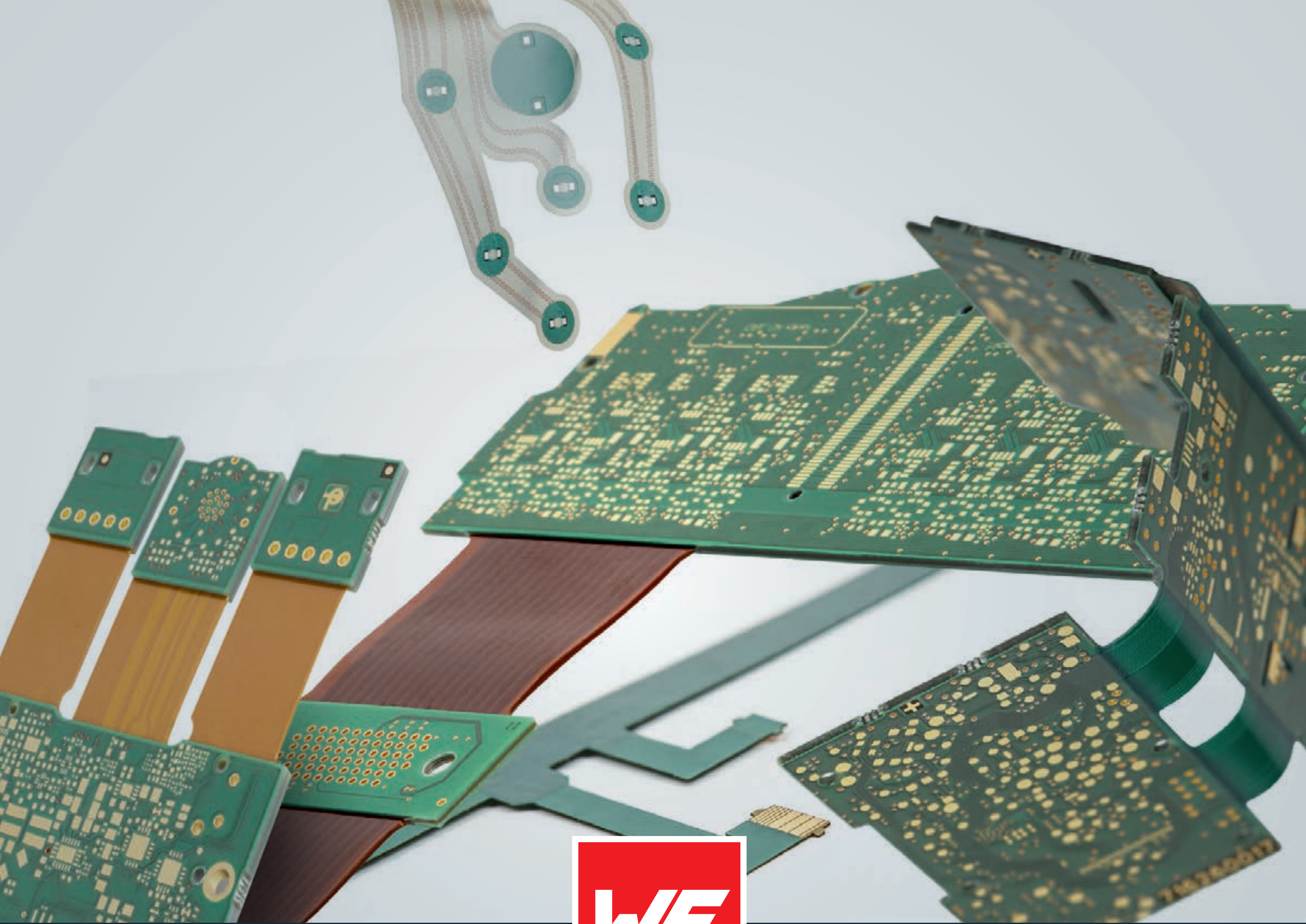


FLEX SOLUTIONS
DESIGN GUIDE

EN

WÜRTH ELEKTRONIK MORE THAN YOU EXPECT

CONTENTS

The trend towards miniaturization in electronics continues unabated. Efficient use of the ever-shrinking package volume in all three dimensions with a mechatronic PCB solution is becoming increasingly popular. At the same time, system benefits in terms of reliability, signal integrity and system cost are becoming increasingly important.

Flexible polyimide films with a typical thickness of 50 µm are high temperature resistant materials and can be used with copper lamination as base material for PURE.flex or in combination with rigid base materials as RIGID.flex PCBs using all common soldering methods. Thin FR4 layers are also bendable and are used in SEMI.flex and BEND.flex printed circuit boards. Finally, thermoplastic polyurethane is used in STRETCH.flex stretchable PCBs.

Over the past decades, Würth Elektronik has accumulated extensive know-how through the projects and orders it has already realized for a wide range of structures and applications, from aerospace to dental equipment, and currently supplies more than 600 customers. Due to the broad technology spectrum offered, the best possible selection in terms of performance and cost can be made for every requirement.

In the following, you will find system considerations that explain the different variants of Flex Solutions and give practical advice on design:

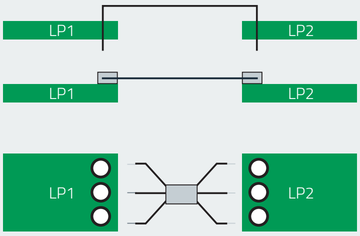
Basically, standards must be observed such as IPC-2223, IPC-6013, Basic Design Rules from Würth Elektronik as well as variant-specific design rules and our drying regulations.

!

