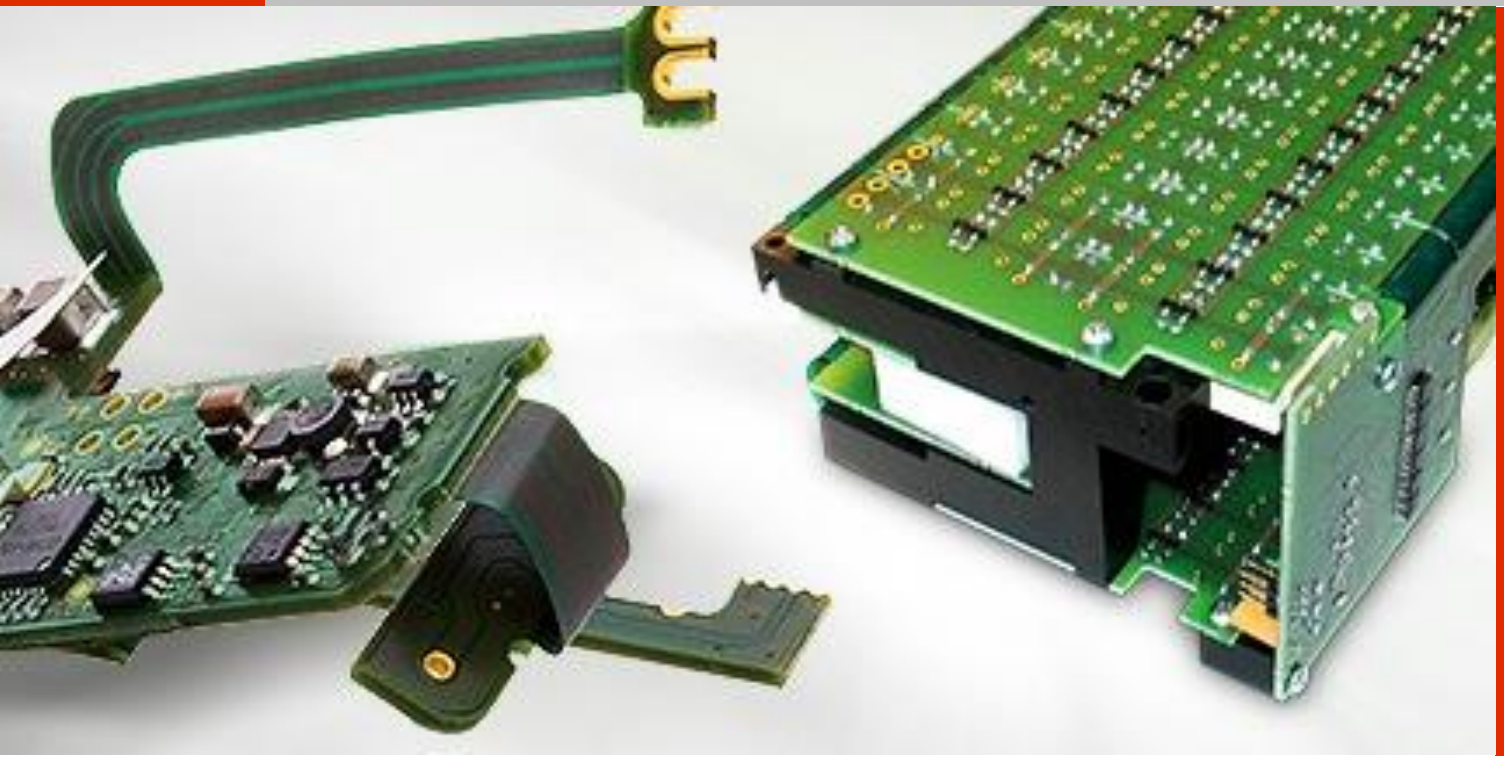
# Flex-rigid: Design mistakes and their effects

## Würth Elektronik Circuit Board Technology

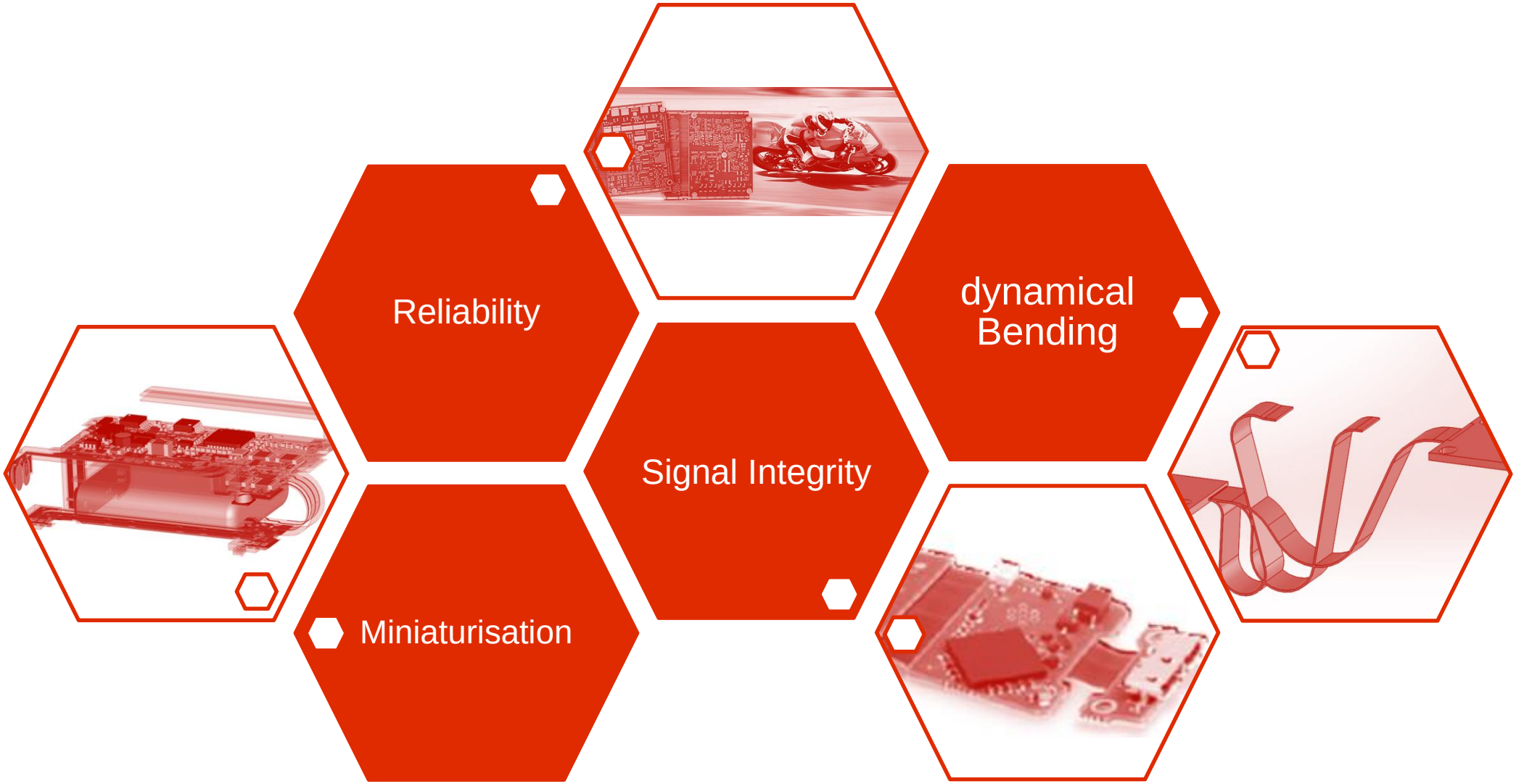


**Webinar December 5, 2017**
**Speaker: Andreas Schilpp**

# Agenda

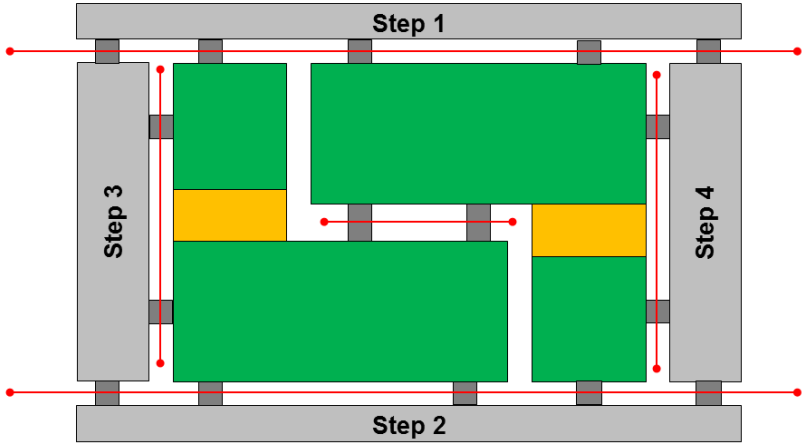
- Introduction Flex-Rigid
- Design Standards, Design Rules
- Examples, Consequences, Tips
- Combination of Technologies
- Summary, Q&A

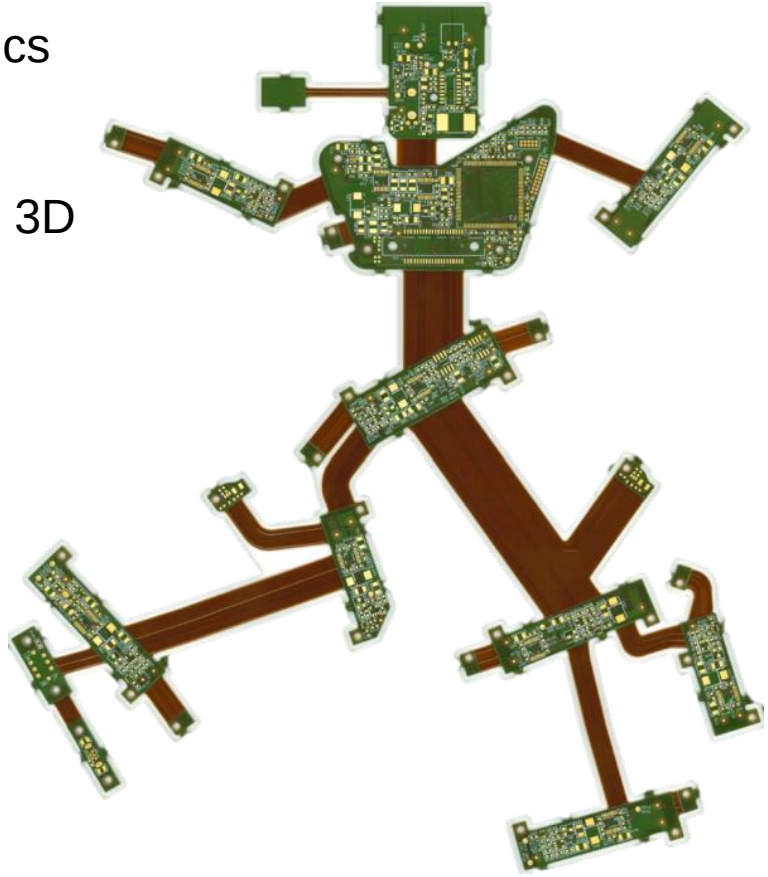
# Introduction: Integration of Module Wiring
## The Advatages of Flex-Rigid



# 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

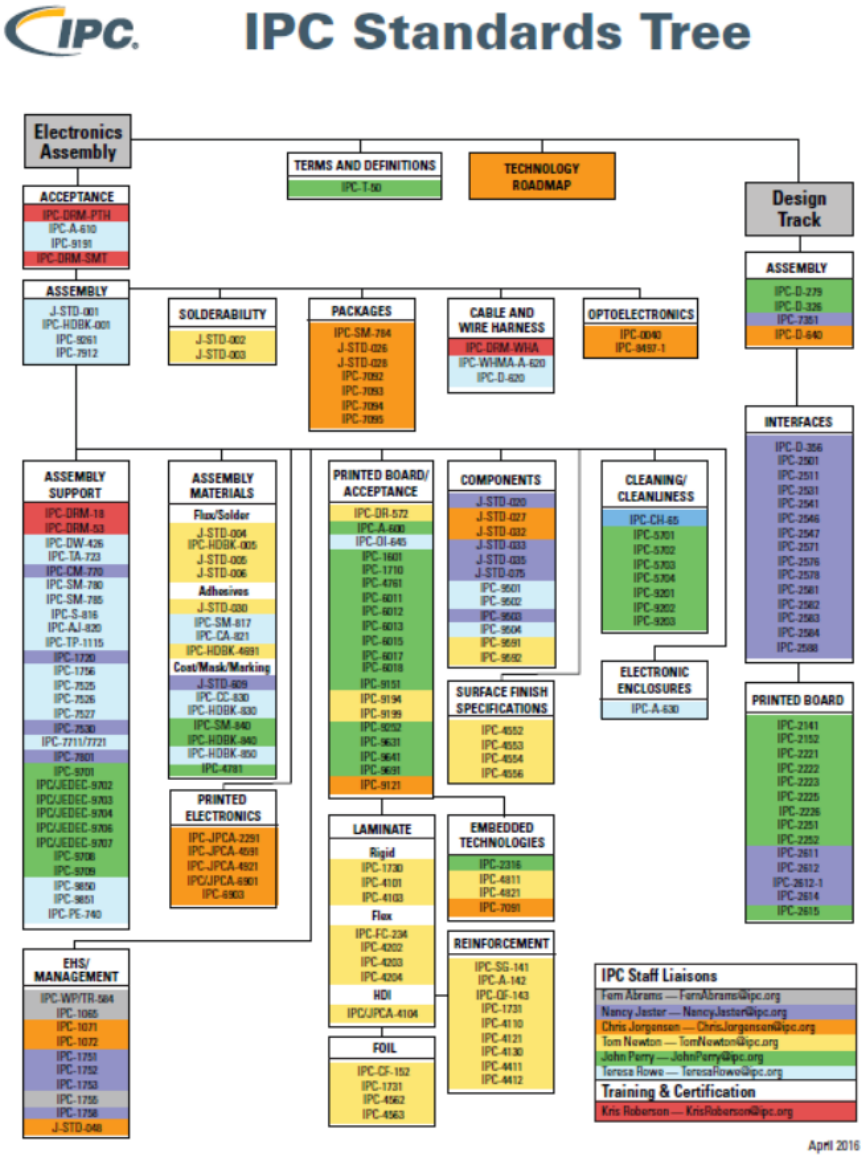
- Introduction Flex-Rigid
- **Design Standards, Design Rules**
- Examples, Consequences, Tips
- Combination of Technologies
- Summary, Q&A

