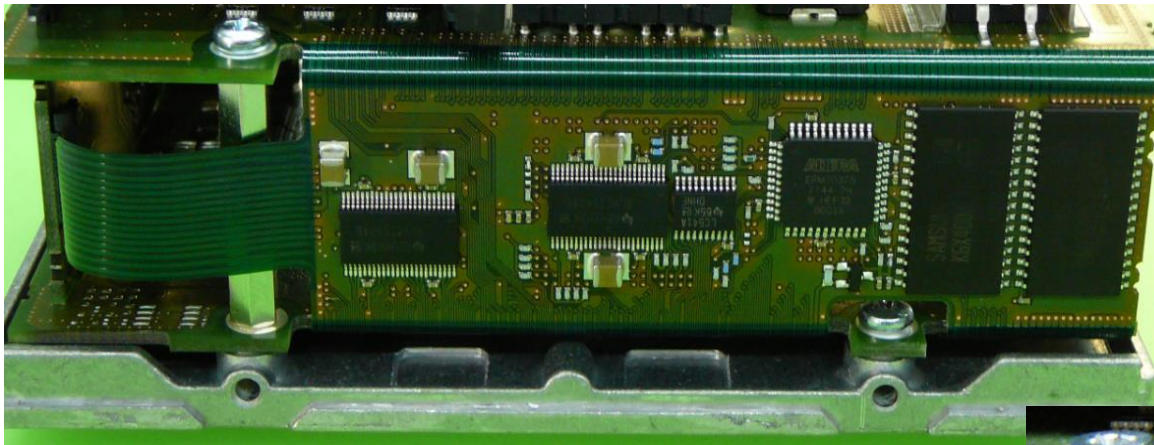
- Recommendation:
 - $H \geq 50\text{mm}$

Customer													
Job													
WE-number													
Engineer		W. Ohsen											
date		15.04.2014											
Rigidflex 1Ri-2F+2F+2F-1Ri													
PCB Thickness : 2.02 mm +/- 10% Flex Thickness (each): 0.29 mm +/- 0.05mm													
Rigid area Structure	Flex area Thickness	Rigid area Thickness	Material describing		Flex area structure	Via type	Layer usage	Impedance					
								Z[Ohm] / Line / Space					
Soldermask		20											
L1		45											
Prepreg		65	FR4 HTG 150°										
Core		250	FR4 HTG 150°										
Prepreg		150	FR4 HTG 150°										
		50	Coverlay LF0210										
L2		70											
Flex Core		50	Polyimide AP9222R										
L3		70											
Prepreg		50	Coverlay LF0210										
		50	FR4 HTG 150°										
		50	Coverlay LF0210										
L4		70											
Flex Core		50	Polyimide AP9222R										
L5		70											
Prepreg		50	Coverlay LF0210										
		50	FR4 HTG 150°										
		50	Coverlay LF0210										
L6		70											
Flex Core		50	Polyimide AP9222R										
L7		70											
Prepreg		50	Coverlay LF0210										
		140	FR4 HTG 150°										
Core		250	FR4 HTG 150°										
Prepreg		65	FR4 HTG 150°										
L8		45											
Soldermask		20											