1 FLEX SOLUTIONS IN THE SYSTEM VIEW

A SYSTEM CAN BASICALLY BE FORMED IN DIFFERENT WAYS

Inhomogeneous system



Rigid printed circuit board + cable harness/flex soldered or plugged (detachable)

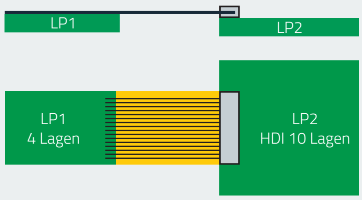
- Only a few connections
- Non critical applications only
- Wiring errors possible
- Many individual components
- High test and assembly effort

Homogeneous system

Identical stackup in all rigid areas and integrated flex layer(s) throughout

- Significantly higher wiring density
- Saves valuable space by eliminating connection points (solder pads or footprint of connectors)
- Best wiring reliability

Partially homogeneous system



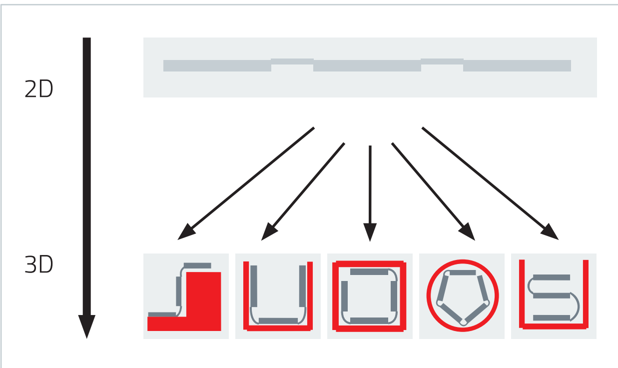
Subsystems vary greatly in terms of technology and size: e.g. RIGID.flex/connector combination

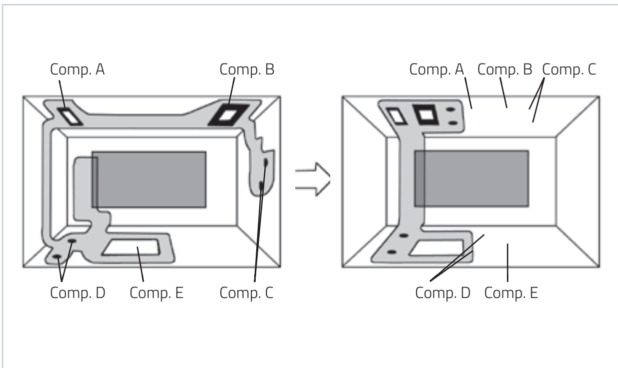
- Separable
- Suitable for modular systems

TIP:
The smaller and simpler part PCB1 should be provided with the integrated wiring.

Flex solutions are mechatronic components. In addition to the electronic function, great attention must be paid to the mechanical constraints.

Because the design phase determines the later cost structure, consideration of all electrical and mechanical intersections must already be made in the concept phase.



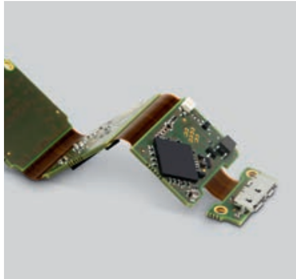


In addition, a precise selection of the optimum components and substrate technology is necessary in order to reliably meet the required operating conditions. Product development also includes having a precise idea of the assembly, soldering process, testing and device assembly.

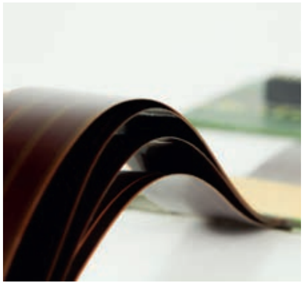
2 INNOVATIVE FLEX SOLUTIONS

Reliability

Signal integrity



Miniaturization



Dynamic bending

ADVANTAGES OF USING HOMOGENEOUS FLEX SOLUTIONS

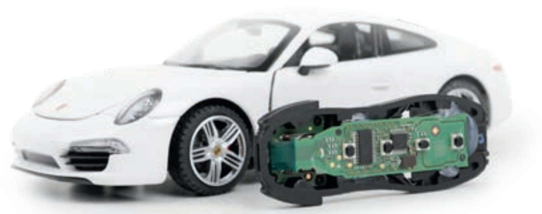
Miniaturization
- Additional layers for components
- Possibility of folding
- Saving of area for connector footprints
- Weight reduction
- Integration of interfaces
- Ultra-thin structures

Signal integrity
- Elimination of impedance jumps at connectors / cables / solder joints
- Reference layers are possible
- Reference layers with grid advantageous, calculation is possible

System cost reduction with improved performance
- Simple assembly
- Smaller housing
- Savings in cables and connectors
- Simplified testability
- Reduced complexity and cost for assembly and logistics

Reliability
- Fewer contacts in the overall system
- Wiring errors are eliminated
- Less mass (weight)
- Shock and vibration resistant
- Easier testability
- Easy mounting

Dynamic bending
- Flexibility and stability are opposing
- Properties
- Dynamic flexes are ideally
- Only 1-layer in the bending area
- Special copper grades are sometimes available
- Project specific stackup



Idea Concept → Specification → Mechanics development → Software development → Electronics development → PCB design

Projects with Flex solutions:
Interdisciplinary collaboration during development is essential!

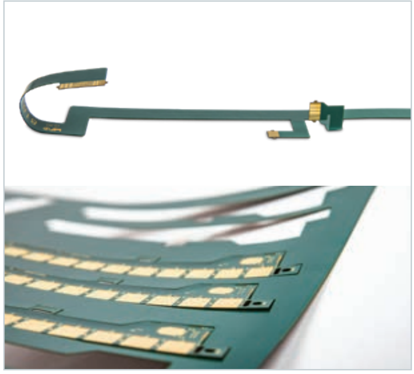
3 EXAMPLES OF APPLICATIONS

Rigid-flexible printed circuit board in a camera with high-resolution sensors, USB3 interface and a data transfer rate of 420 MByte per second.

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

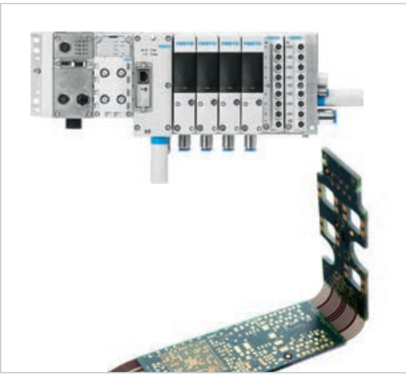
Dynamic printed circuit board system used in the control system of a patient couch for medical radiation therapy.