# 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-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.

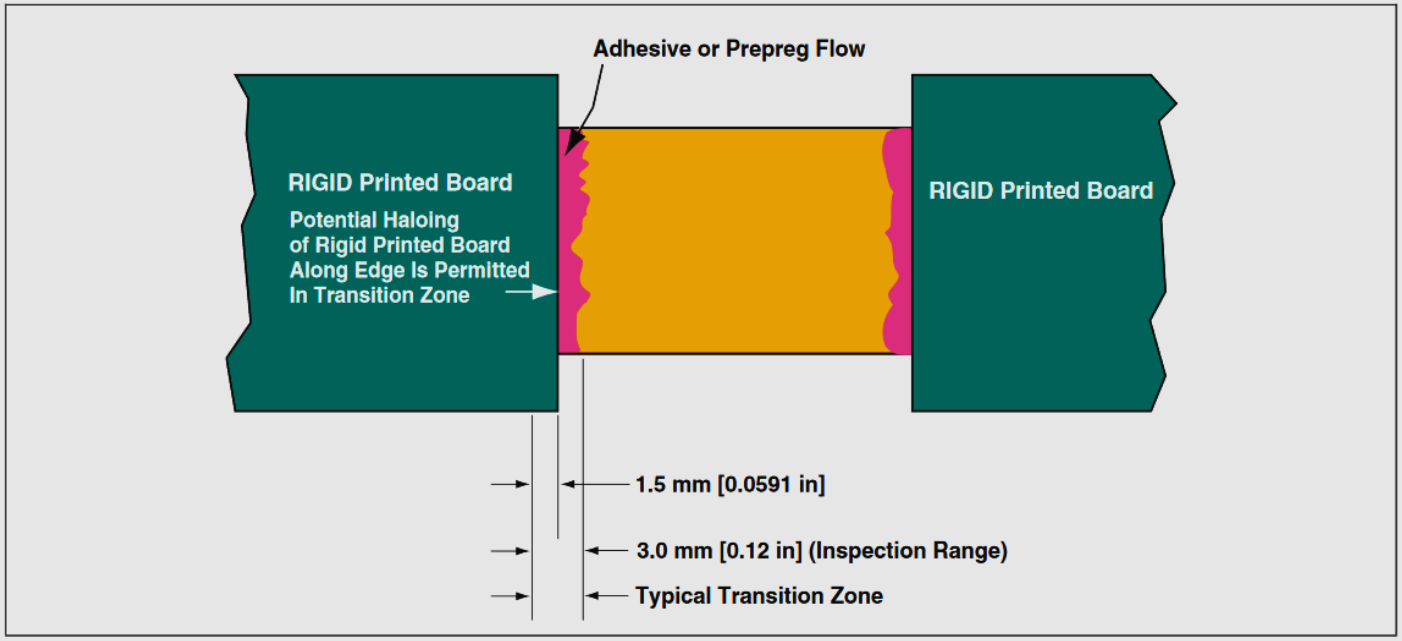



Figure 416a

# IPC-6013C - Definition



**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.

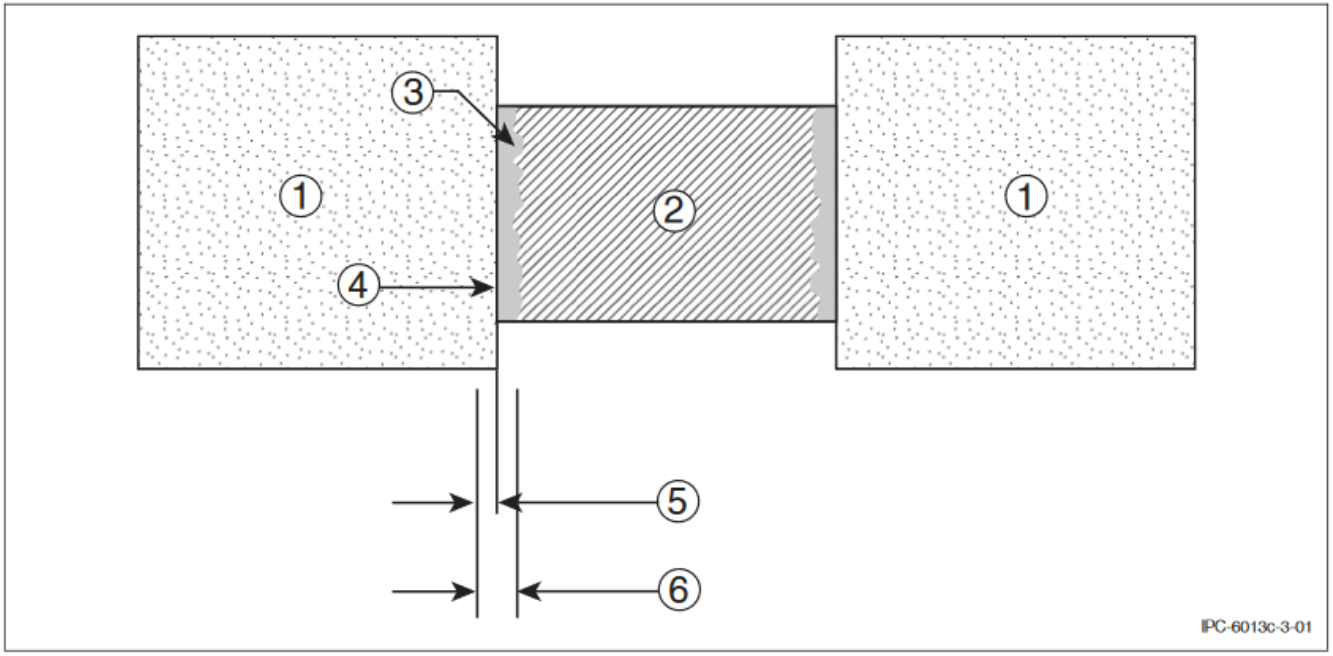



**Figure 3-1 Transition Zone**
**Note 1.** Rigid Printed Board
**Note 2.** Flex Circuit
**Note 3.** Adhesive or Prepreg Flow
**Note 4.** Potential Haloing of Rigid Printed Board along Edge is permitted in Transition Zone
**Note 5.** 1.5 mm [0.060 in]
**Note 6.** Limit of Inspection Range (3.0 mm [0.118 in]), centered on the Transition Zone.

**Grey Area**

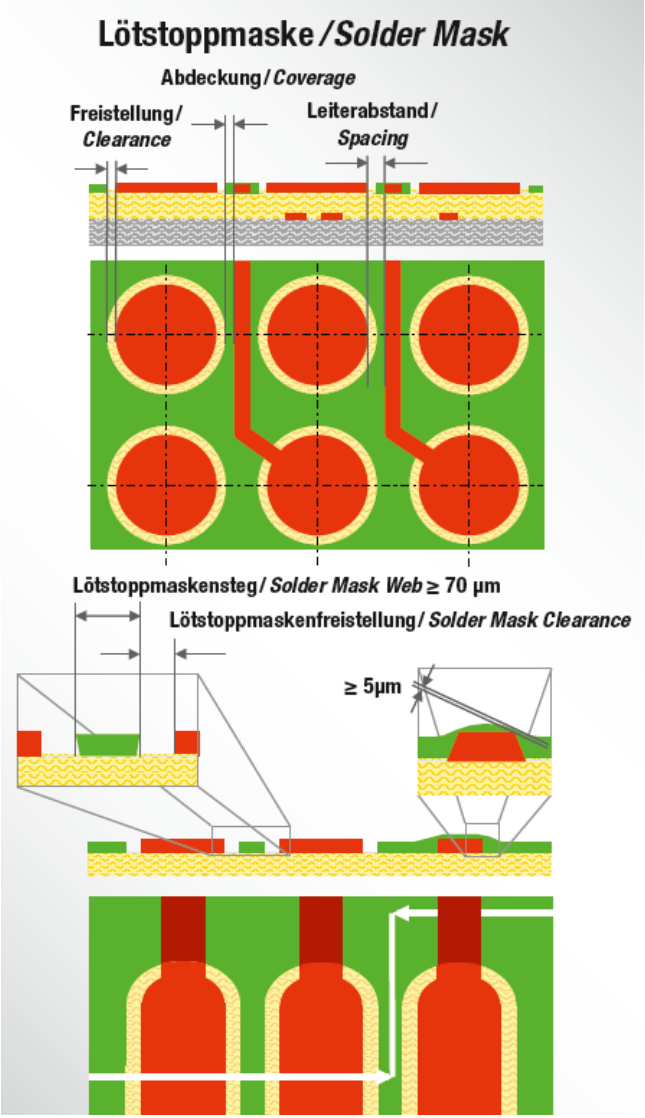
## Hierarchy WE Design Guides / Design Rules

# Basic Design Guide

| | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|
| **sect-ional Design Guides** | HDI | TM | Wire bonding | Power | 3D | | | | | |
| **Design Rules** | **HDI** | **TM** | **Wire bonding** | **Wirelaid** | **Semiflex** | **Flex / TWINflex** | **1F-xRi** | **xRi -≥2F- xRi** | **2F-xRi** | |

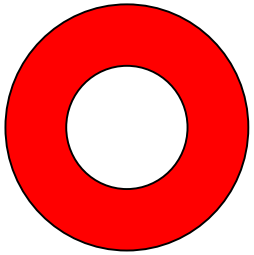
# Basic Design Guide

- **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 spcace values
  - Soldermask Web and Clearance
  - Legend Print
  - Copper Clearance to
    - milled / V-scored contour
    - NPTH

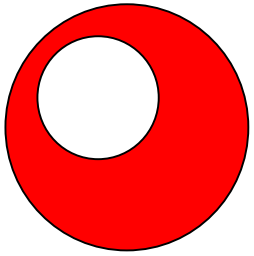


# Sense of Annular Rings

- **Layout / Screen:**



- **Real life**

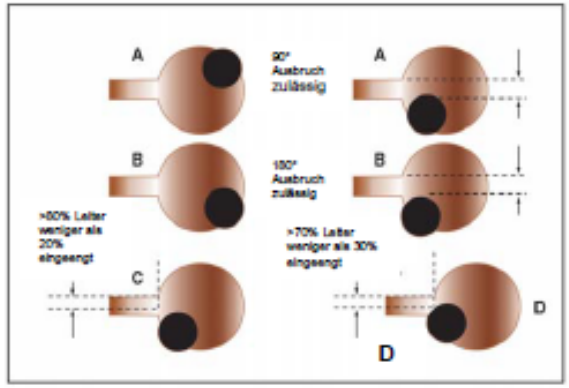


**IPC-A-600H:**

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**

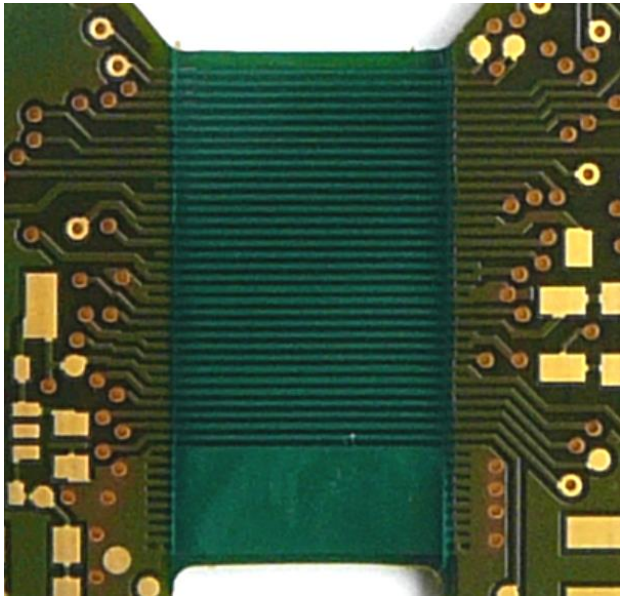
- Abweichungen, die gegen obige Kriterien verstoßen.

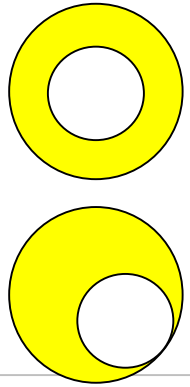
# Design Rules Flex-Rigid – Spacing Vias / Pads

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

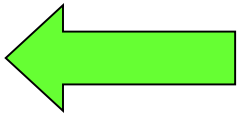
IPC-2223:

– **Spacing Vias to Flex-Rigid Transition**





**Distance to the Via Pad!**



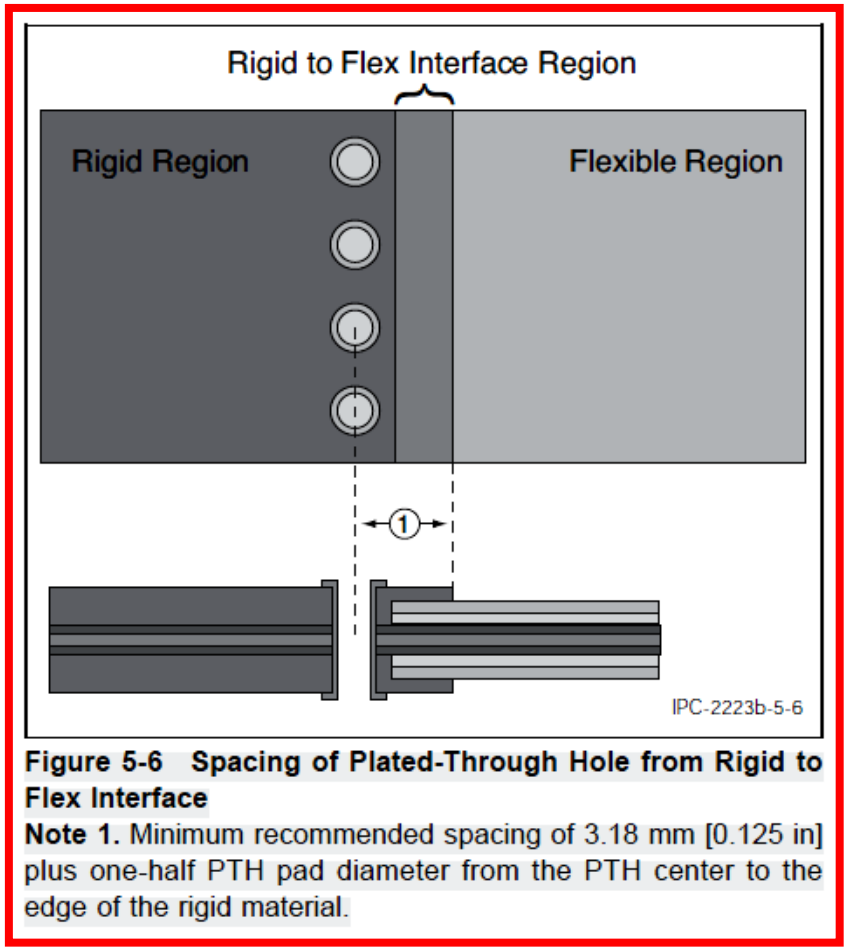


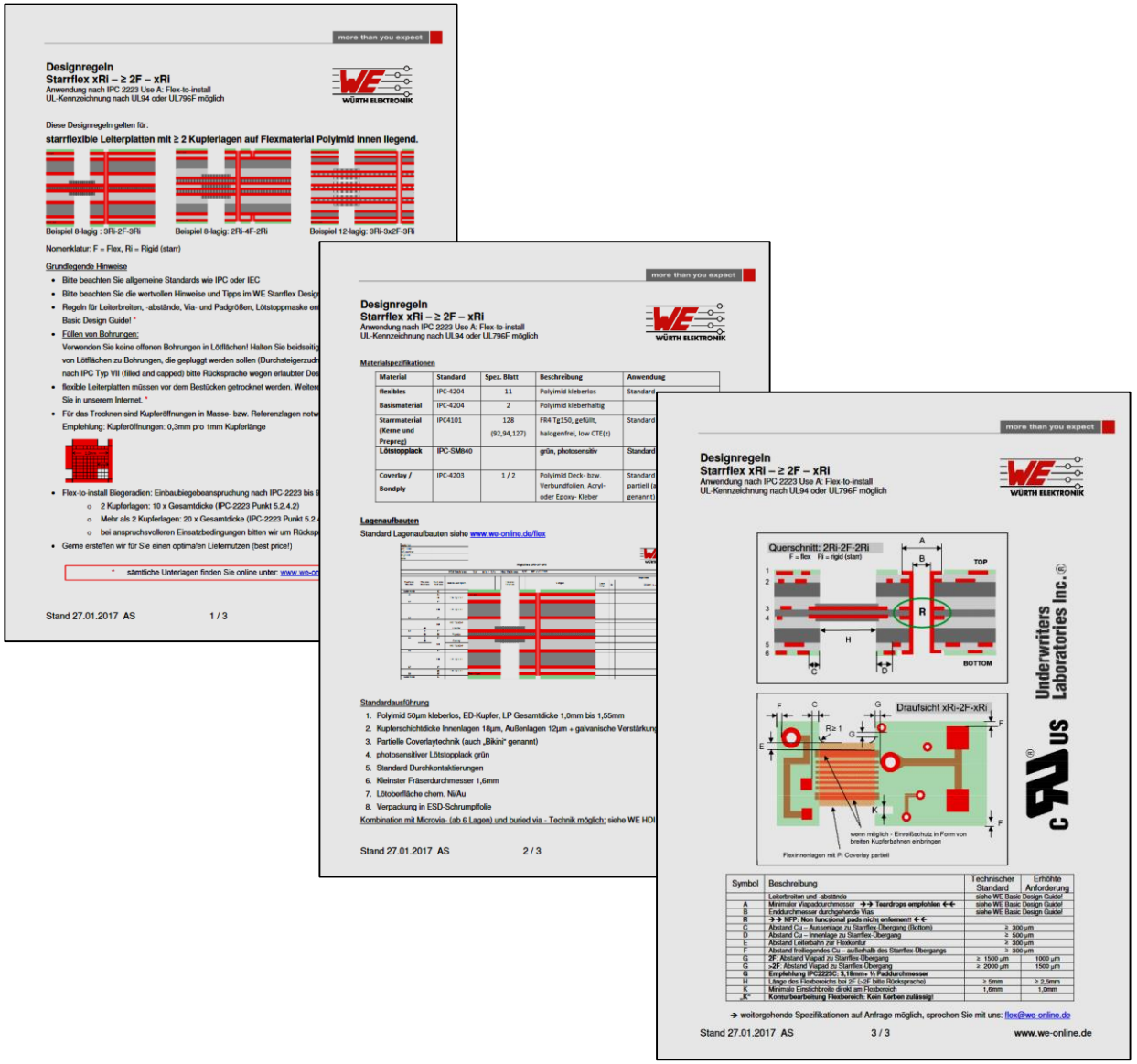

**Figure 5-6 Spacing of Plated-Through Hole from Rigid to Flex Interface**

**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

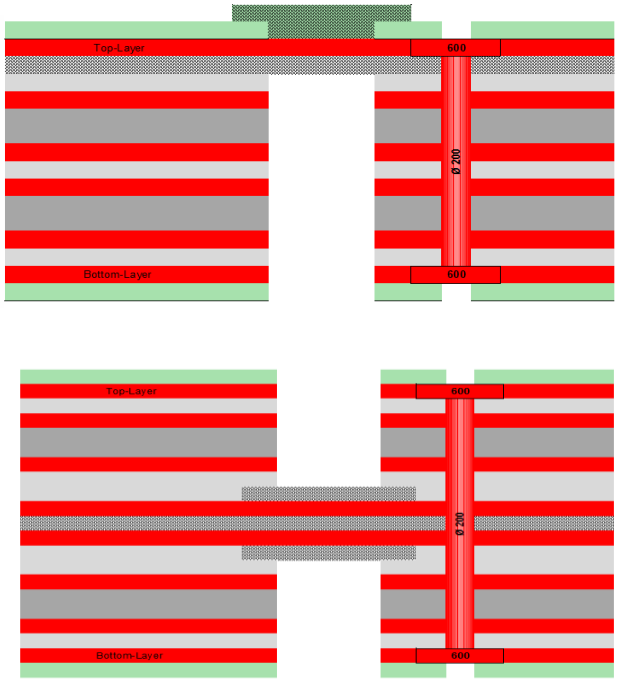
- Application definition, stack-up examples
- Basic Information
- *Bending Radii*
- Processing Conditions
- Material Specificatons
- Link to Standard Stack-up´s
- Definition Standard
- specific Layout Parameters

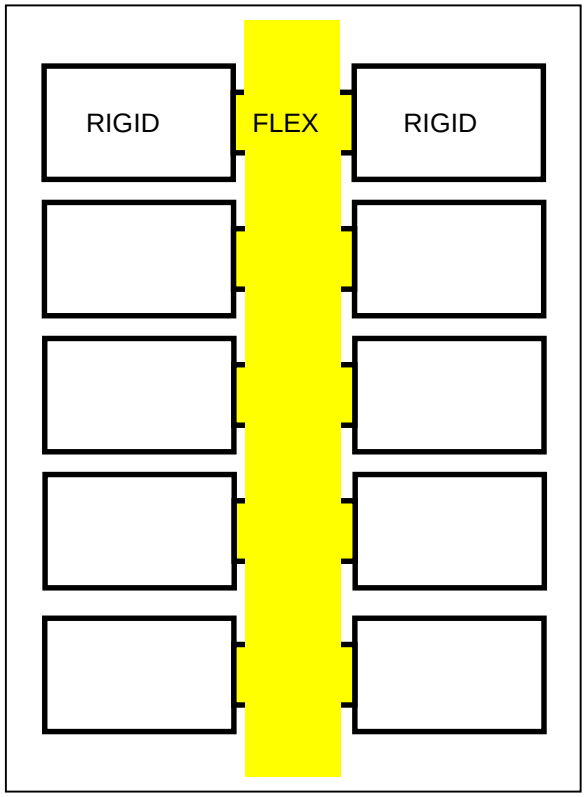
- additionally for Semiflex:
  - Tips for Bending and Fastening