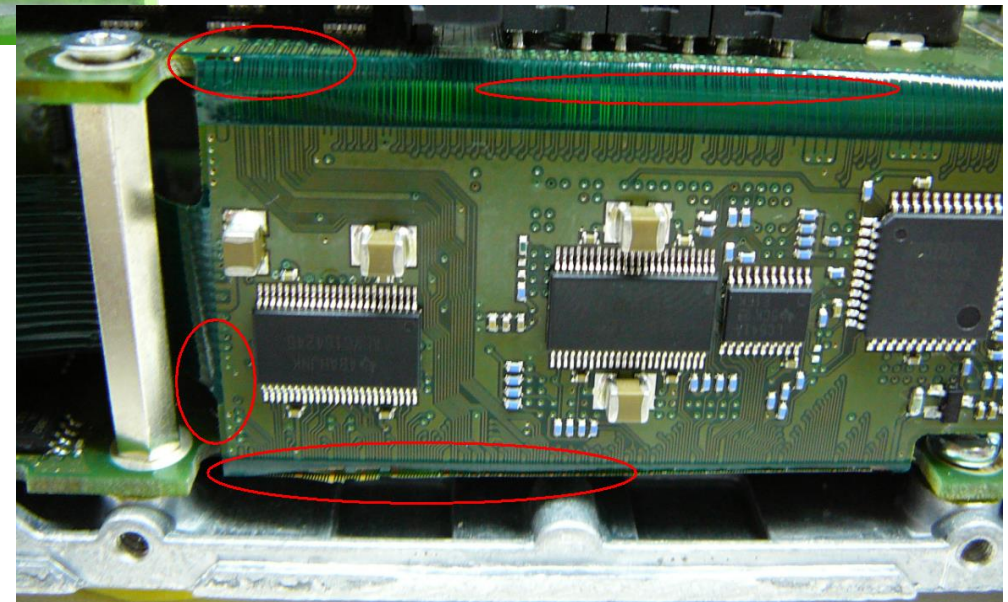


Flex-Rigid Mechanics: Mistake 2

NO fixation of one rigid area

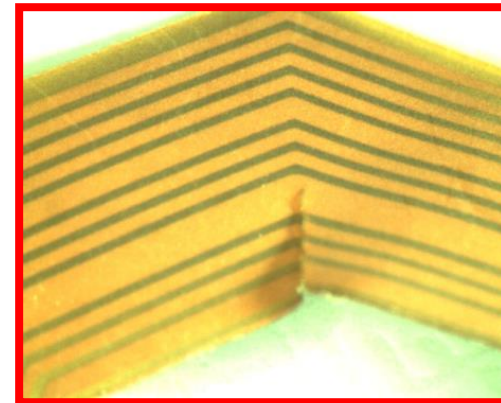


**... leads to resonance
and destruction even
with Rigid-Flex!**

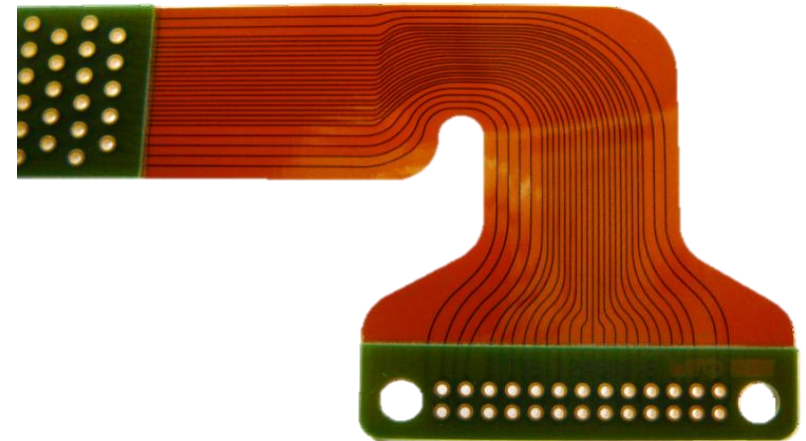


Flex-Rigid Design: Mistake Contour

- small Radii
- sharp edges
 - stress concentration results in cracks



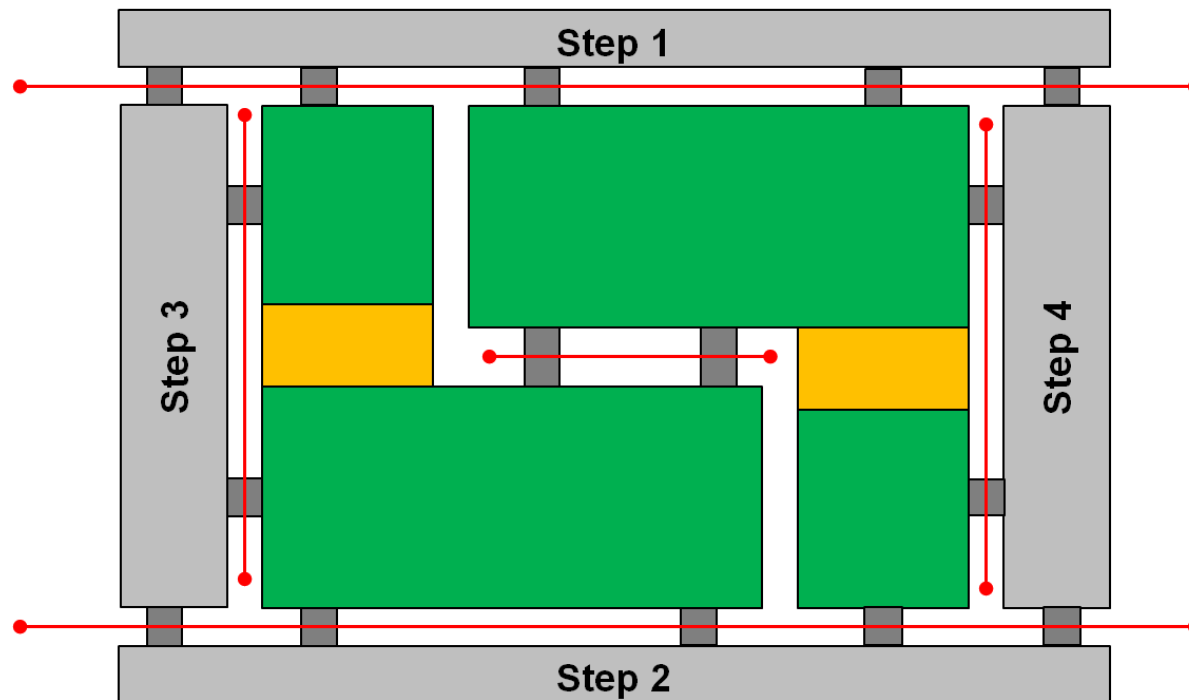
→ Design for plastics!