- RIGID.flex 1Ri-2F+2F+2F-1Ri



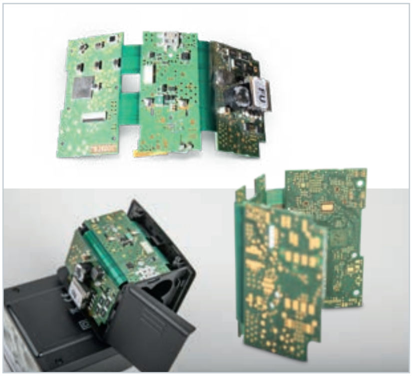
Sensor application with maximum miniaturization for most demanding environment (top) and high-tech cable harness (bottom).

- SLIM.flex 4F (top) and SLIM.flex 6F (bottom)



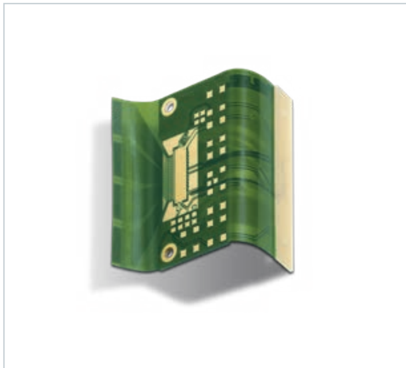
Heart of a pneumatic valve for digitized pneumatics. Robust and cost-effective alternative to RIGID.flex.

- BEND.flex 2Ri-4Ri + HDI 2-2-2



Semi-flexible printed circuit board around the battery box of a compact and powerful system flash unit.

- SEMI.flex 1Ri-3Ri



Sensor electronics for medical technology in flexible PCB technology with local stiffener.

- PURE.flex 2F-Ri

!

Your challenges drive us
We support you in the early development phase of your products and develop customized solutions together with you in projects. Contact us by mail (flex@we-online.com) or by phone at +49 7940 946-flex (3539).

4 PROJECT CHECKLIST FOR SYSTEM REQUIREMENTS

a) Technical requirements for the end product:
Target market, key features, unique selling proposition if applicable, life cycle, size, visual appearance.

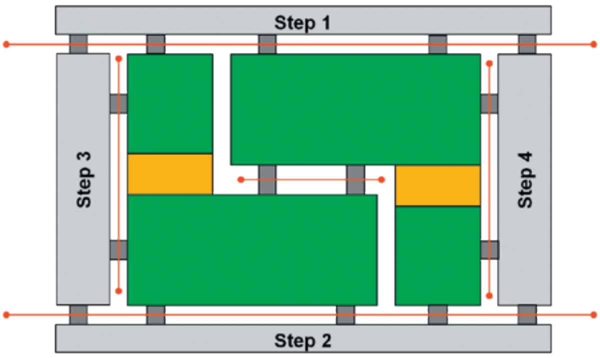
b) Commercial requirements:
Unit quantities, cost targets, schedule prototypes, pre-production, production release, ramp-up, second source, audit planning if applicable.

c) Regulatory requirements:
Approvals, regulated market medical technology, BAFA relevance, UL: Check UL Guidelines for the application: check minimum requirement for the PCB – no over-specification!

d) Reliability requirements:
E.g. classification according to IPC class 1/2/3, failure risk analysis, product liability, quality management agreement, industry-specific requirements e.g. APQP (Advanced Product Quality Planning) or PPAP (Production Parts Approval Process), Traceability.

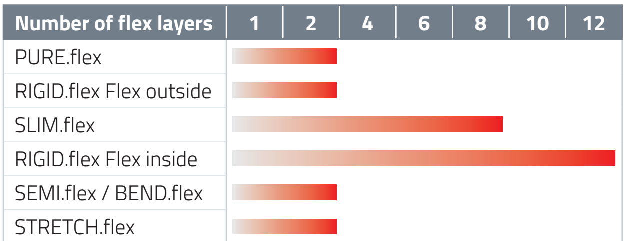
e) Product operating conditions:
Environmental conditions such as temperature, thermal cycling, heat dissipation, humidity, shock and vibration, flame resistance, assembly/soldering/repair, test procedures for environmental and reliability tests.

f) Casing size, material and shape:
Analysis of all mechanical and electrical interfaces, display, switches, connectors, interfaces to other devices or modules. Construction of a 3D model (paper+scissors / mCAD+eCAD) with the aim to find an area optimum for the 2D development of the circuit carrier.



g) Mechanical requirements PCB:
Static or dynamic application, PCB thickness, stability, aspect ratio drill diameter/PCP thickness, bending radii, ratio bending radius/flex thickness, bending shape, number of bending cycles, bending frequency.

h) Electrical requirements PCB:
Power, dielectric strength, current, insulation and shielding, EMC, number of signals over the flex area, number of flex layers, signal integrity, impedance requirements, surface resistance



i) Type and placement of components, packaging and connection technology:
Viatechnology-determining components such as BGA, stackup, PTH, microvia, buried via, anylayer via, filling, bare chip technology, solder surface, delivery panel, legend print, press-fit technology, embedded components.

j) Testing and packaging:
Electrical and mechanical test of the PCB, documentation product test (detailed initial sample test report recommended for complex stackup, specify test criteria), packaging.

k) Further processing of the PCB:
Design-for-drying, options of drying prior to soldering, logistics, dry storage, depaneling, handling, bending tools, casing assembly, installation tolerances and fastening options.

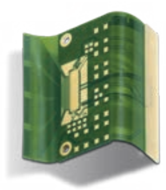
!

"L" shape better than "T"!
Würth Elektronik is happy to offer the optimum benefit (best price!)