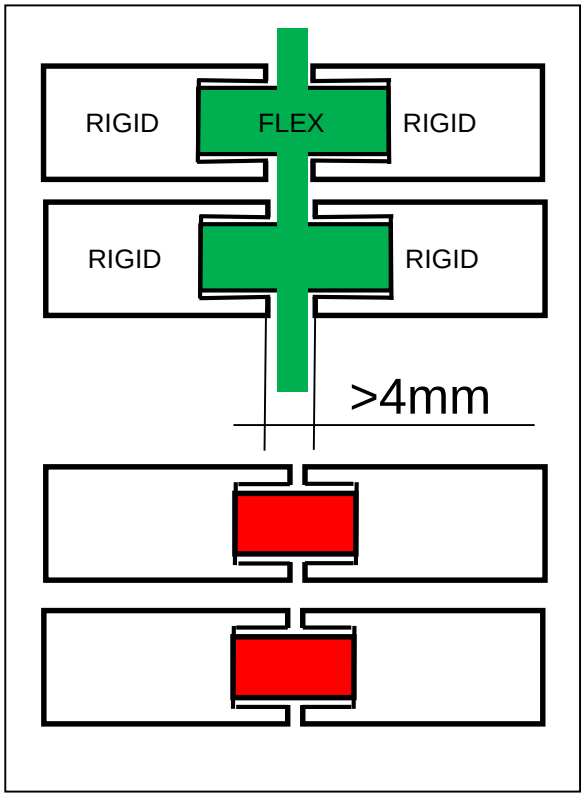


# Design Flex Area: Application Coverlay Folie

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

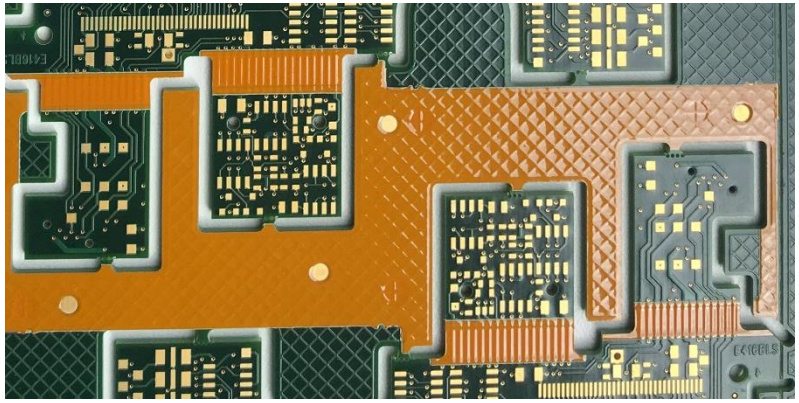


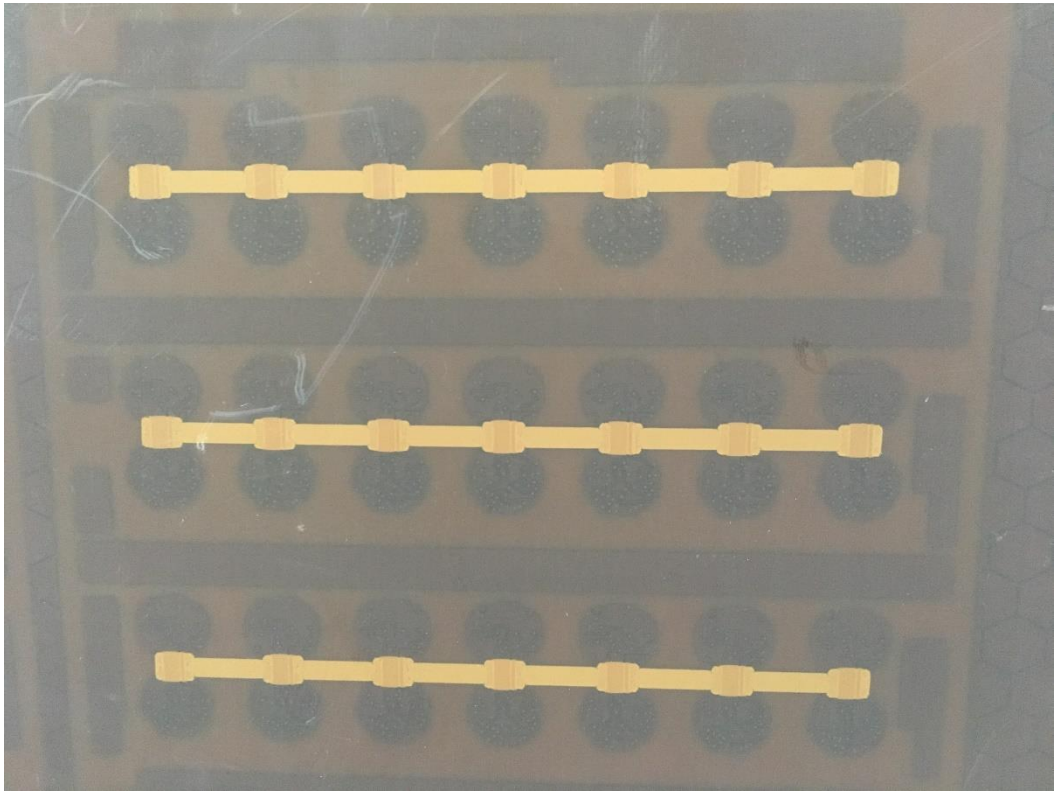



# Design Flex Area: Application Coverlay Folie

- **Examples:**





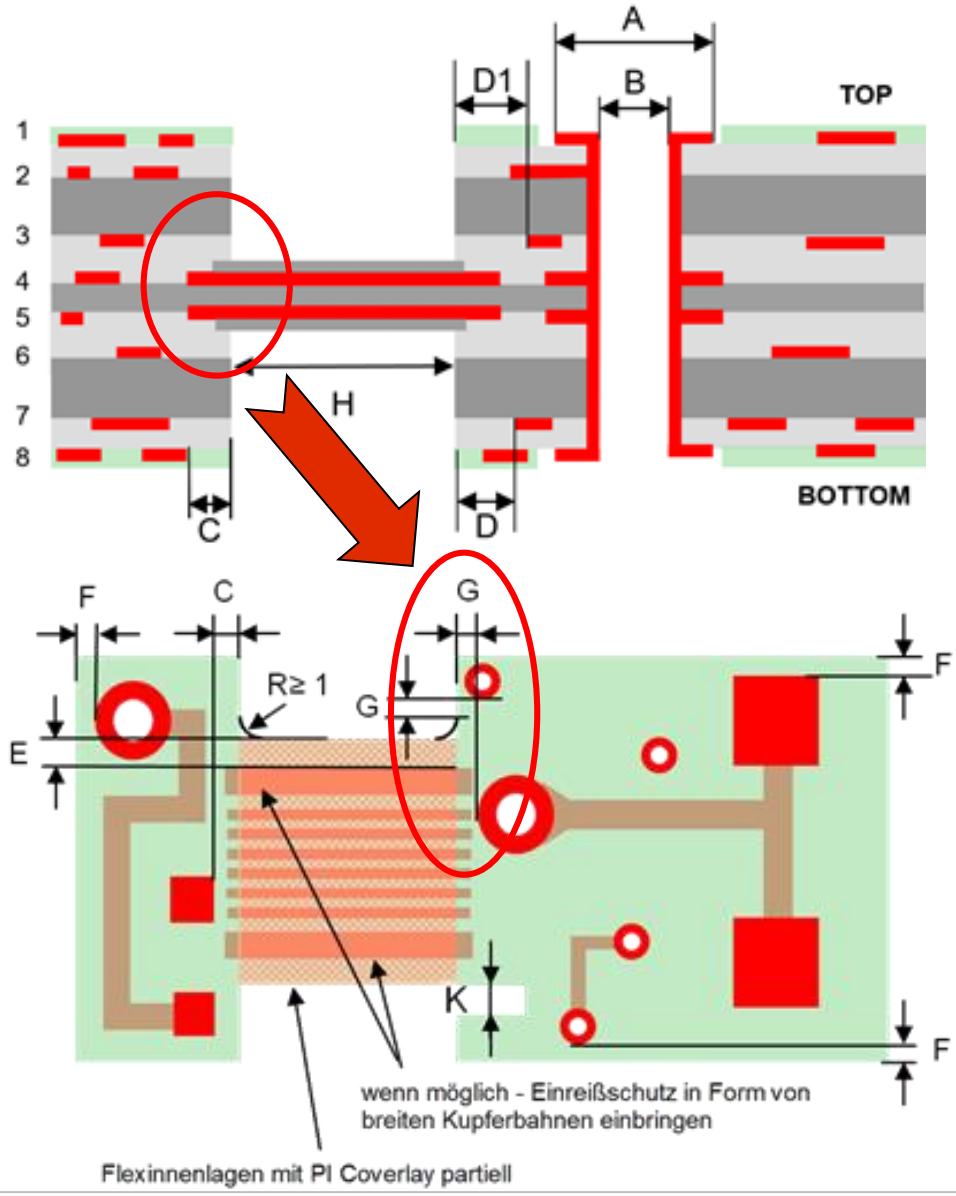
# Design Rules

- **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+ ½ Pad Diameter**

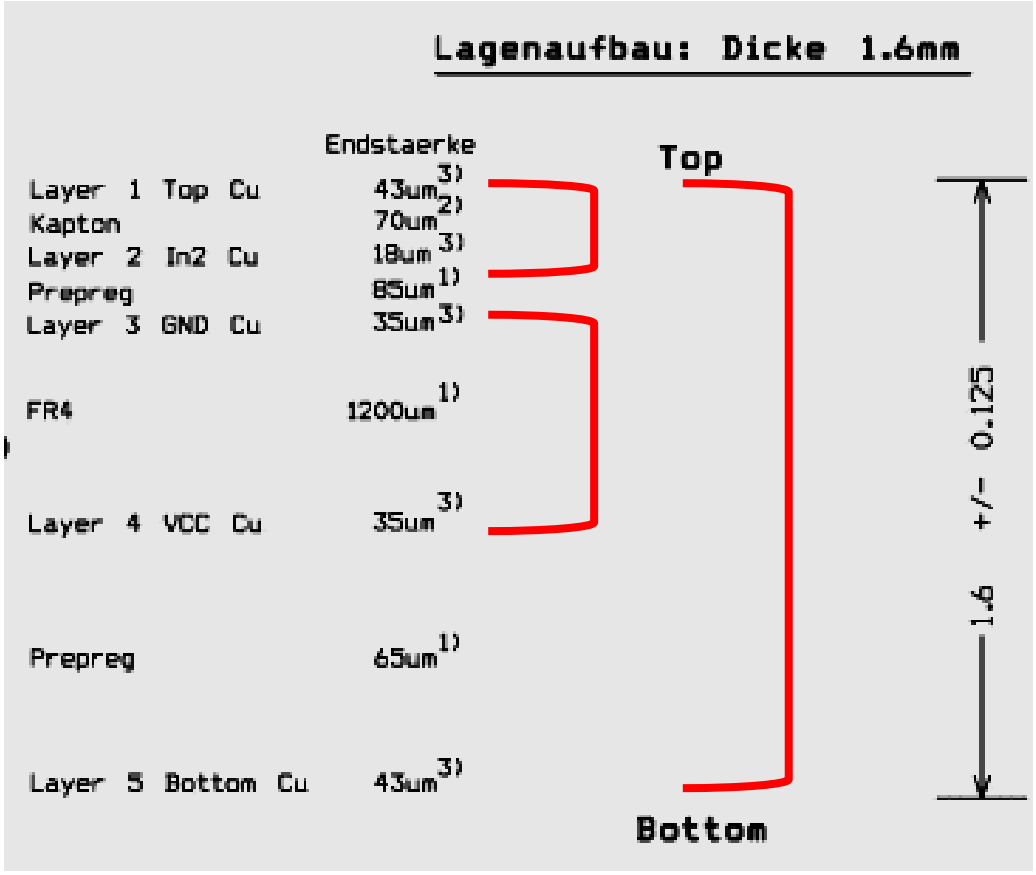


# Agenda

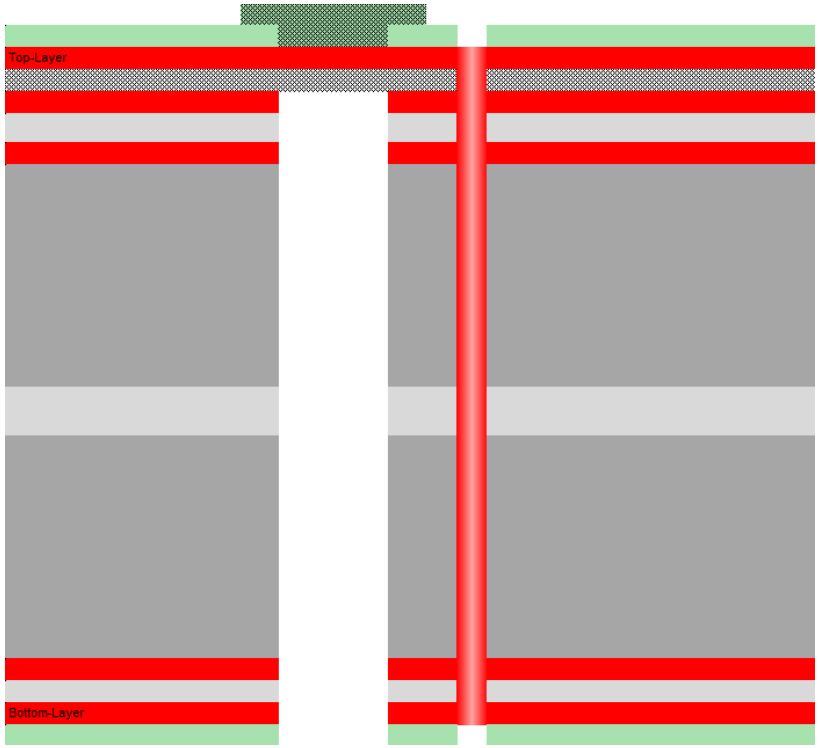
- Introduction Flex-Rigid
- Design Standards, Design Rules
- **Examples, Consequences, Tips**
- Combination of Technologies
- Summary, Q&A

# Flex-Rigid Stack-up: Mistake 1

- **Excerpt of PCB drawing:**




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

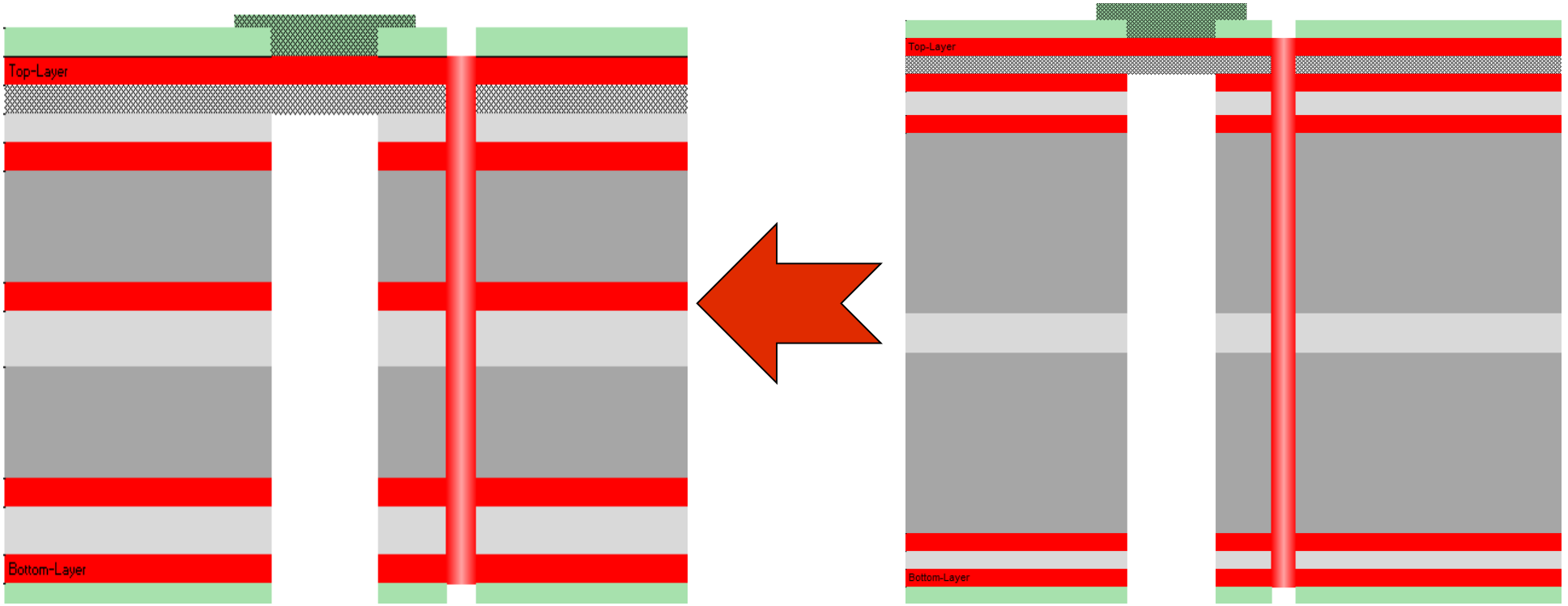



- **BUT: no copper Layout on Layer 2 of flex core!**

# Flex-Rigid Stack-up: Mistake 1

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

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

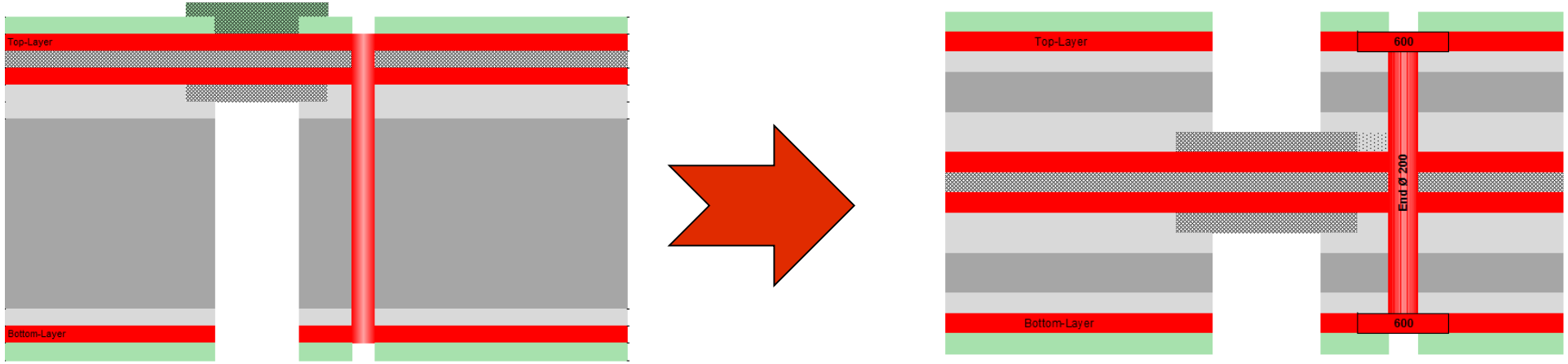



- Cost Savings: about 30% with same Performance!
- at the same time reduced Risk of Warpage!

# 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!**

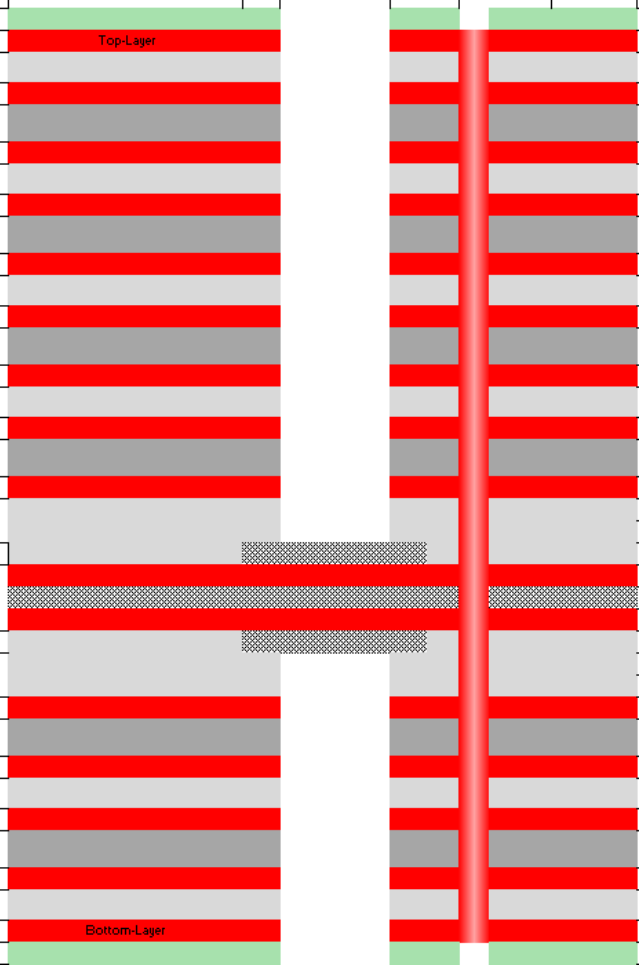
# Flex-Rigid Stack-up: Mistake 3

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

**Rigidflex 9Ri-2F-5Ri**

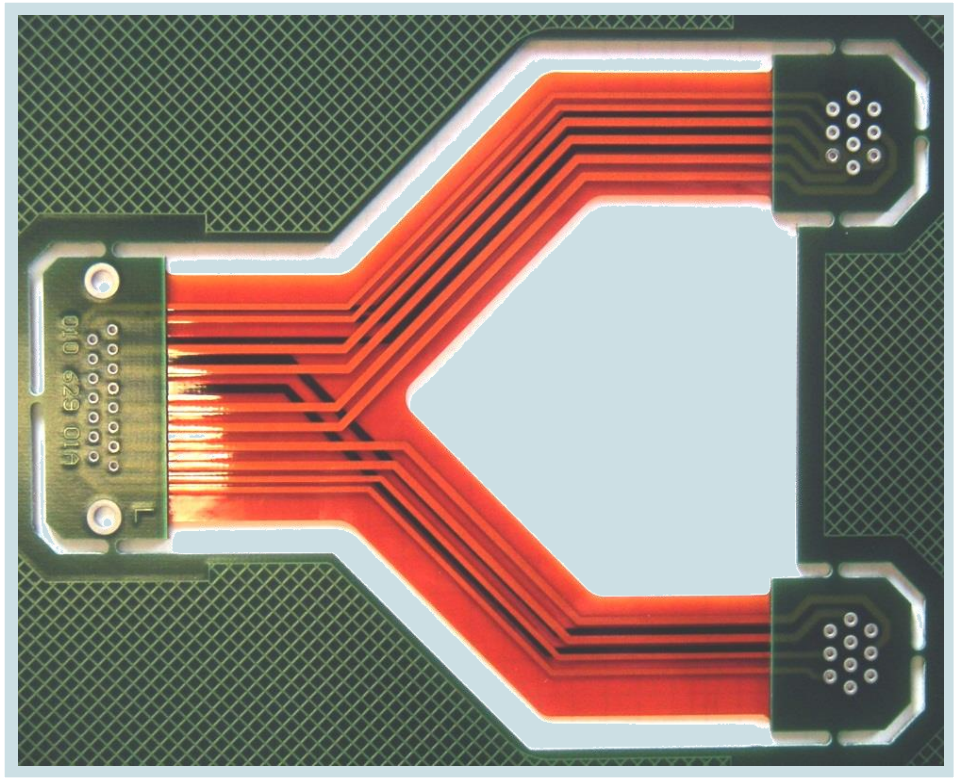
PCB Thickness : 2,81 mm +/- 10% Flex Thickness: 0,20 mm +/- 0,05mm