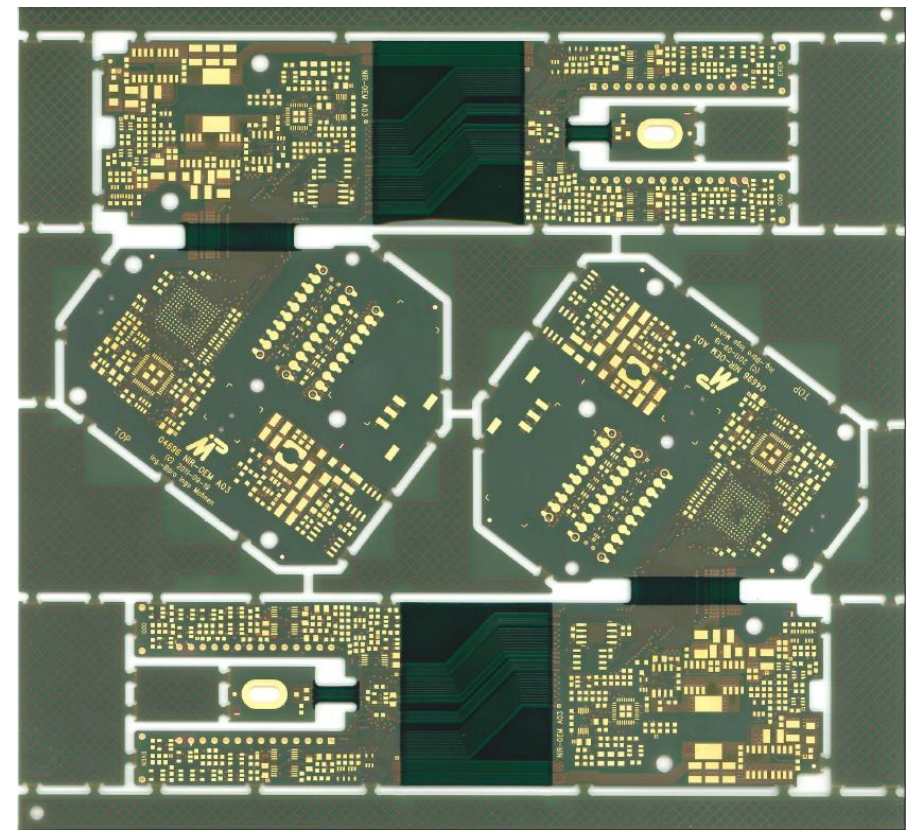
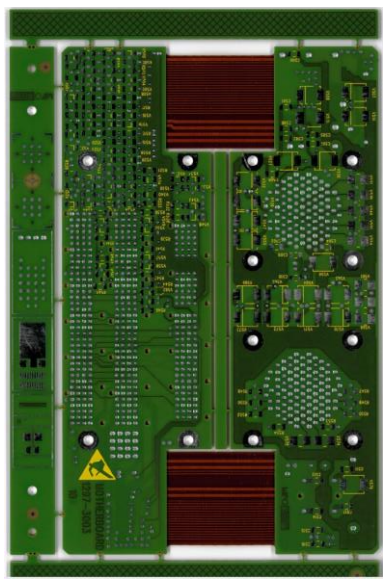
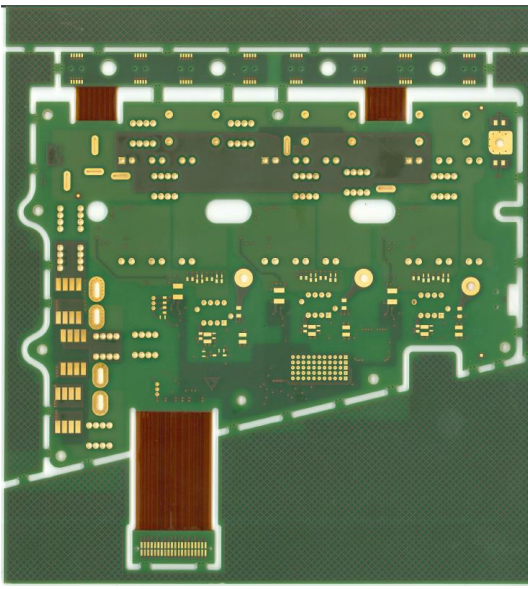
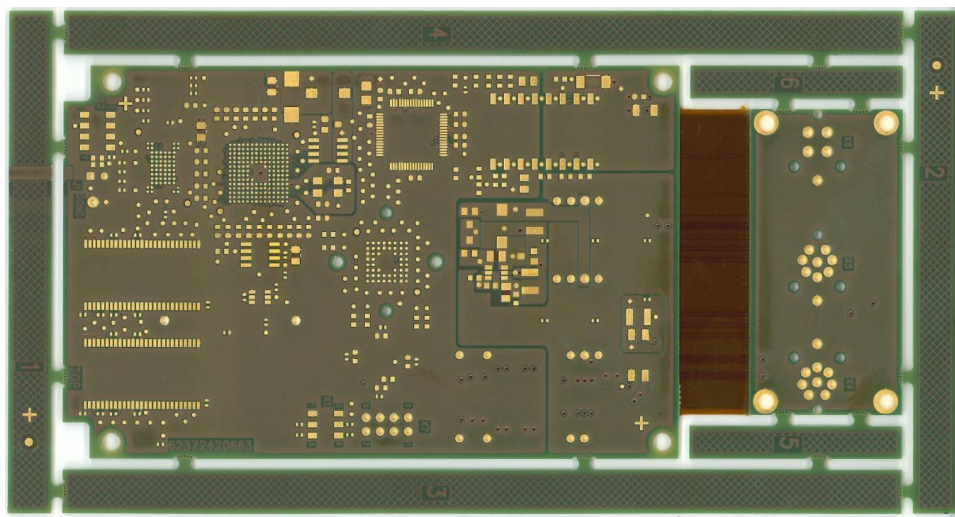
A good example!