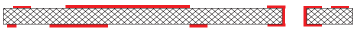
5 THE CHOICE OF THE RIGHT TECHNOLOGY

POSSIBLE VARIANTS:

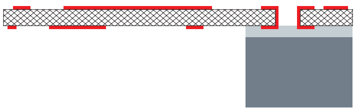
PURE.flex



2F (PURE.flex)



2F-Ri (PURE.flex with stiffener)



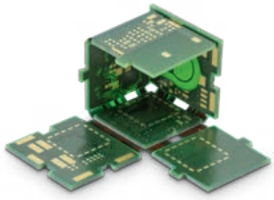
- Very thin flex film polyimide
- Up to two copper layers
- Partially reinforced by "stiffener"
- Photosensitive solder mask or cover foil (Polyimide Coverlay)
- Delivered individually or as panel array

Note
Stackup RIGID.flex 1F-0Ri can be cheaper than a PURE.flex 1F-Ri

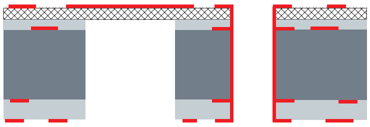
For comparison
1F-Ri (PURE.flex with stiffener)

1F-0Ri (RIGID.flex with depth milling)

RIGID.flex outside

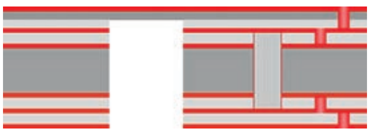


1F-3Ri



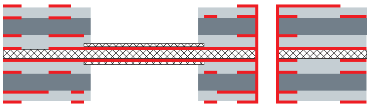
2Ri-4F

1F-5Ri + 2-2(4b)-2

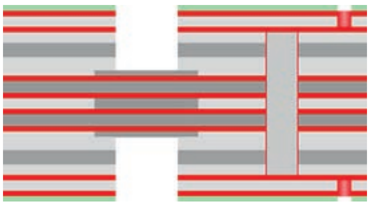


RIGID.flex inside

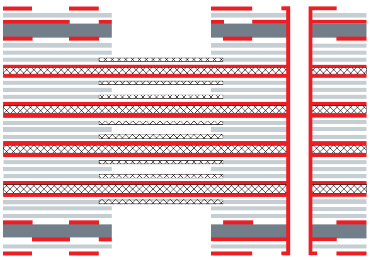
3Ri-2F-3Ri



2Ri-4F-2Ri + HDI 1-6b-1



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

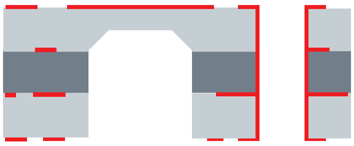


- Place components on stable rigid section
- Flexible area 1 to 12 layers, bonded/unbonded (with airgap)
- Flexible layers outside or symmetrical inside made of polyimide
- Rigid area: with standard solder mask
- Flex area: highly flexible flex solder mask or cover film (polyimide coverlay)

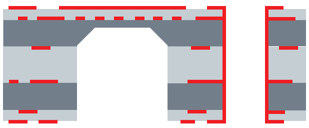
POSSIBLE VARIANTS:

SEMI.**flex** / BEND.**flex**

SEMI.flex 1Ri-3Ri



SEMI.flex 2Ri-4Ri



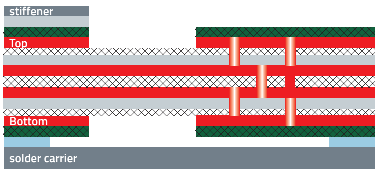
- FR-4.0 printed circuit board with bending area by deep milling process, bendable in one direction
- Cost-effective

BEND.flex
- FR-4.1 PCB with bending area, manufactured like RIGID.flex without polyimide foil, bendable in both directions

Notes
- Often cheaper than a plug-cable-connector solution
- Significantly better and cheaper than shielded cables and connectors

SLIM.**flex**

SLIM.flex 4F-Ri in Anylayer microvia technology



- Anylayer HDI flex technology
- Very thin
- Highly resilient, robust
- Flexible and reliable
- Impedance-defined structure possible
- Optionally with solder carrier and stiffener

Applications
- Vision Technology
- Medical technology
- Sensor technology
- High-tech cable harness

STRETCH.**flex**

STRETCH.flex 1S-Ri

- Stretchable circuit carriers made of thermoplastic polyurethane
- Conductors in e.g. meander or snake shape for extreme flexibility
- Dynamic extensibility depending on design from 5 – 20%.
- Almost any shape can be realized
- Skin friendly
- Can be combined with textiles
- Thermoformable

INDICATORS FOR PREFERRED USE OF INDIVIDUAL VARIANTS:

Variant	Indicators for	Comments
PURE.flex / SLIM.flex xF	Very small, dense circuits	Microvias and contour possible by laser
	Very limited installation space	Flex foil 50 µm thick
	Use in vacuum	Practically no outgassing
	High operating temperatures	PI can be used up to 200 °C (without solder mask)
	High frequency applications	Good thickness tolerance, Cu treatment flat, small loss factor
	Vias in flex area	ATTENTION: elaborate, (see table ff)
PURE.flex / SLIM.flex with Stiffener xF-Ri	Stability for placement and use	Reinforce placement areas mech. / solder carrier, heat dissipation problem / metal heatsink bonded
SEMI.flex	Flex-to-install with large bending radii	Inexpensive solution, miniaturization
	Large PCB with angled connector	Only bendability required
	Flex material not allowed	Only rigid base materials
BEND.flex	Bending direction arbitrary	Unprocessed glass mats can be loaded in tension and compression
RIGID.flex 1F-xRi	Large area Flex	Laser cut panels very stable
	1:1 wiring over flex area	Less expensive compared to xRi-2F-xRi
	Small bending radii Flex area	Thin, highly flexible flex coating or cover film
	Short drying times	Flex layer outside
RIGID.flex 2F-xRi	High-frequency connection component-to- connector via flex area with reference layer	No vias required for layer change
	High dynamic continuous bending	In flex area only layout on layer 2 - No metallization of the flex area
RIGID.flex xRi-1F-xRi	High dynamic continuous bending	Copper in neutral phase ideal, no metallization of the flex area
RIGID.flex xRi-2F-xRi	Reference layer in flex area due to signal integrity	Polyimide also with 75 or 100 µm possible for impedance control
	High reliability requirements	Robust technology, mechanical stability
RIGID.flex xRi-4F-xRi	Coaxial impedance structures in flex area	Air gap xRi-2F+2F-xRi is not suitable here

INDICATORS AGAINST THE USE OF INDIVIDUAL VARIANTS:

Variant	Indicator against	Remarks
PURE.flex / SLIM.flex	Wired components or connectors	Low mechanical stability
PURE.flex with stiffener	Many individual reinforcements	Better use RIGID.flex 1F-xRi
SEMI.flex	Bending with milled surface outside, S-shaped bending in one surface	Do not load glass mat in tension
	Fastening to multi-part housing	Mounting tolerances affect Semiflex area
RIGID.flex 1F-xRi	-	-
RIGID.flex xRi-2F+2F-xRi	Length of flex area shorter than 50 mm	Due to buckling during bending, this is made more difficult or impossible if the flex area is too short.