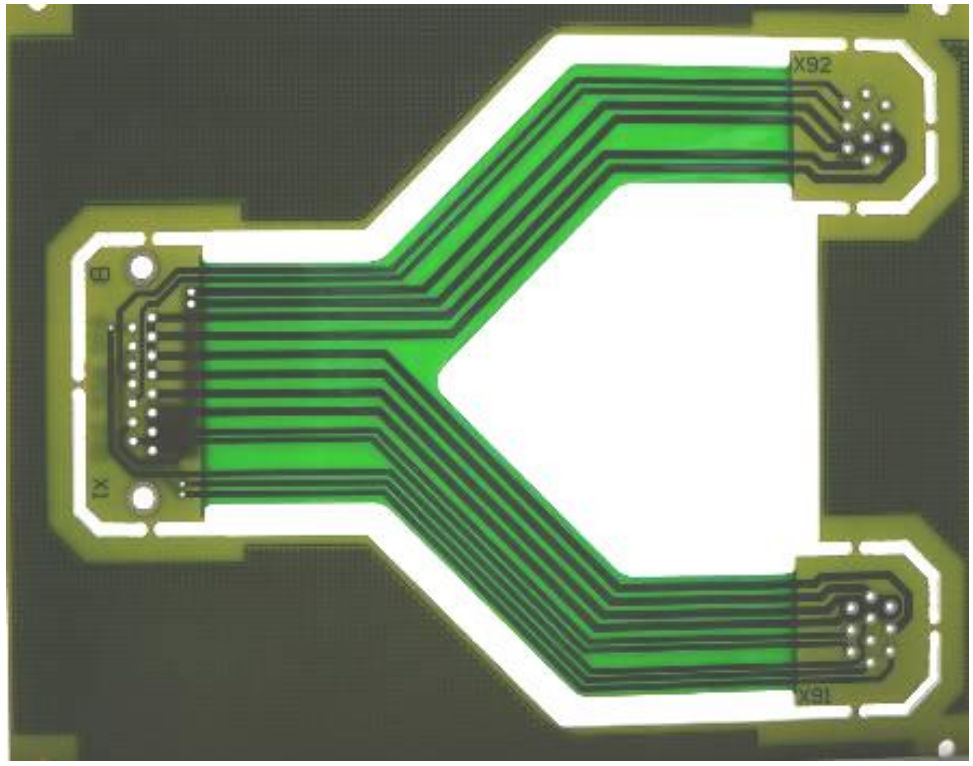
| Rigid area Structure | Flex area Thickness | Rigid area Thickness | Material describing |
|---|---|---|---|
| Soldermask | | 12 | |
| L1 | | 35 | |
| Prepreg | | 360 | 2 x 7628 |
| L2 | | 35 | |
| Core | | 100 | HTG 170° |
| L3 | | 35 | |
| Prepreg | | 125 | 2 x 1080 |
| L4 | | 35 | |
| Core | | 100 | 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 | 40 | 180 | Coverlay |
| | | | 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 | |
| Soldermask | | 12 | |

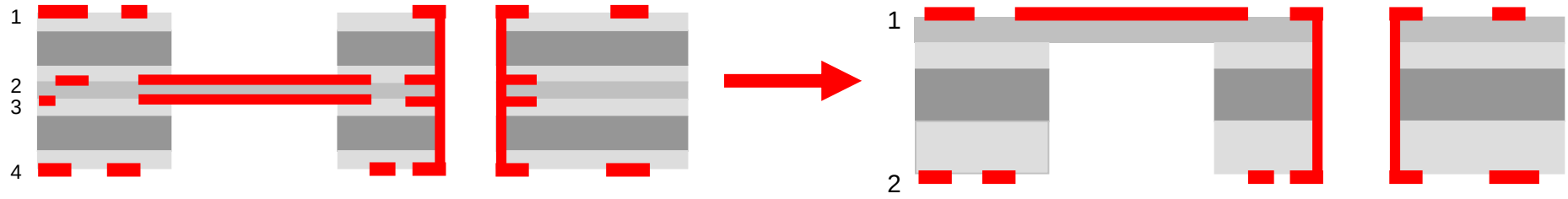


# Flex-Rigid Stack-up: Mistake 4

2 flexible layers inside → 1 flexible layer outside

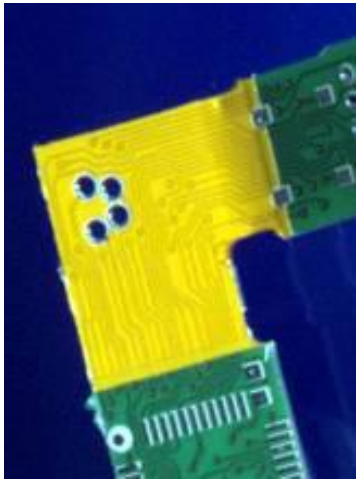


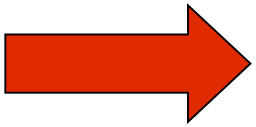


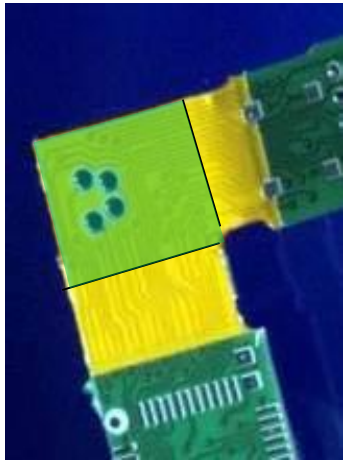



# Flex-Rigid Layout: PTH

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





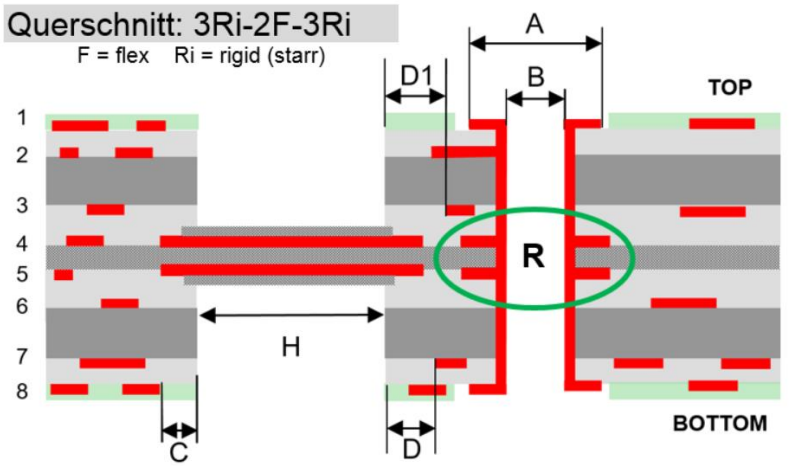


➔ *Avoid Vias in Flex in general !*

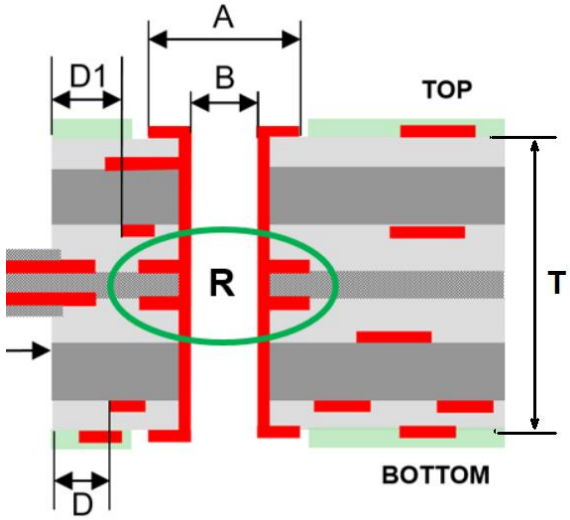
# Flex-Rigid Layout: PTH

- **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“**

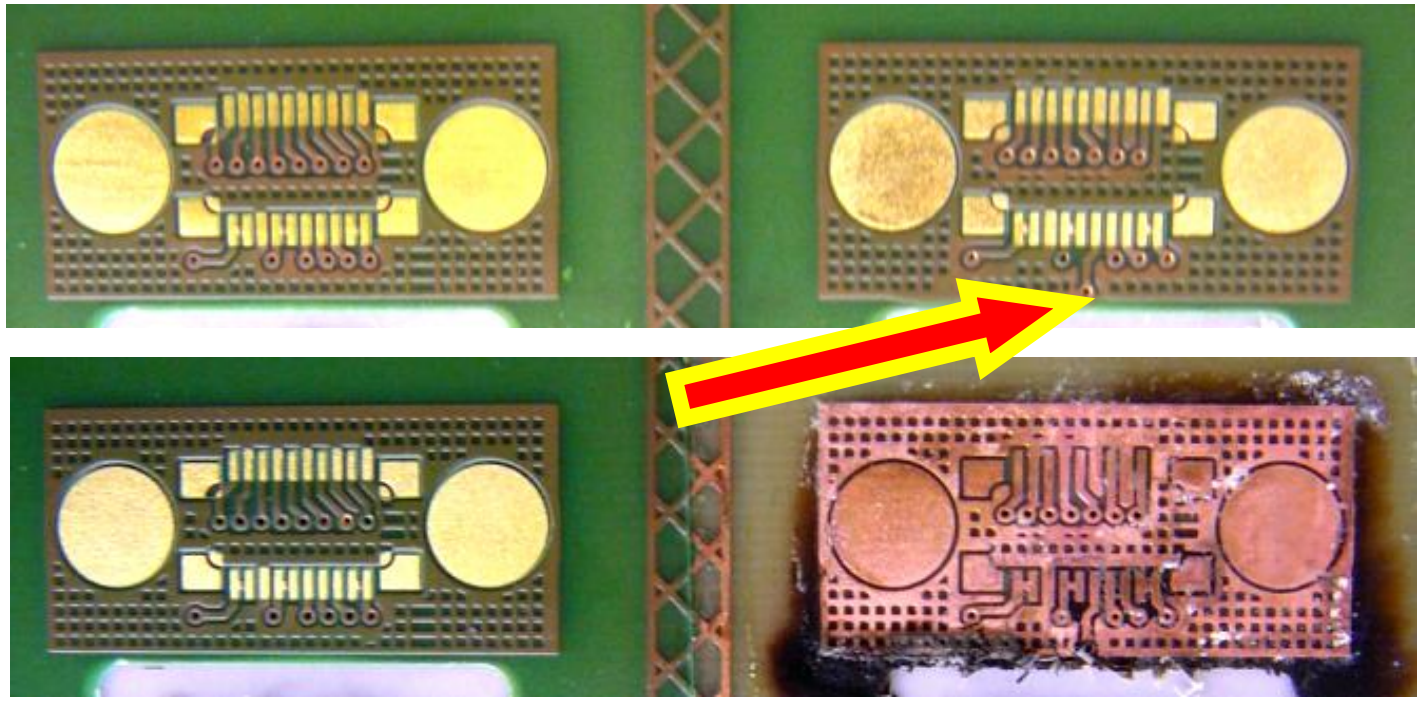


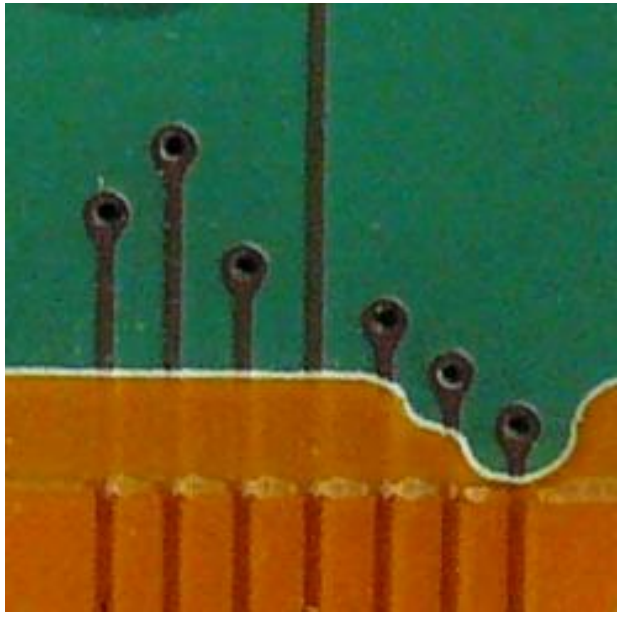

- **Mistake 3: AR Aspect Ratio too large (PCB Thickness T / PTH-Diameter B)**
  - **reduced copper distribution in barrel**
  - **reduced barrel stability, current capacity and heat transfer**
  - **Risk of barrel cracking in case of thermal load**
  - **for Flex-Rigid with Flex core inside even more problematic**



# Flex-Rigid Layout: PTH

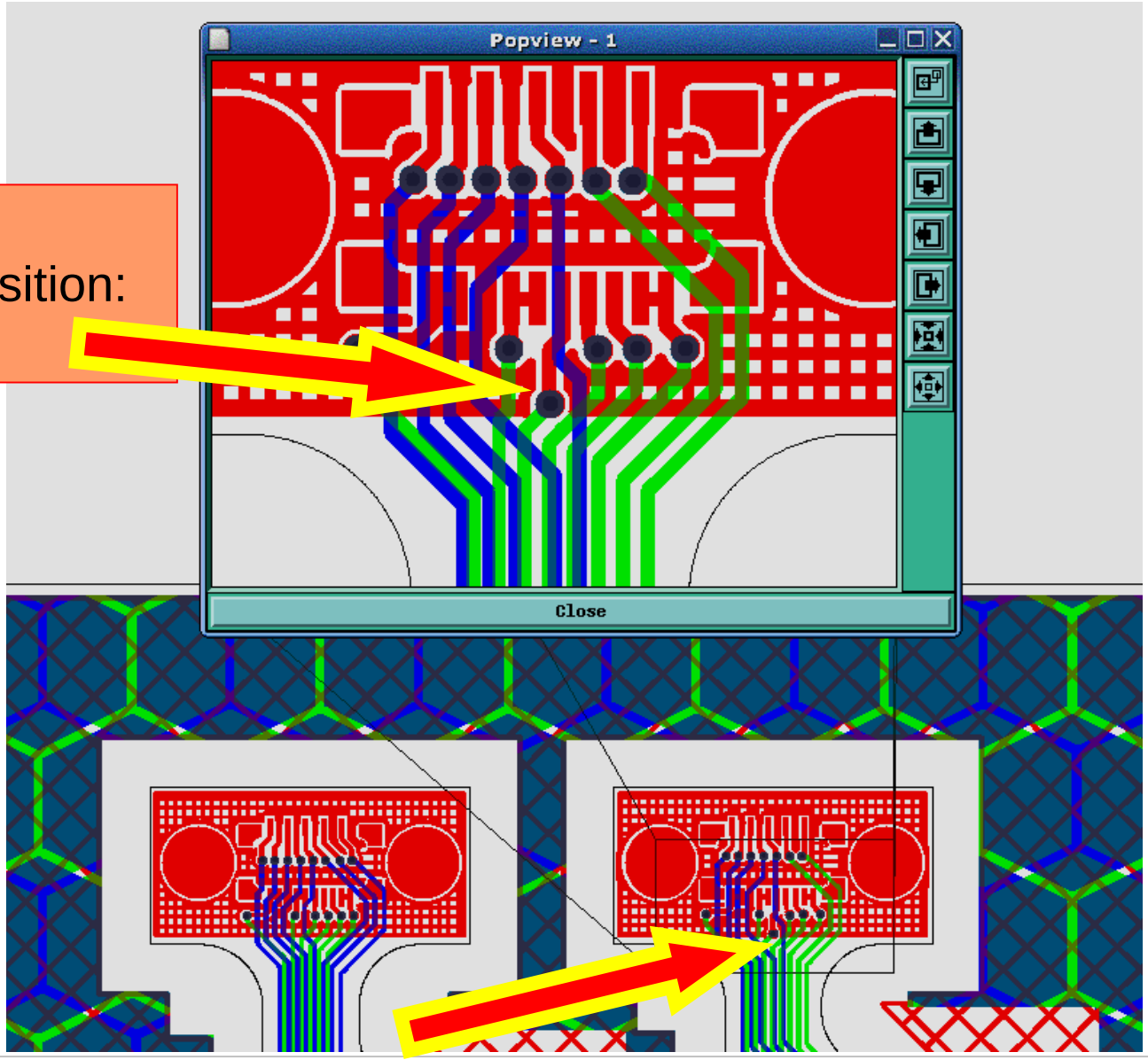
- **Mistake 4: Spacing PTH to Flex-Rigid Transition**
  - **Risk of Scrap during Production and**
  - **Risk of Failure in the Field in worst case**