Flex-Rigid Design: Mistake Delivery Array

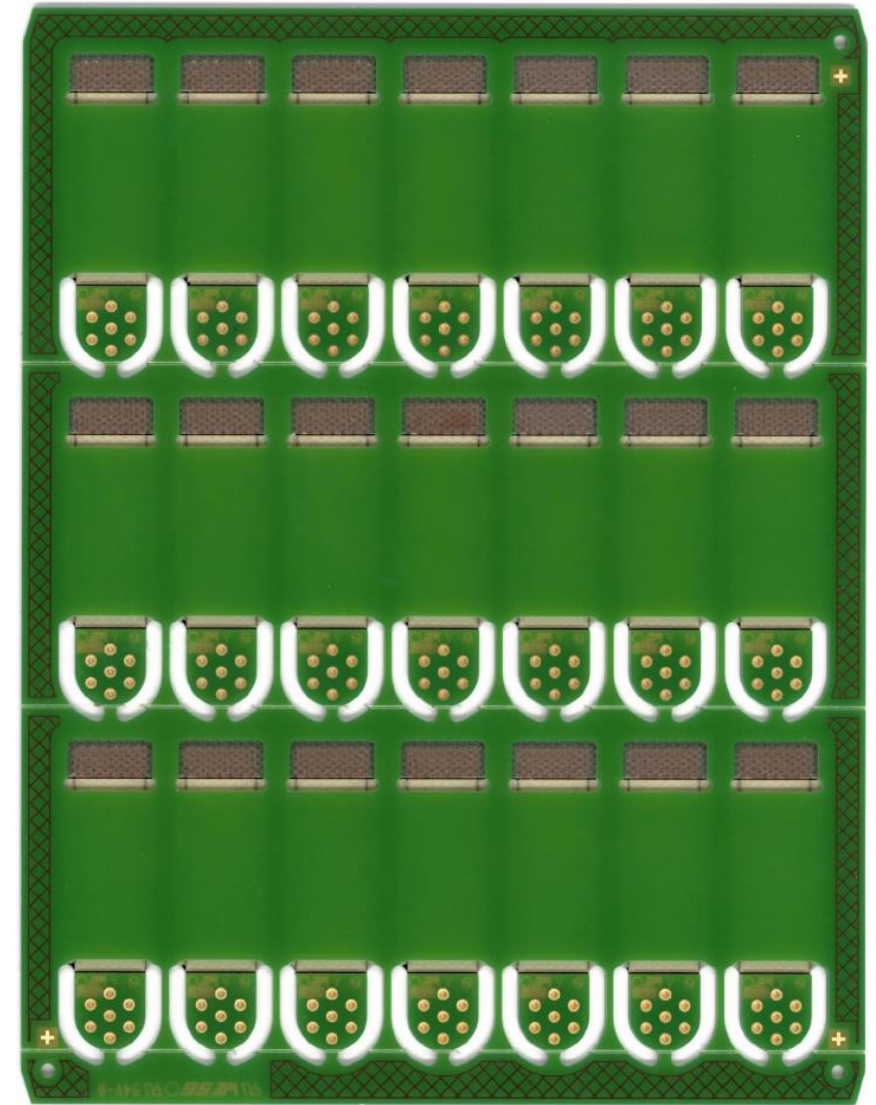
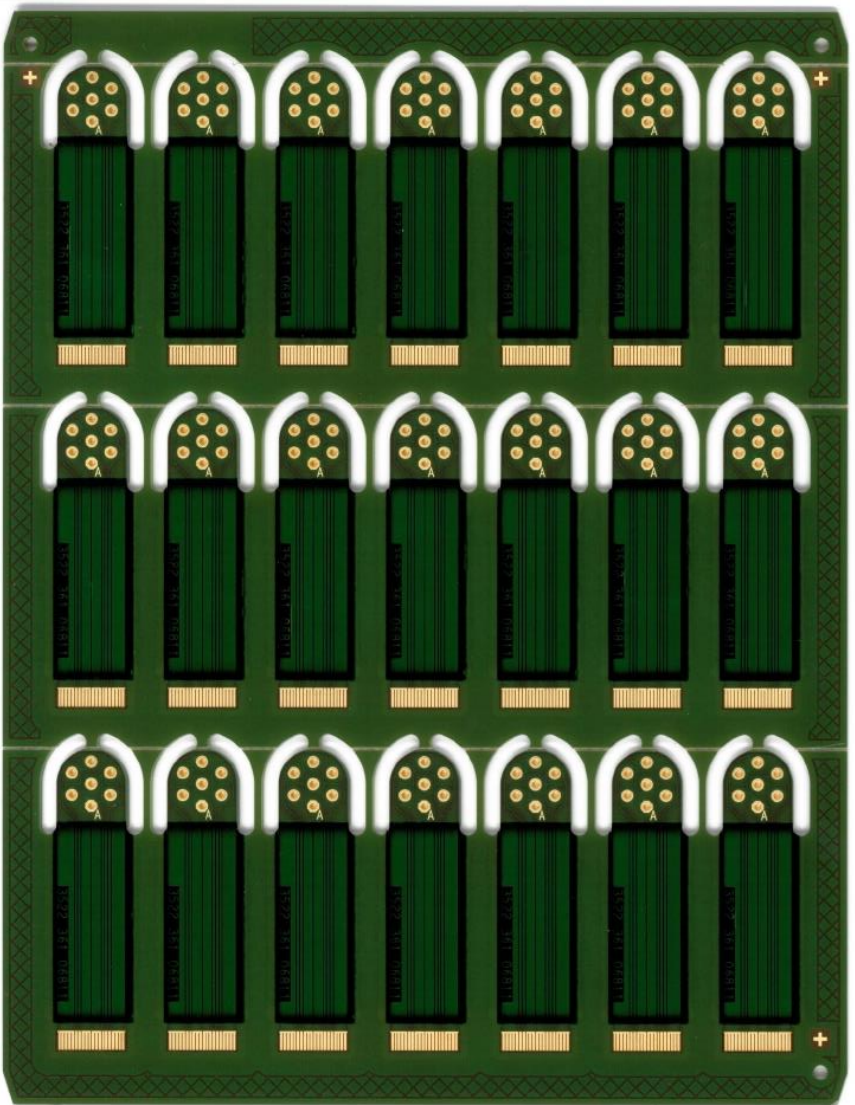
- Array too solide, hard to separate
- Missing of break points
- circulating stable frame
 - Damaging of flexible Area during Separation