!

Rules for copper structures, via sizes and solder mask can be found in BASIC Design Rules from Würth Elektronik.

!

A more detailed analysis of the requirements from the project checklist (chapter 4) should be made in a project discussion with our specialists.

6 MATERIALS AND DESIGN PARAMETERS – STANDARDS

IPC CLASS 2, USE A (FLEX-TO-INSTALL)

Flex material	Polyimide core 50 µm standard, up to 70 µm copper, adhesiveless, coverlay film 25 µm partial or flexible solder mask
Rigid material	FR-4.1 Tg150 °C, filled, halogen free: IPC-4101C / 128 (92,94,127)
Copper thickness	inner layers 18(standard)/35/70 µm // outer layers 12/18/35 µm + plating (for 1F-xRi Flex on outer layer)
PCB thickness	depending on number of layers: PURE.flex / SLIM.flex > 100 µm, RIGID.flex / SEMI.flex ≥ 0.8 mm
Solder surface	ENIG and optionally nickel-free immersion solder surface

Further requirements e.g. regarding materials, stackup, use case Use B etc. on request!

STANDARD STACKUPS FLEX SOLUTIONS

Standards make our lives easier, ensure the quality of products and services and reduce costs. That's why we offer standard stackups for many technologies that are customary in the market and cost-optimized.
You can use these standards as a basis for your solutions and modify them according to your requirements.

- PURE.flex (**www.we-online.com/pureflex-stackups**)
- RIGID.flex **(www.we-online.com/rigidflex-stackups)**
- SLIM.flex **(www.we-online.com/slimflex-stackups)**

Of course, we also produce stackups according to your individual, project-specific requirements, after we have checked them for technical feasibility. Please contact us!

COMBINATION WITH OTHER TECHNOLOGIES

It is possible to combine Flex solutions with other technologies, for example with

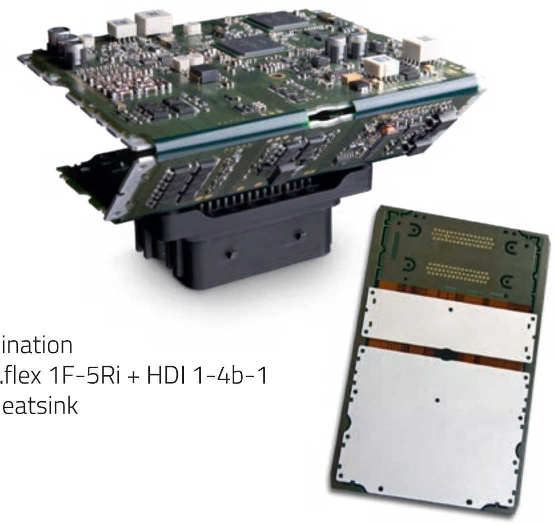
- High Density Interconnect MICROVIA.hdi
- DEVICE.embedding
- Printed Polymer
- Heatsink

An adaptation of the individual design rules is usually necessary.

SOLDER SURFACES

To ensure solderability, exposed copper surfaces are provided with a solder surface.
When making your selection, please consider, among other things, the suitability with regard to

- drying processes: Aging due to temperature stress
- bending: Nickel containing surfaces such as ENIG are not suitable for bends.



Combination
RIGID.flex 1F-5Ri + HDI 1-4b-1
and Heatsink

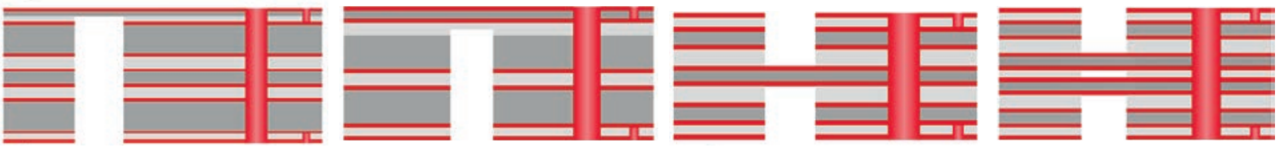
MINIATURIZATION TO THE POWER OF TWO: DIVERSE COMBINATIONS OF RIGID.FLEX AND HDI

Little space in the housing and micro-BGA components on the circuit: Miniaturization to the power of two?

WE HAVE THE SOLUTION:
With combinations of RIGID.flex and elements from High Density Interconnect, you can alleviate even tight spaces.

Microvias, filled holes, locally finest copper structures and thus a clever unbundling of BGA footprints save a lot of space. RIGID.flex technology also saves installation space for connectors and cable harnesses, and the circuit can be folded into the housing to save space.

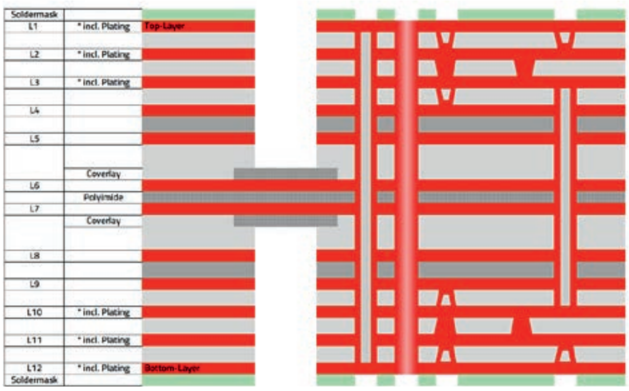
- The HDI design rules must be adapted to the stackups and processes of the RIGID.flex technology and cannot be transferred 1:1. The aspect ratio between hole depth and hole diameter has to be considered.
  Click here for the **HDI Design Guide.**
- Standard RIGID.flex stackups with HDI 1-x-1: All higher layer stackups are suitable for microvias on the outer layers.



from left: 1F-xRi / 2F-xRi / xRi - 2F - xRi / xRi - 4F - xRi

- For the use of multiple microvia layers and buried vias, the stackup must always be adapted. Here we are happy to make a suggestion.
- After selecting the via strategy, the stackup must be adjusted accordingly.
- **IMPORTANT:** In addition, the following must be taken into account:
- The minimum possible copper structures in the PCB design depend on the copper layer thickness.
  - The requirements for minimum copper layer thicknesses are based, among other things, on the IPC class specification.
  - Filling and plating processes generate additional copper buildup and thus limit the use of ultra-fine structures.
  - The combination of different via types can lead to multiple plating processes and thus defines or limits the smallest possible structures in the PCB design.