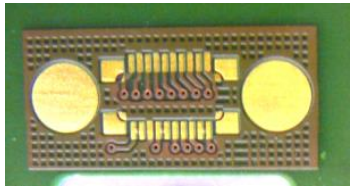


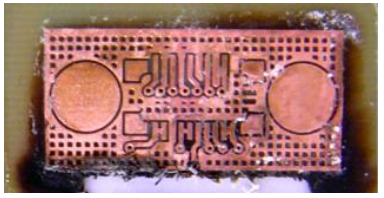


# Flex-Rigid Layout: PTH

Spacing
PTH wall – Transition:
**0,45mm**

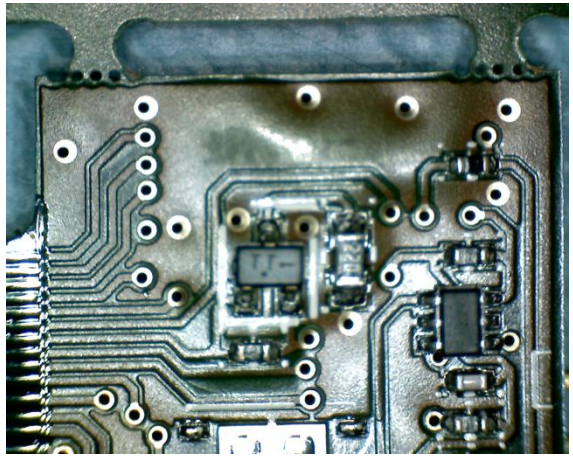


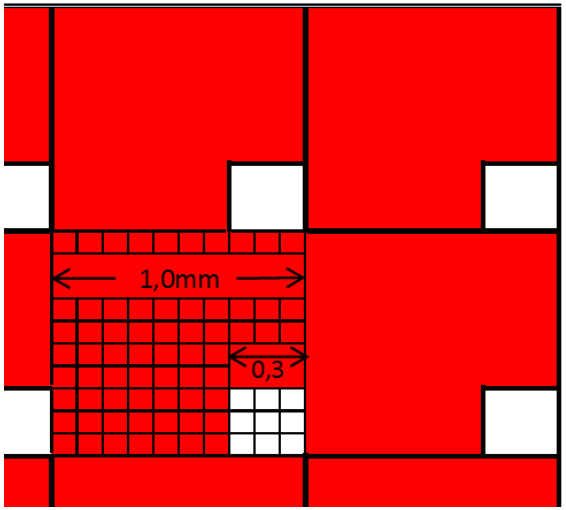


# Flex-Rigid Design: Copper Pattern Mistake 1

- **Problem: large unpierced copper areas**



- **Correction: shield opening**




**<u>Other possible reasons for Delamination</u>:**

- **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

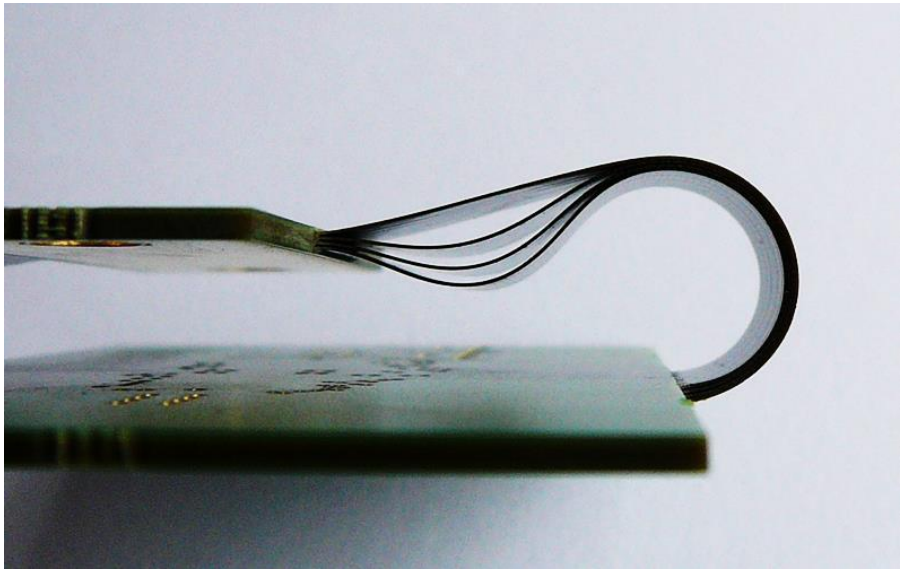
- Airgap-Stack-up
  - Flex Length „H“ too small
  - „Buckling“ makes flexible area very stiff
- Recommendation:
  - H ≥ 50mm

| | |
|---|---|
| customer | |
| pcb | |
| WE-number | |
| engineer | W. Öchsien |
| date | 15.04.2014 |

Rigidflex 1Ri-2F+2F+2F-1Ri

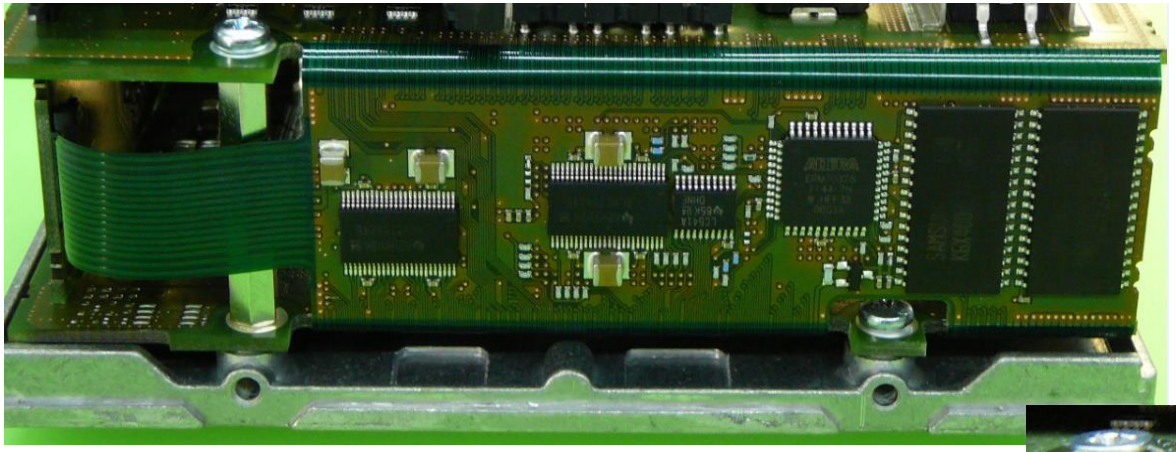
PCB Thickness : 2,02 mm +/- 10% Flex Thickness (each): 0,29 mm +/- 0,05mm

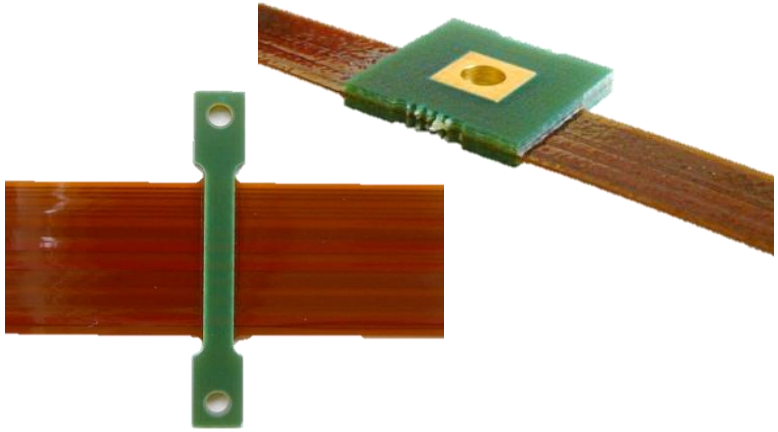
| Rigid area Structure | Flex area Thickness | Rigid area Thickness | Material describing | Flex area Structure | Viatypes | Layer usage | Impedance Er | Impedance Z[Ohm] / Line / Space |
|---|---|---|---|---|---|---|---|---|
| Soldermask | | 20 | | | | | | |
| L1 | | 45 | | Top-Layer | | | | |
| Prepreg | | 65 | FR4 HTG 150* | | | | | |
| Core | | 250 | FR4 HTG 150* | | | | | |
| Prepreg | | 150 | FR4 HTG 150* | | | | | |
| | 50 | | Coverlay LF0210 | | | | | |
| L2 | 70 | 70 | | | | | | |
| Flex Core | 50 | 50 | Polyimid AP9222R | | | | | |
| L3 | 70 | 70 | | | | | | |
| Prepreg | 50 | 200 | Coverlay LF0210 | | | | | |
| | | | FR4 HTG 150* | | | | | |
| | 50 | | Coverlay LF0210 | | | | | |
| L4 | 70 | 70 | | | | | | |
| Flex Core | 50 | 50 | Polyimid AP9222R | | | | | |
| L5 | 70 | 70 | | | | | | |
| Prepreg | 50 | 200 | Coverlay LF0210 | | | | | |
| | | | FR4 HTG 150* | | | | | |
| | 50 | | Coverlay LF0210 | | | | | |
| L6 | 70 | 70 | | | | | | |
| Flex Core | 50 | 50 | Polyimid AP9222R | | | | | |
| L7 | 70 | 70 | | | | | | |
| Prepreg | 50 | 140 | Coverlay LF0210 | | | | | |
| | | | FR4 HTG 150* | | | | | |
| Core | | 250 | FR4 HTG 150* | | | | | |
| Prepreg | | 65 | FR4 HTG 150* | | | | | |
| L8 | | 45 | | Bottom-Layer | | | | |
| Soldermask | | 20 | | | | | | |