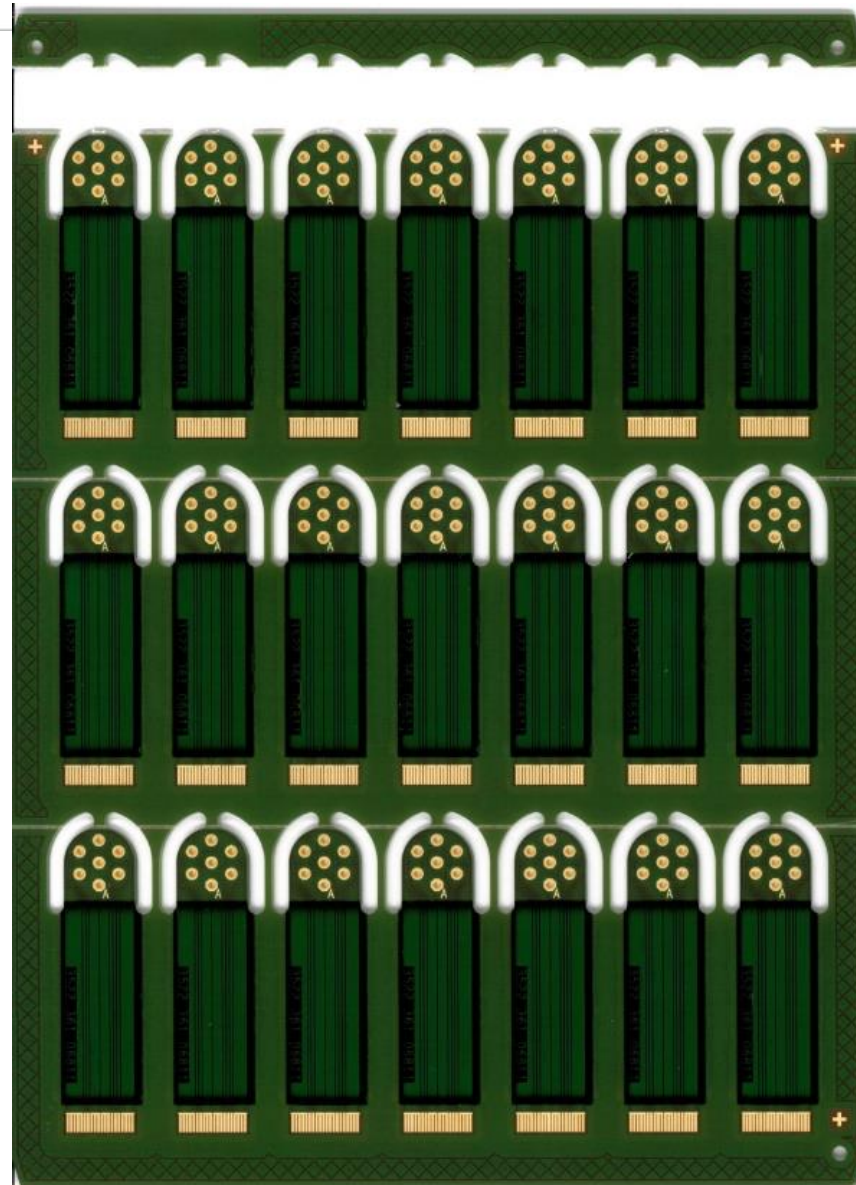
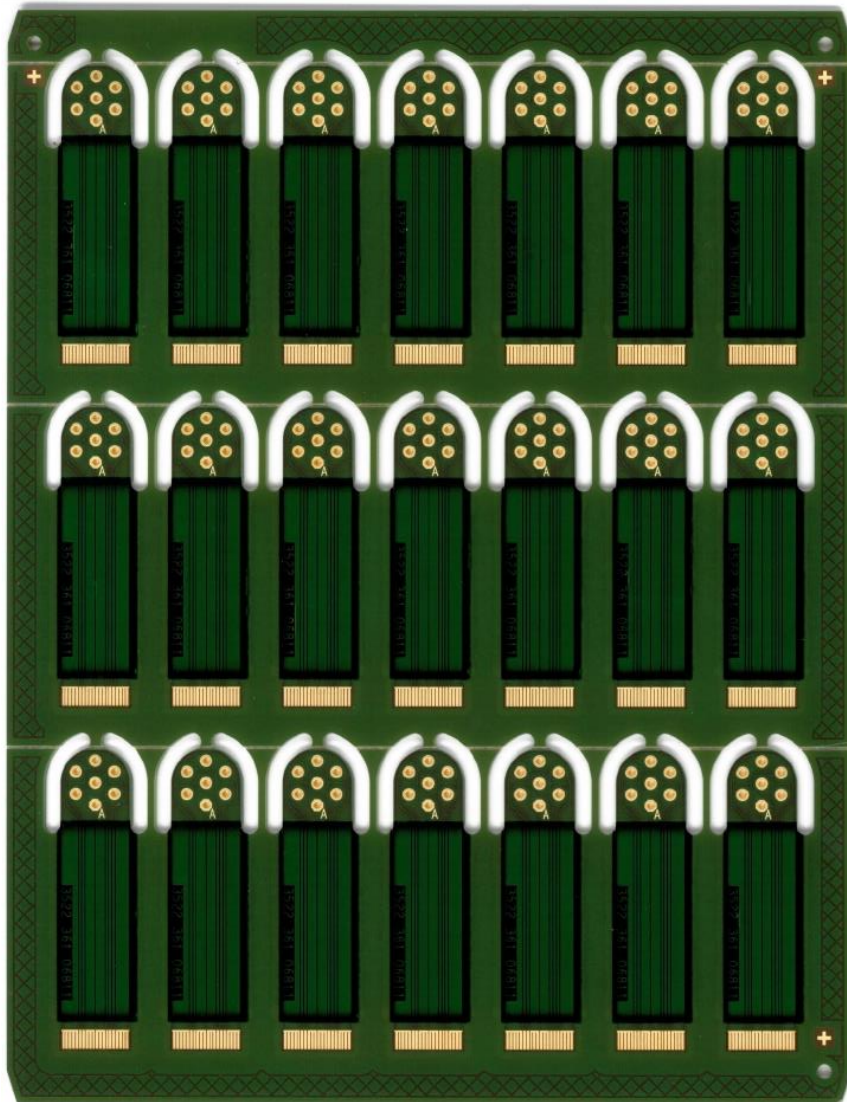
Flex-Rigid Design: Good Examples