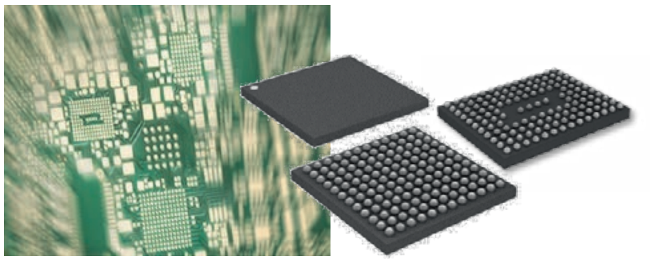
EXAMPLE



- Layers 1 and 12 (top and bottom): PTH (IPC class 2) + PTH Filled&Capped (type VII) + Microvia
- Layers 2 and 11: Microvia with copperfilling
- Layers 3 and 10: Microvia + Buried Via Filled&Capped (IPC-4761 type VII) (IPC class 2)
- Layers 4 to 9: inner layer copper without plating

EXAMPLE FROM PRACTICE

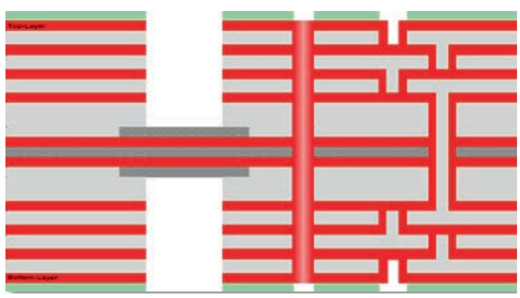
The cooperation of the PCB designer with the specialists of our technical project management led to a reliable and well manufacturable product by small adjustments of the design rules. For example, stacked microvias with copper filling could be avoided.



UFBGA 169 P 0.5 mm W-CSP138 P 0.4 mm

Technology: HDI-RIGID.flex with BGA component pitch 0.4 mm

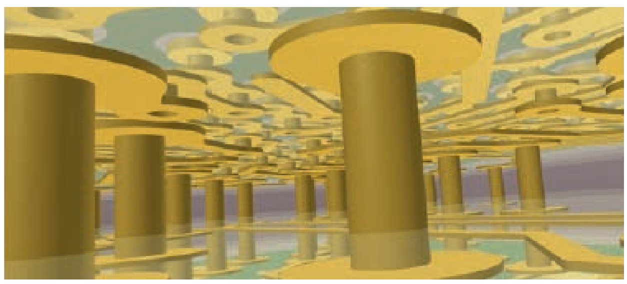
- RIGID.flex 4Ri-2F-4Ri + HDI 3+4b+3
- Special design rules local in µBGA area
  - Microvia pad smaller : 250 µm
  - µBGA pad smaller: 240 µm
  - Solder resist clearances 50 µm, partly 40 µm
  - basically Microvia-in-Pad
  - partial conductor width / spacing: 70 µm / 90 µm asymmetric (layers 2/3/8/9 with plating)



left stackup RIGID.flex 4Ri-2F-4Ri + HDI 3+4b+3, right spatial via structure

For all via combinations, individual coordination on the stackup and structures with the FLEXperts is required. Please contact us as early as possible! flex@we-online.com

!



RIGID.FLEX WITH IMPEDANCE-DEFINED STACKUPS, IMPEDANCE MEASUREMENT WITH RIGID.FLEX.

The variants within the RIGID.flex technologies allow the selection of different impedance models in the design phase. By tuning and adapting the stackups and the design parameters, all common interface specifications can be simulated and measured.

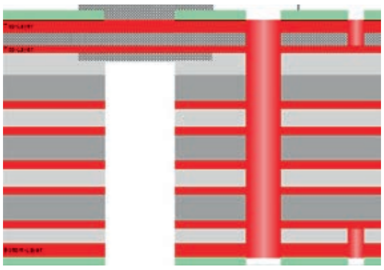
BASIC ADVANTAGES OF RIGID.FLEX REGARDING SIGNAL INTEGRITY

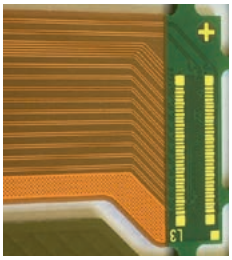
- Reproducible impedance behavior between rigid areas, elimination of impedance jumps at connectors, cables and solder joints as well as fluctuations due to manufacturing tolerances
- Flexible areas as cable harness with reference or shielding, coplanar is also possible
- Higher flexibility in comparison with cables, can be designed with copper pitches.

VARIANT-SPECIFIC ADVANTAGES OF THE RIGID.FLEX VARIANT 2F-XRI

This design has some specific advantages

- 1:1 transmission of signals over the flex area without layer change, thus no impedance jumps and reflections
- Contacting of the reference layer via microvias possible without major additional effort
- Signal routing optionally also possible on inner layer / layer 2 without plating tolerances.





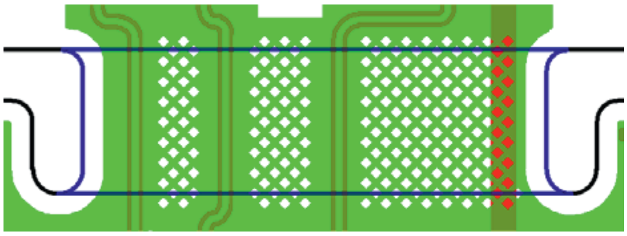
left stackup RIGID.flex 2F-6Ri - HDI 1-6-1, right printed circuit board