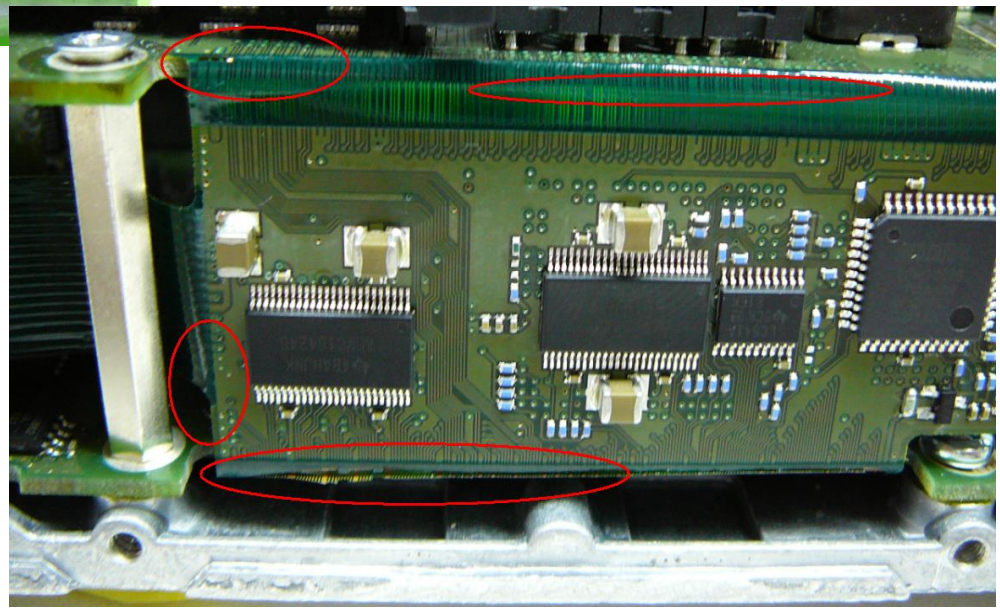
# Flex-Rigid Mechanics: Mistake 2

**NO fixation of one rigid area**



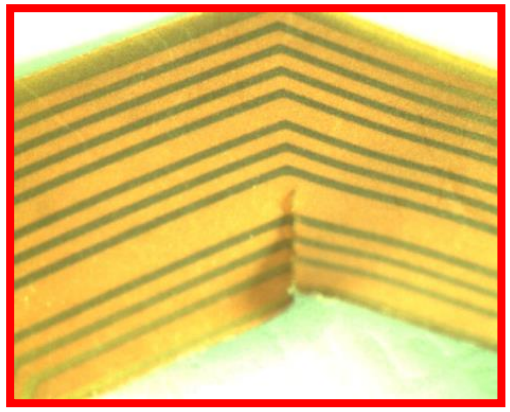


**… leads to resonance and destruction even with Rigid-Flex!**

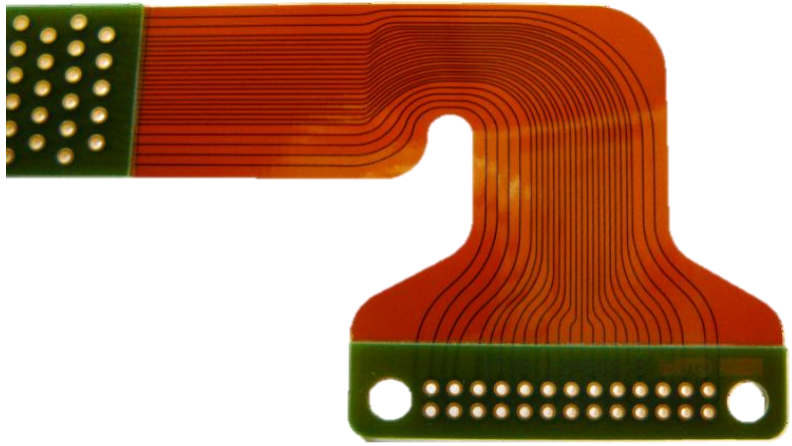


# Flex-Rigid Design: Mistake Contour

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



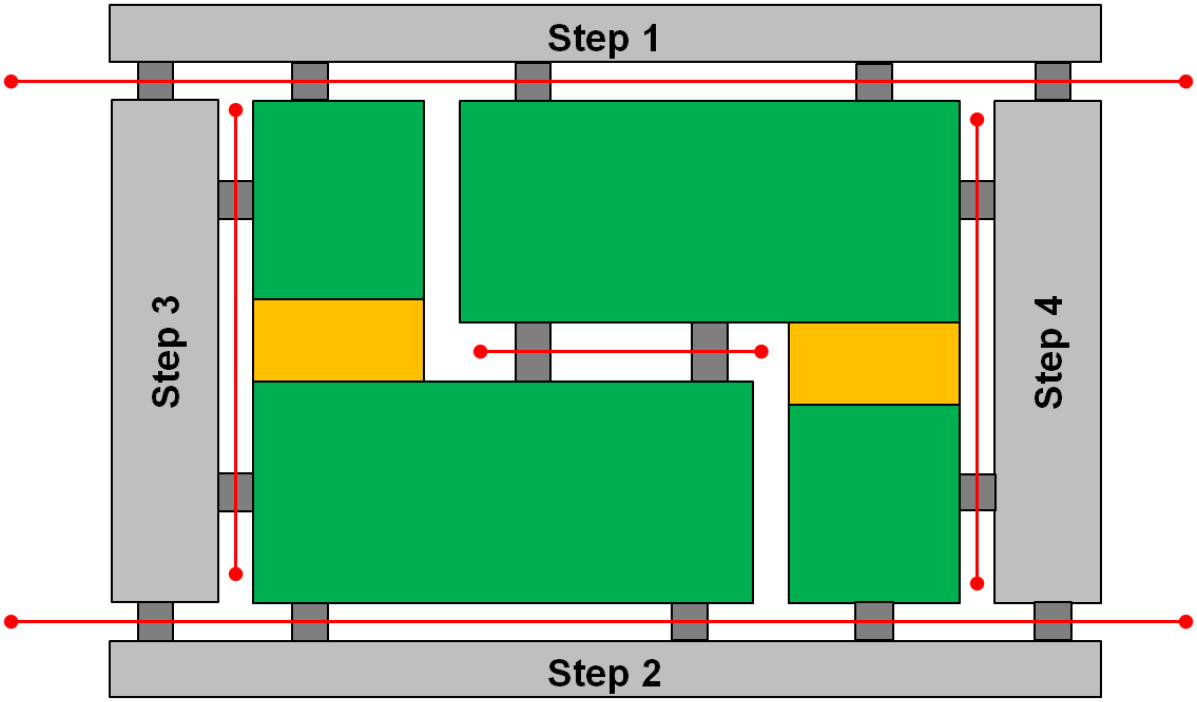
➔ **Design for plastics!**



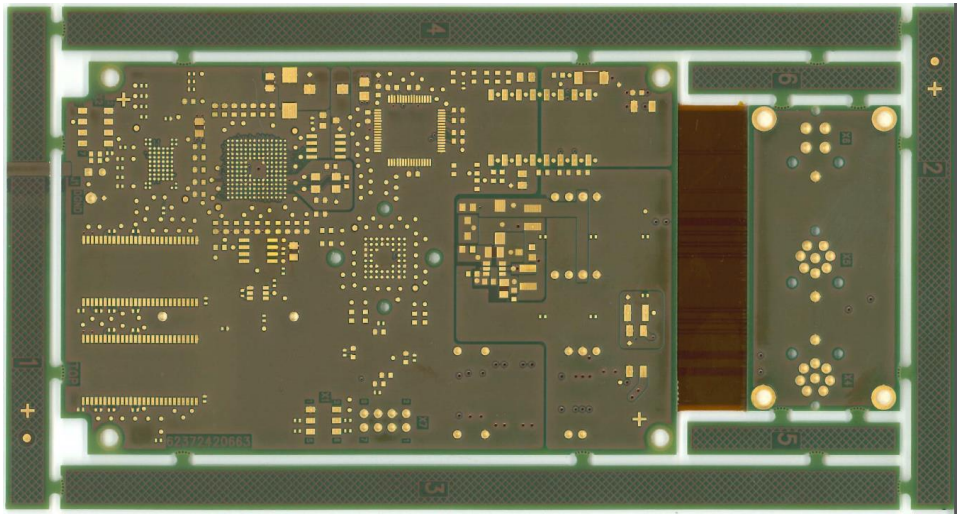
**A good example!**

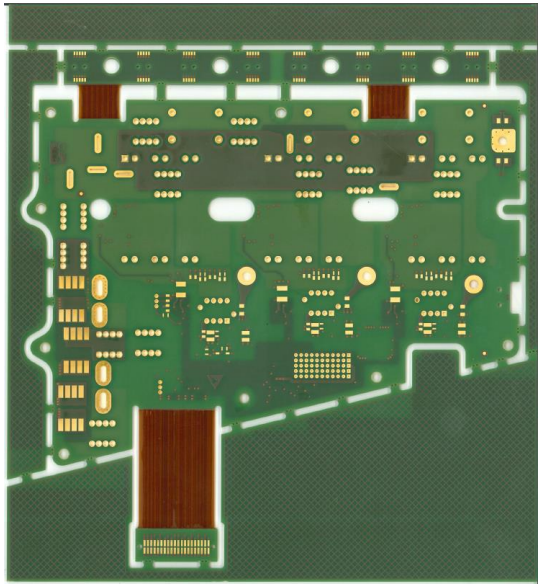
# Flex-Rigid Design:  Mistake Delivery Array

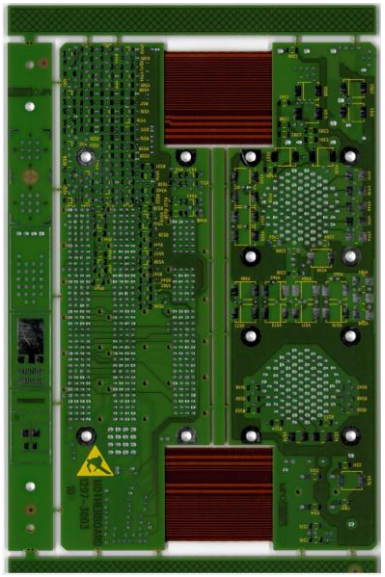
- Array too solide, hard to separate
- Missing of break points
- circulating stable frame
  - Damaging of flexible Area during Separation