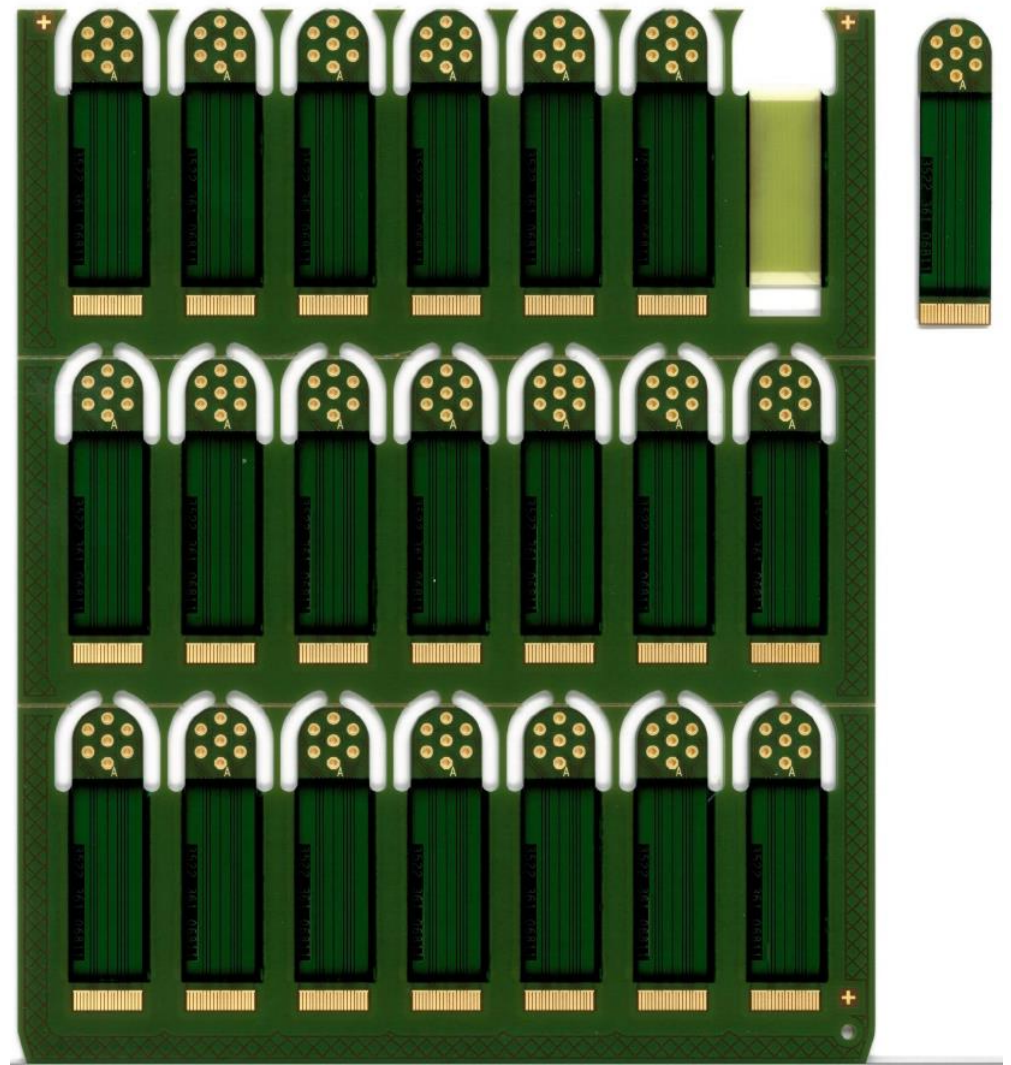
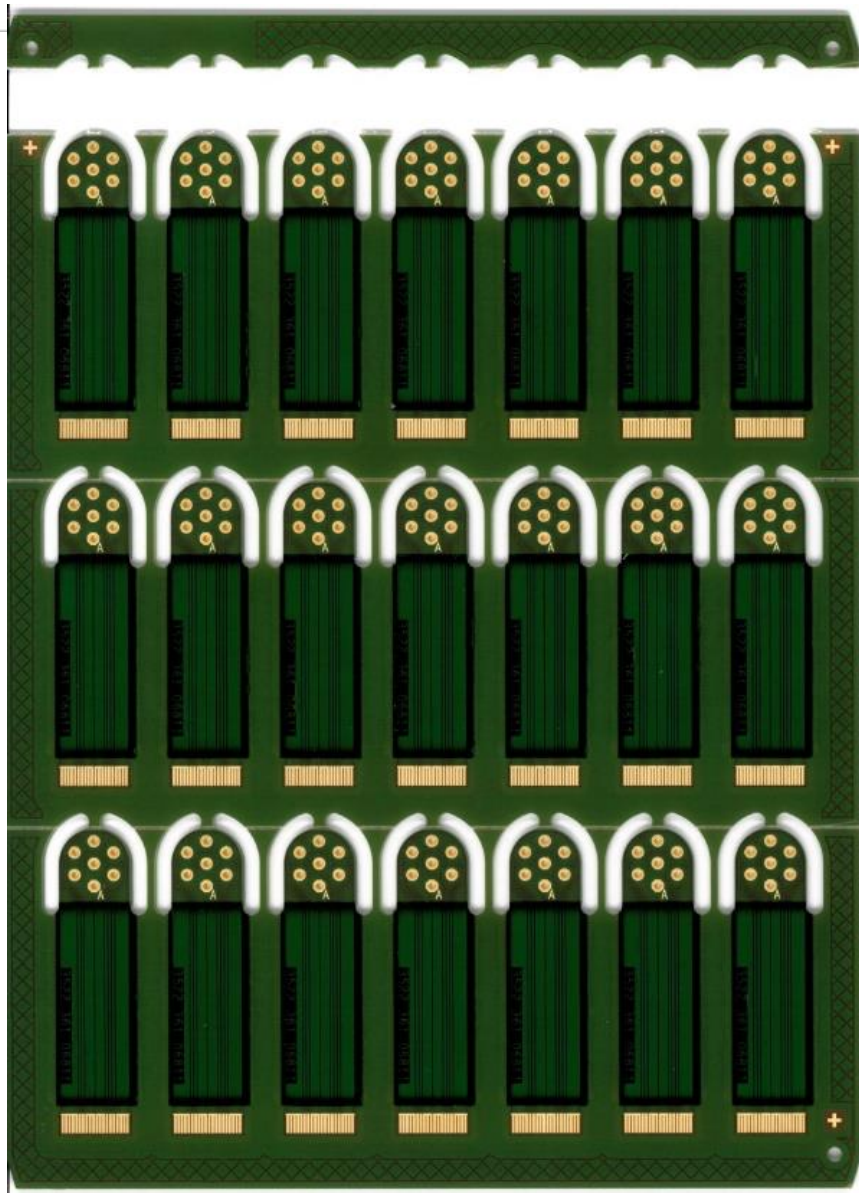
Flex-Rigid Design: Good Examples