STACKUP AND COPPER DESIGN: HATCH

A cross-shaped rasterization of the reference layers leads to

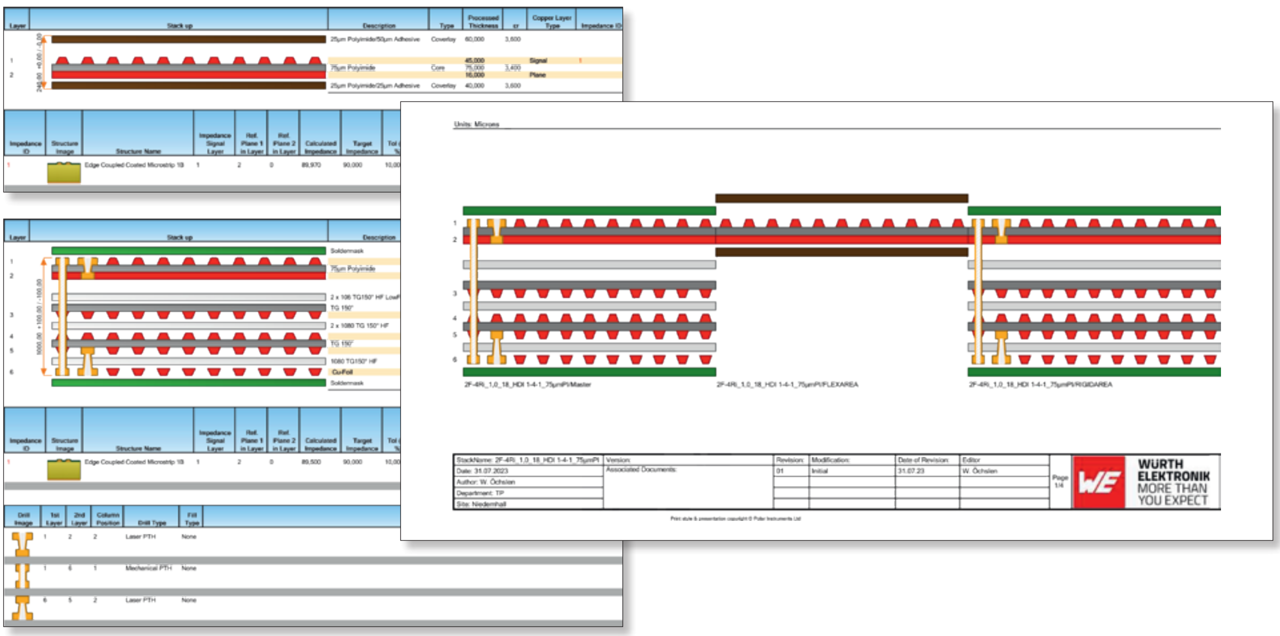
- Improvement of bendability
- improvement of drying possibilities before soldering
- Increase impedance, thinner, less expensive polyimide foils can be used.

Screening should always be used in non-critical areas and up to 5 to 6 GHz. As a compromise, non-screened copper can be used locally for an undisturbed return path under critical signals.



IMPEDANCE CALCULATION

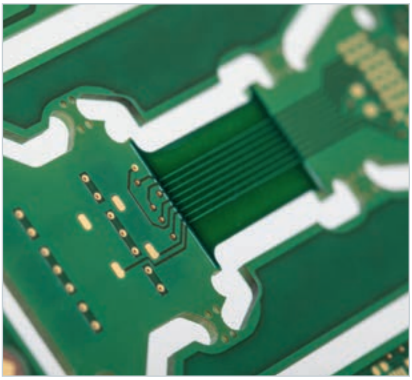
With the specifications of the impedance values and the relevant signal or reference layers, we can modify a stackup for you according to production requirements as well as determine the necessary design parameters.
As a result, you will receive the calculation and documentation as shown below using the example of a RIGID.flex 2F-4Ri + HDI 1-4-1 stackup.



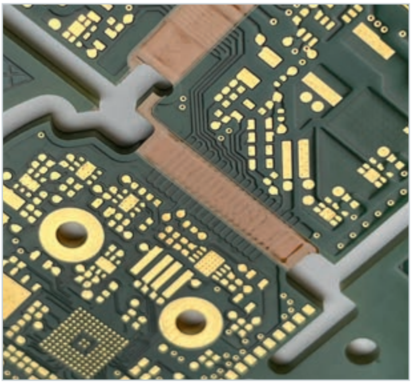
COMPARISON OF FLEXIBLE SOLDERMASK VERSUS COVERLAY

RIGID.flex circuit boards often differ not only in their application-specific design but also in the structure and coating of the flex areas. While the rigid areas are covered with a standard green solder resist, as with rigid PCBs, two different coatings are used for the flexible areas:

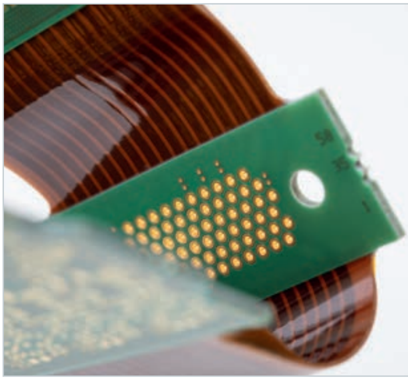
EXAMPLES



Flex outside: left flex solder resist



Center coverlay



Right: Flex inside with Coverlay

FLEXIBLE SOLDER RESIST OR COVERCOAT

Flex soldermask is a varnish, i.e. a mixture of binder, solvent and pigments. In contrast to solder resist for rigid printed circuit boards, flex resist remains flexible after curing, ideally resistant to folding. Flex varnish is applied by inkjet or screen printing.

COVERLAY OR COVER FILM

Coverlay is a composite film consisting of polyimide film and adhesive layer. Polyimide film thickness standard 25 µm, adhesive thickness matching the copper layer thickness. Other polyimi-de film thicknesses optional. Cutting or laser, manual registration and tacking, pressing in vacuum press.

WHEN CAN I CHOOSE BETWEEN FLEXIBLE SOLDERMASK AND COVERLAY?

The choice is only possible for the variants with 1F-xRi and 2F-xRi outer flex layers - and even then only for the outer layer. With Stackup 2F-xRi, copper layer 2 is always protected with a Coverlay, as are all copper layers with inner flex layers.
With PURE.flex, it is also possible to choose between flex coating and overlay.

!

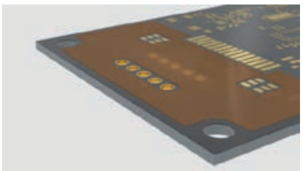
Are you looking for a special solution?
Talk to our FLEXperts!