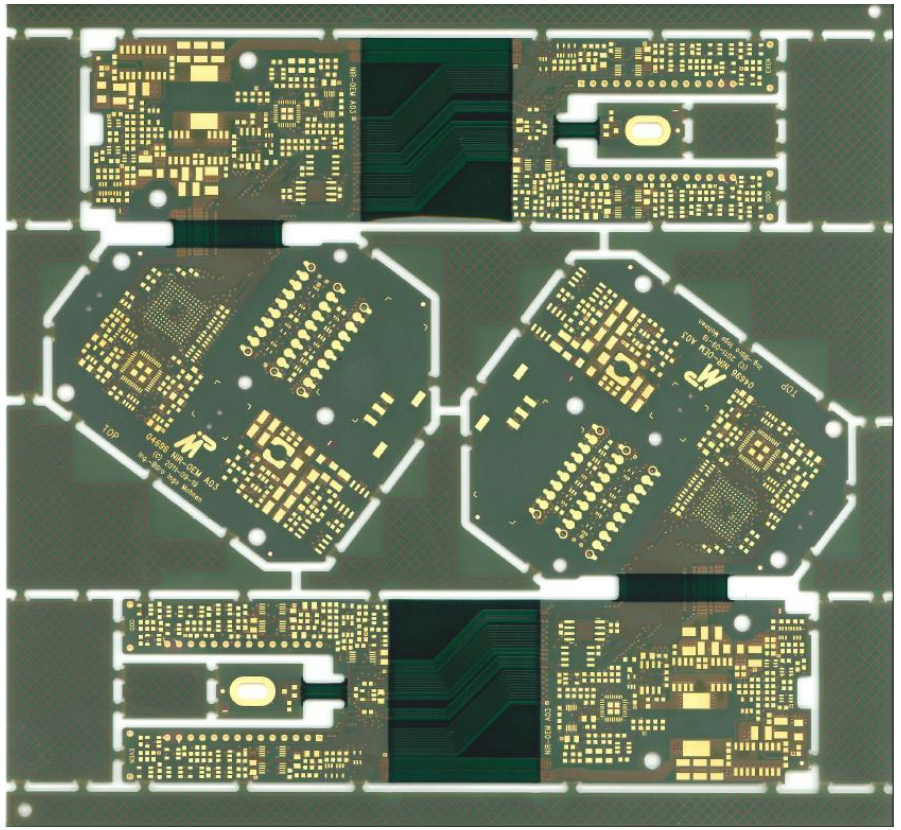


# Flex-Rigid Design: Good Examples

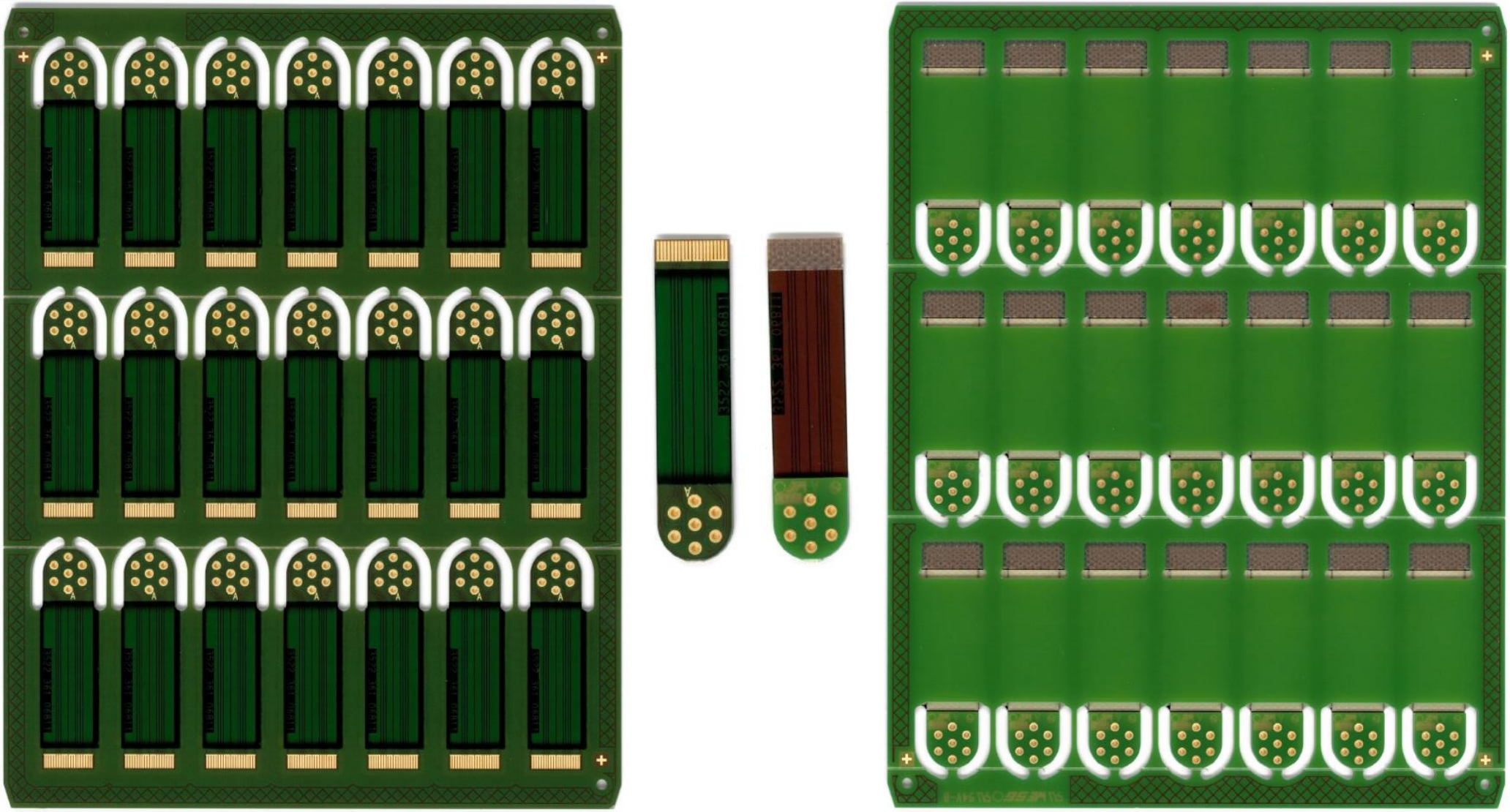




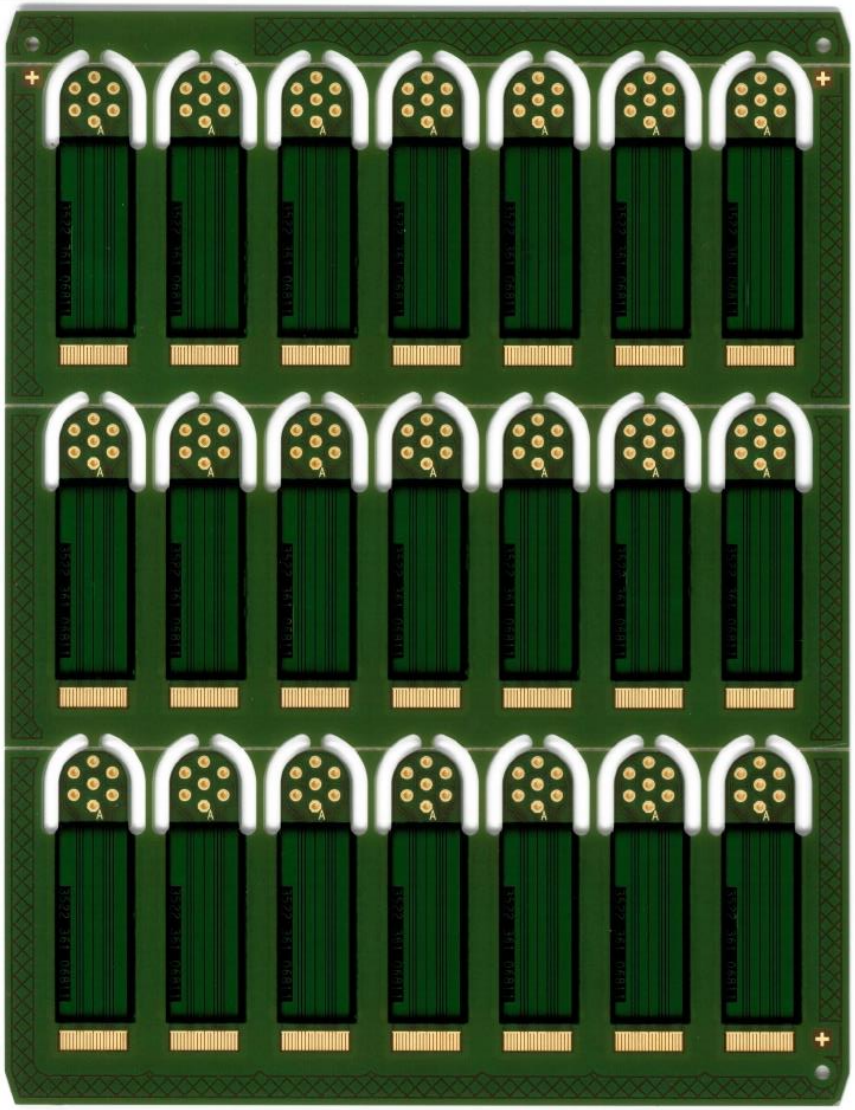


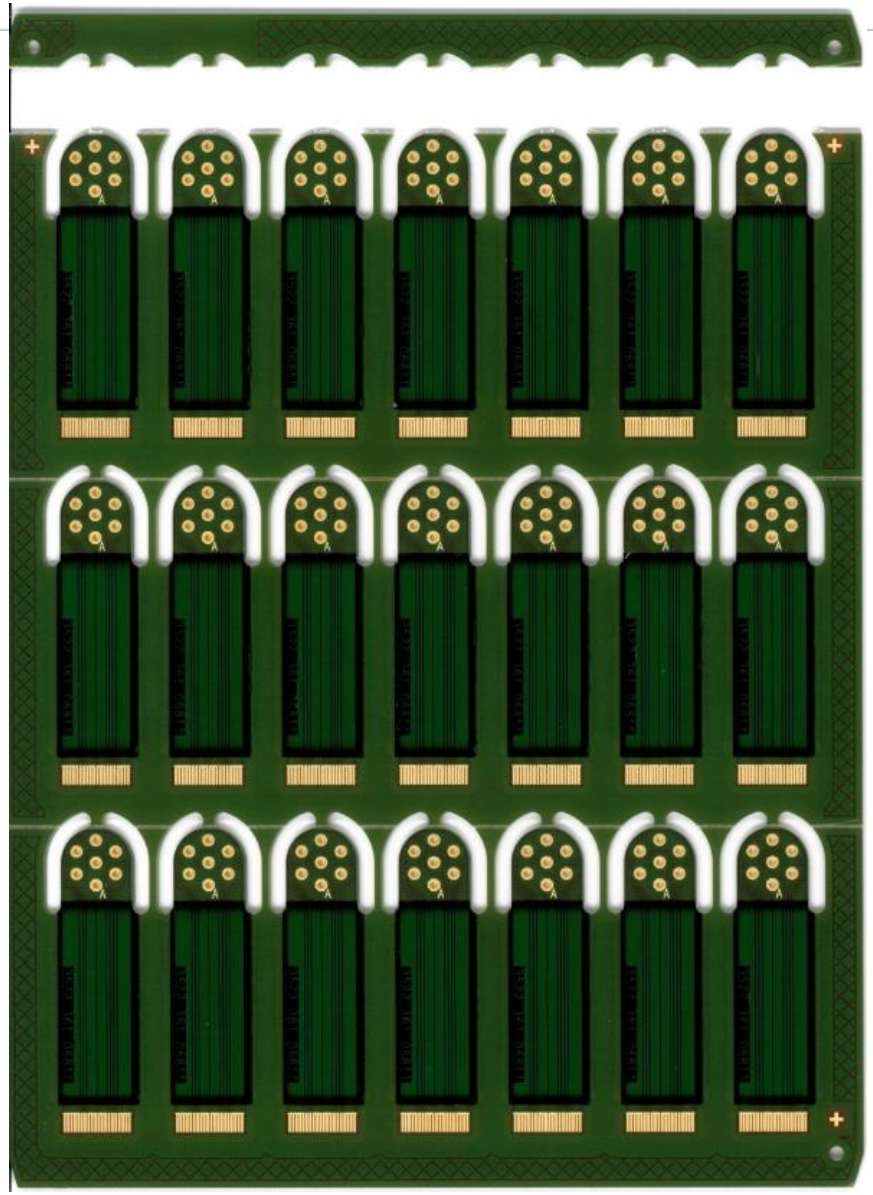


# Flex-Rigid Design: Good Examples

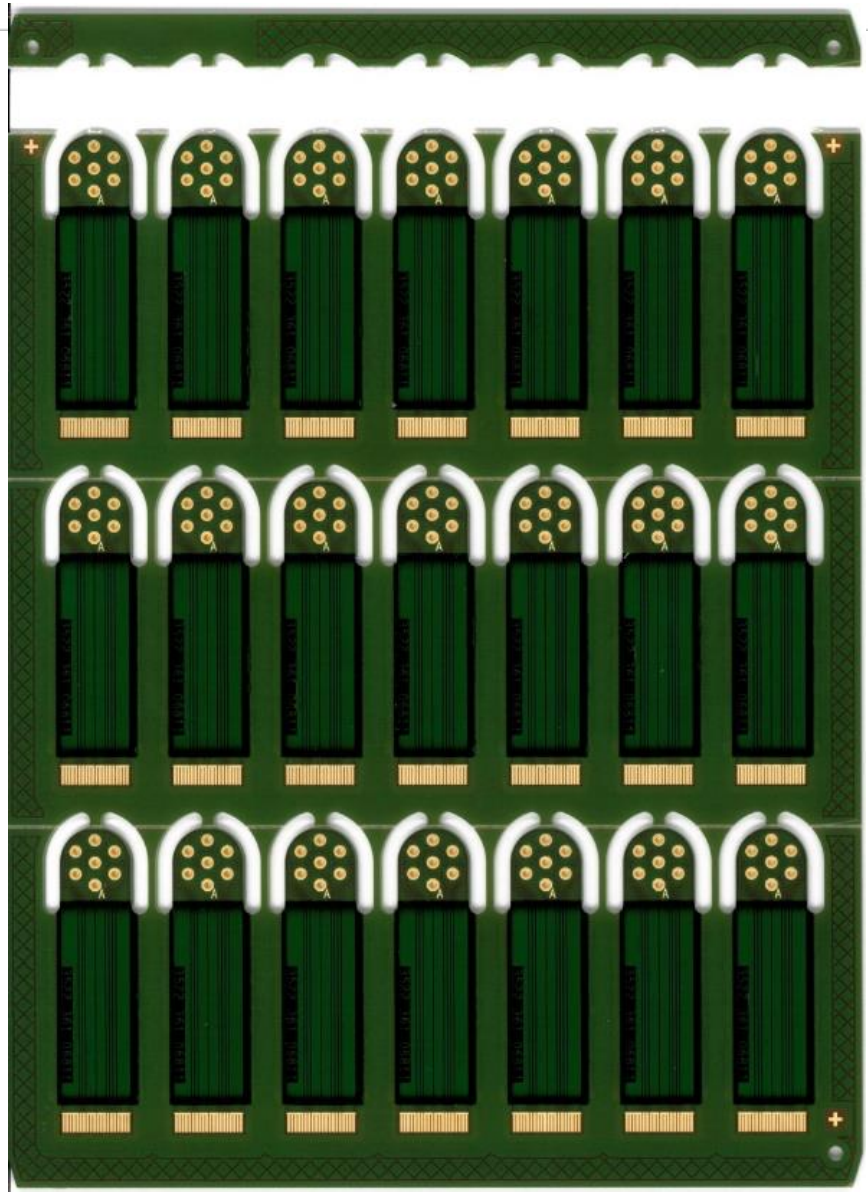


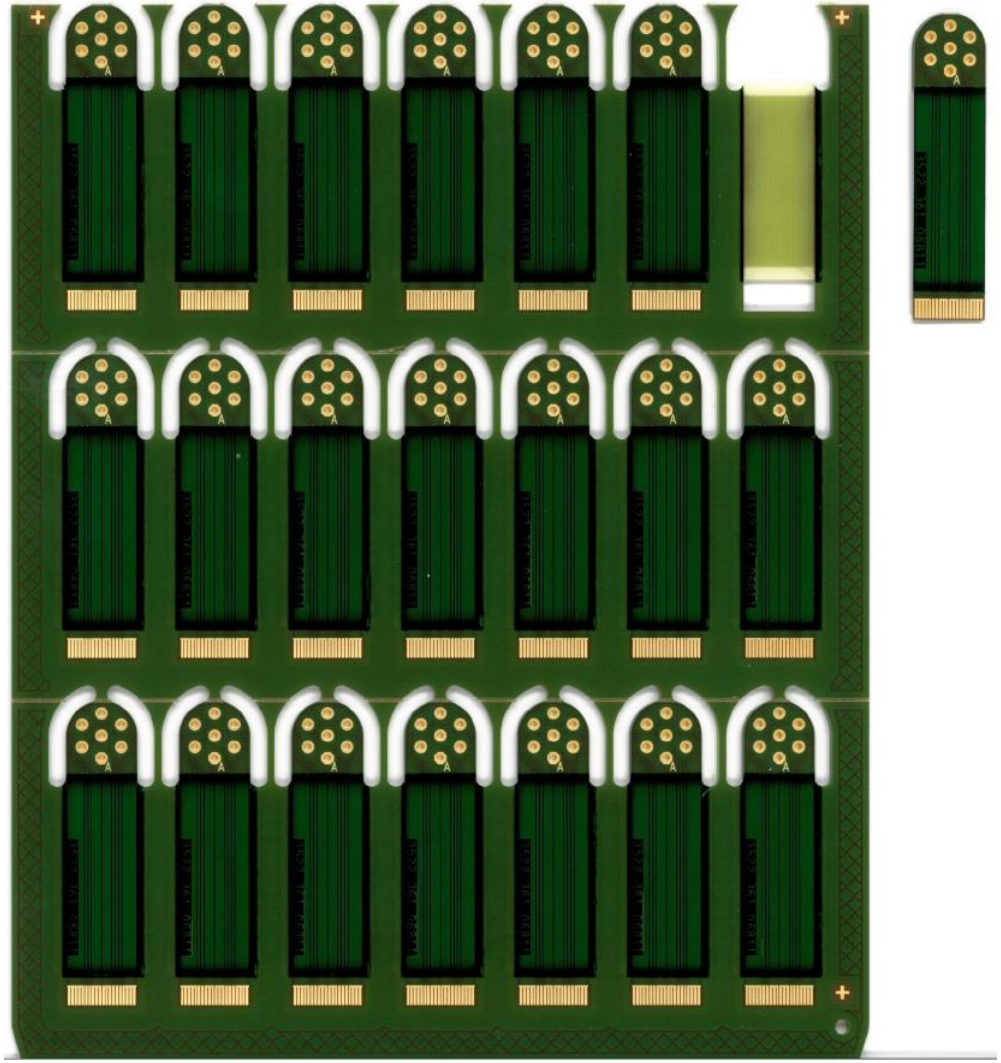
# Flex-Rigid Design: Good Examples



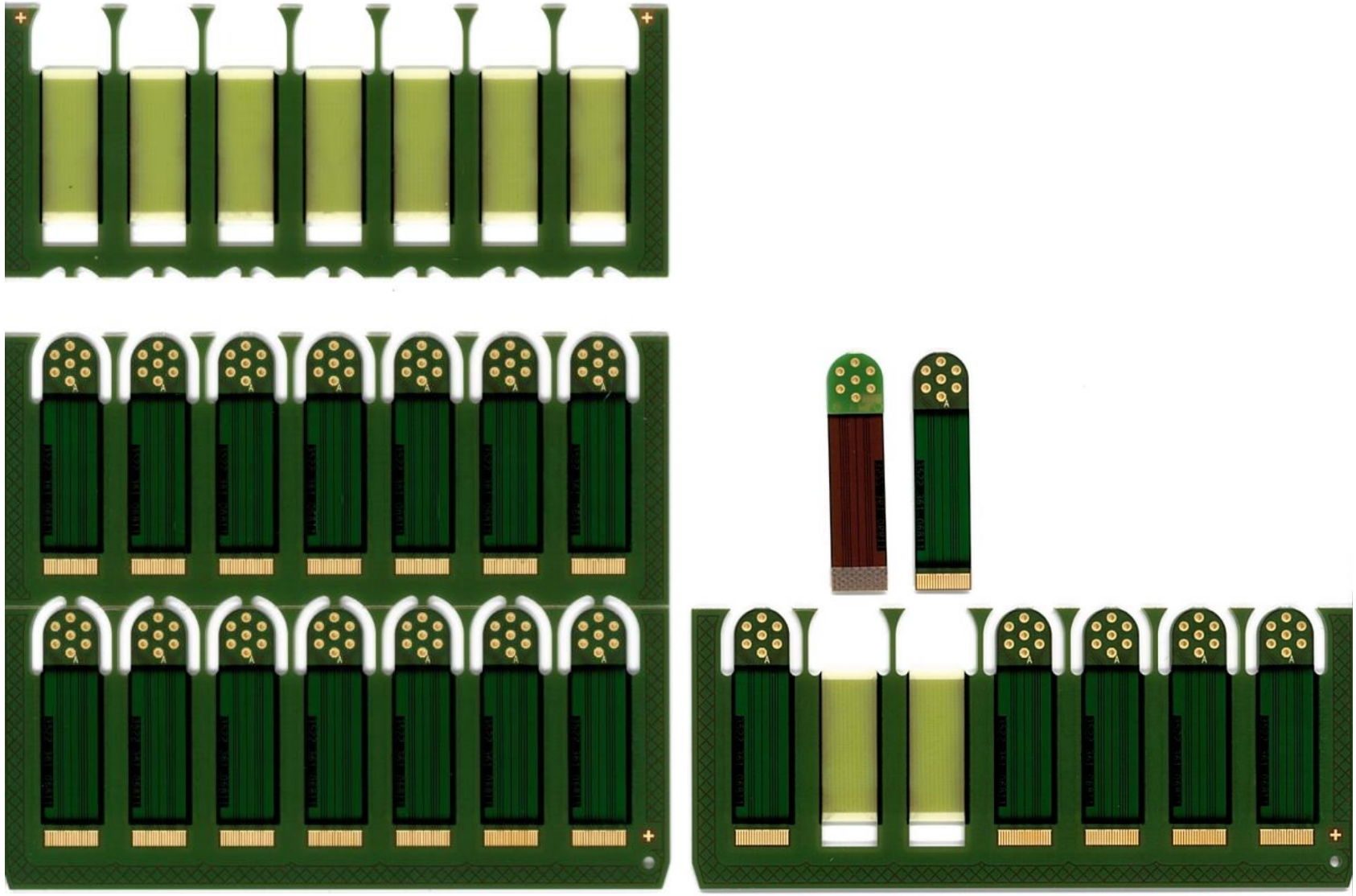


# Flex-Rigid Design: Good Examples





# Flex-Rigid Design: Good Examples

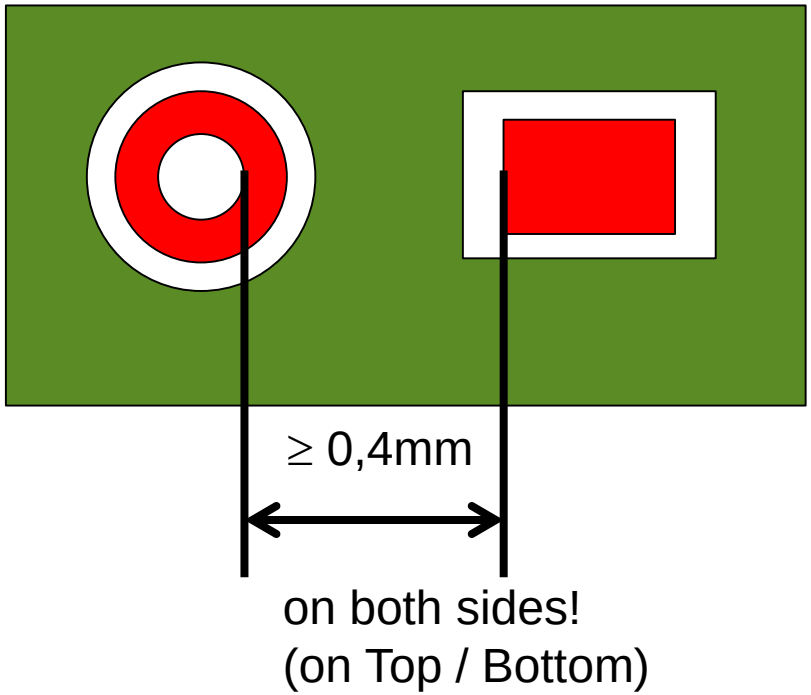


# Agenda

- 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 comtamination on solder pads**



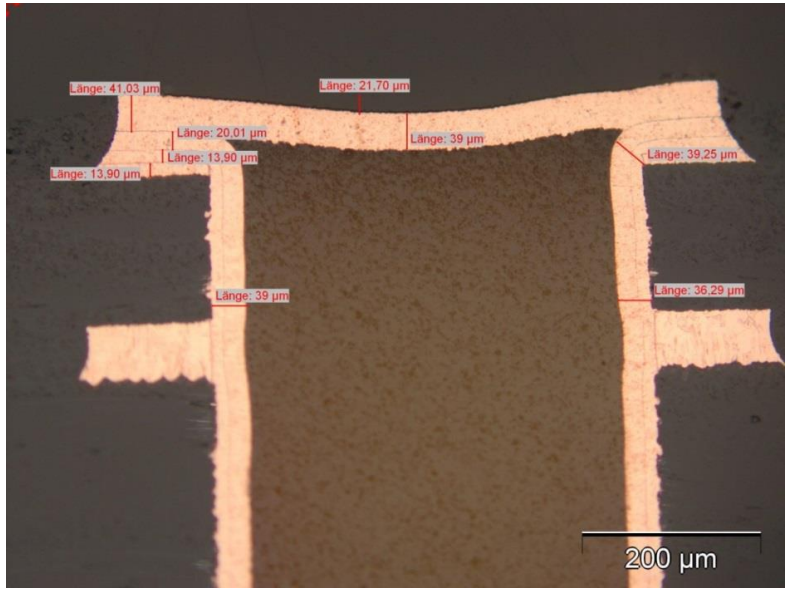

- Filling of plated through holes (PTH):

  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!