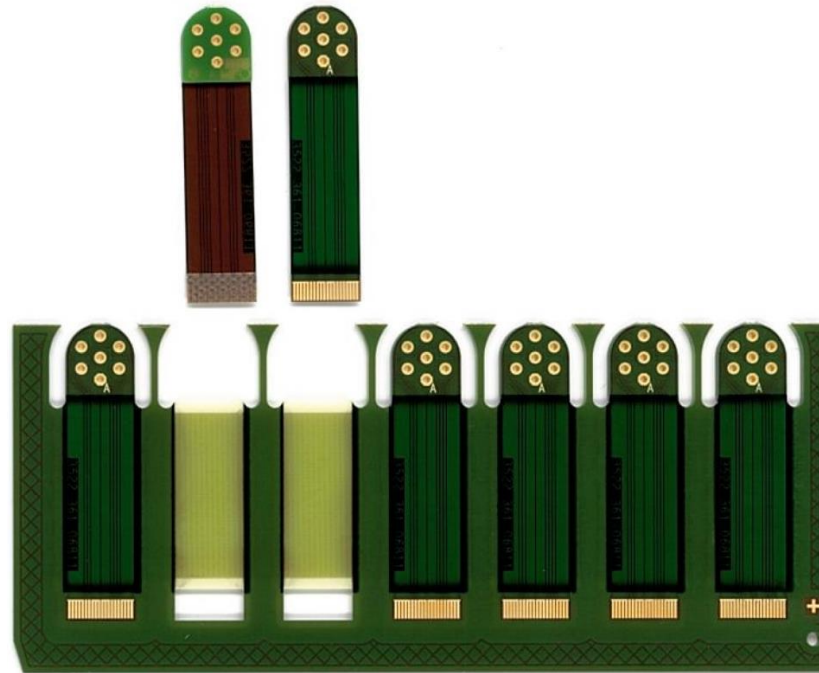
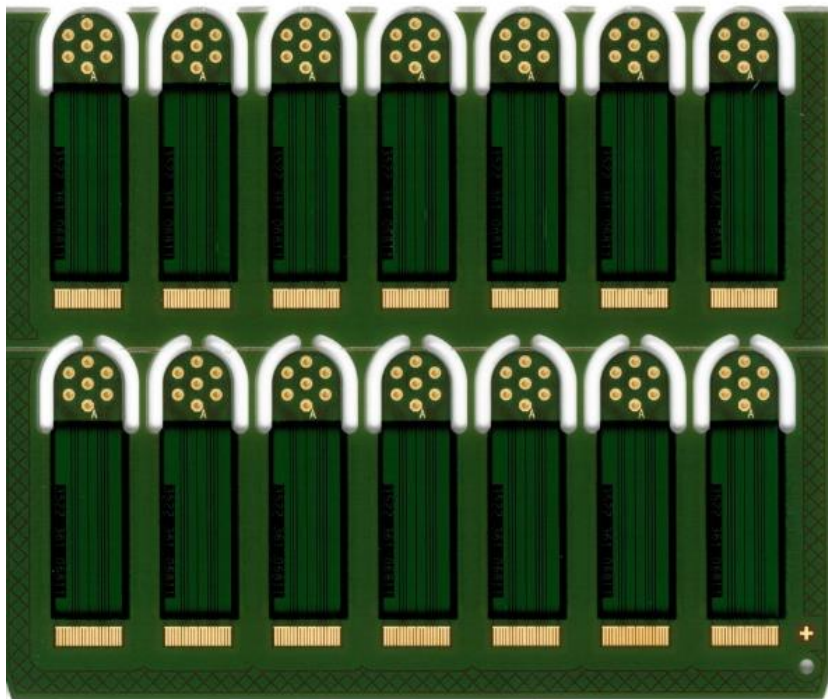
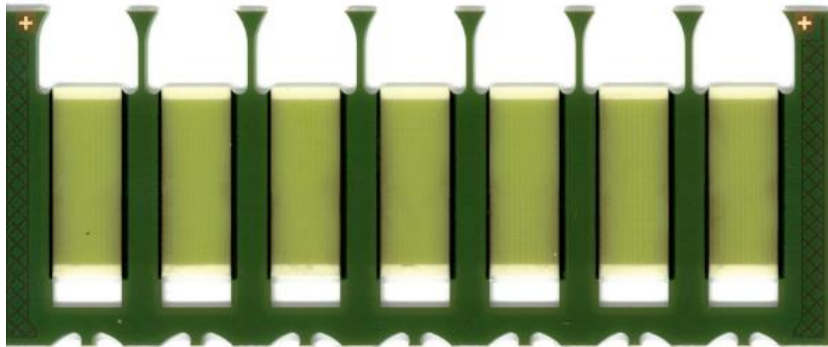
Flex-Rigid Design: Good Examples




Flex-Rigid Design: Good Examples



Flex-Rigid Design: Good Examples



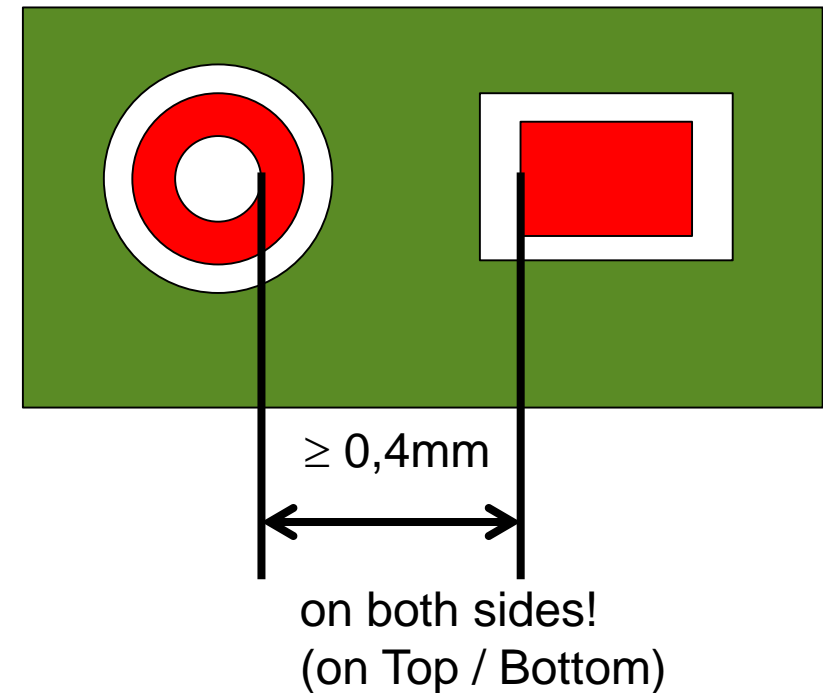
Agenda

- 
- Introduction Flex-Rigid
 - Design Standards, Design Rules
 - Examples, Consequences, Tips
 - Combination of Technologies**
 - Summary, Q&A

In General Layout: Mistakes with Plugging & Filling

■ Design for Via-Plugging (IPC-4761 Type IIIa)

- PTH vacuum sealed
- process: screen printing
- Outgassing leads to contamination on solder pads



• Filling of plated through holes (PTH):

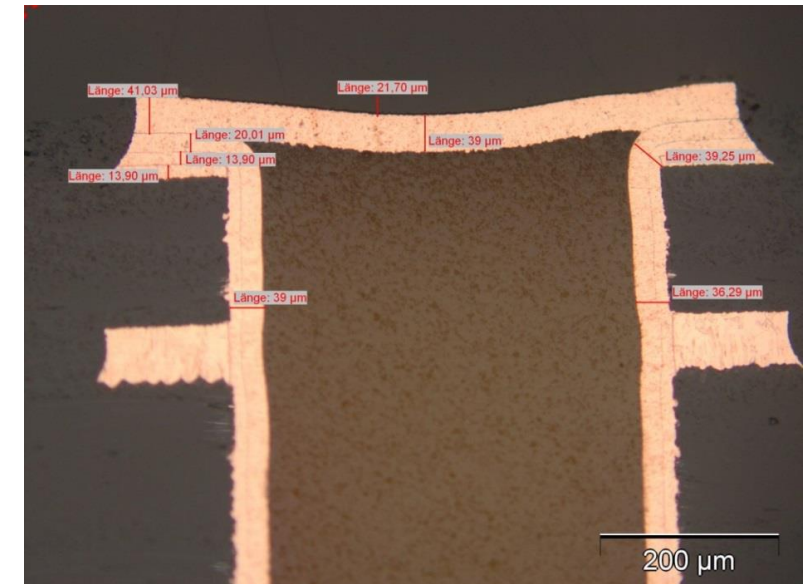
Never use open vias in solder areas! For PTH plugging (IPC Type III) always keep a clearance of 400µm to solder areas on both sides! In case of IPC Type VII (filled and capped) please ask for possible design rules (in special: line space parameters).

In General Layout: Mistakes with Plugging & Filling

- **Design for Via-Filling (IPC-4761 Type VII)**
 - multiple Plating processes increase copper thickness
 - thicker copper = larger design structures (spacing)

- **Limitation of design structures as a consequence of Combination of Technologies und Processes**
 - Microvia copper filling
 - PTH filled & capped
 - AND open PTH !!!!

- **Smallest Structures dependant on combination:**
 - 100µm / 125µm / 150µm



→ in any case please check with your supplier

Summary

- **Knowledge of Design Rules**
 - helps avoiding Mistakes and Costs
 - saves valuable time by avoiding technical queries
 - saves reliable Function of the Application

- **Our Design Rules are revised regularly**
 - please don't use old copies
 - newest Revision always in [Internet](#)

- **In case of questions or suggestions:**



Please contact us!

Emails are welcome via our Team-Address:

flex@we-online.com



Thank you very much for your attention!