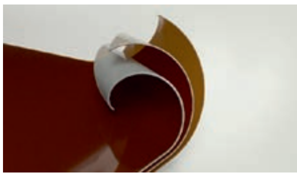
COMPARISON OF SELECTED PROPERTIES

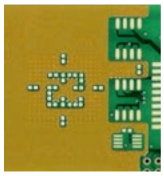
	Flexible soldermask	Coverlay
Colour	Green	Amber / brown
Composition	Mixture of binder, solvent and pigments	Composite film of polyimide film and adhesive layer
Registration	Automatic, optical	mechanical
Application	Liquid by inkjet or screen printing	Manual application, vacuum lamination
Structuring	Partial application, photo process	Cutting, lasering
Design	Very variable, small individual areas possible	Small individual areas must be connected in the PCB and/or in the delivery array
Dynamic bending application	No	Yes
Applicable to inner layers	No	Yes
Applicable to outer layers	Yes	Yes
Maximum copper thickness	up to 70 µm	up to 70 µm
Minimum distance vias and pads to rigid-flex intersection	Smaller, see Design Rules parameter "G"	Larger, see Design Rules parameter "G"
Use in vacuum	Limited	Very good
Mechanical robustness	Lacquer with pencil hardness ≥ 3H	Resistant film
Dielectric strength	Approx. 150 V at 5 µm thickness	Approx. 3500 V/mil (1 mil = 25.4 µm)
Tenting of microvias	Limited	Yes
UL Listing	Yes	Yes
Effort and cost	Low effort, inexpensive	High effort, more expensive

COVERLAY AS INSULATION FOIL

Coverlay foil can be used as a protective and insulating foil, both on the rigid areas of a STARR.flex PCB and on rigid BASIC or HDI PCBs.

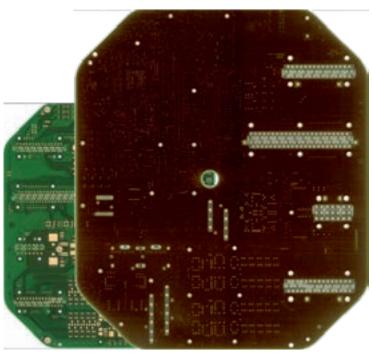


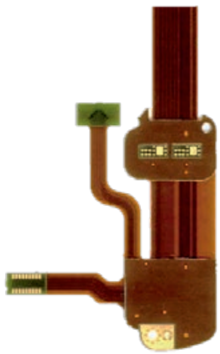




PROPERTIES OF COVERLAY / COVER FOIL

- Polyimide thickness typically 25 µm, adhesive thickness 25 µm or 50 µm
- Flame retardant with rating V-0
- Thermal conductivity $0.2\ W\ m^{-1}\ K^{-1}$
- Dielectric strength ≥ 4 kV/mil
- Low outgassing





AREAS OF APPLICATION

- Electrical insulation
- Thermal conductivity
- Mechanical protection
- Tight spaces
- Safe spanning of bores

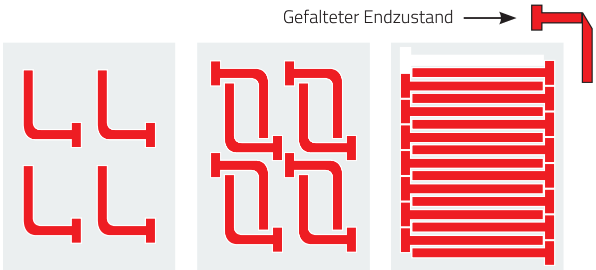
APPLICATIONS

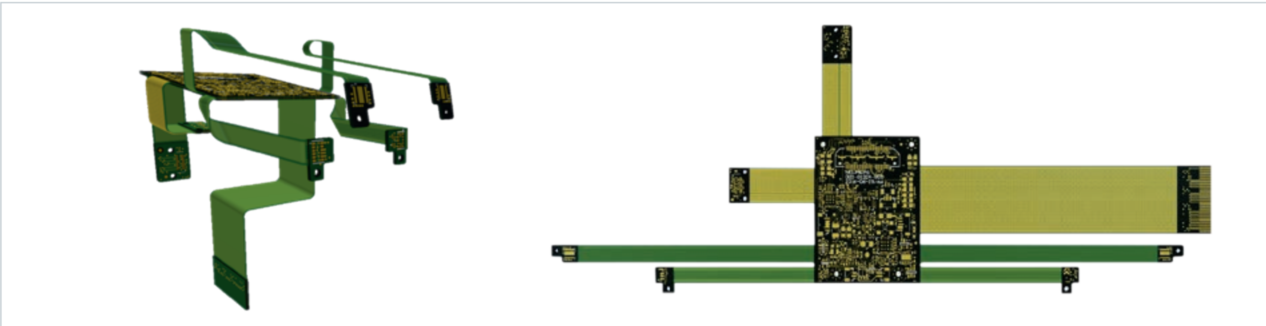
- Engine control units
- Busbars
- High current
- Sensors

Find here our Design Rules:
https://we-online.com/designrulesisolationfoil

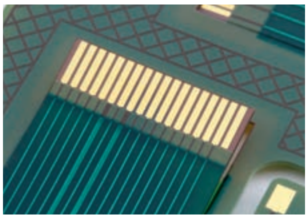
7 THE MECHANICAL DESIGN

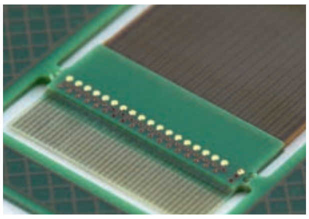
- Design suitable for plastics: Always provide large contour radii on inner and outer radii in the flex areas
- If necessary, provide registration holes for bonding reinforcements or heatsinks
- Arrange flex extensions to save space, combine several extensions if possible, use folding technology

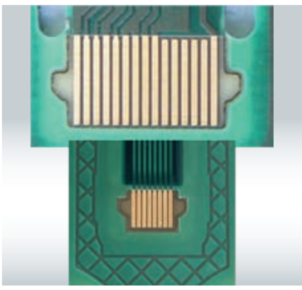




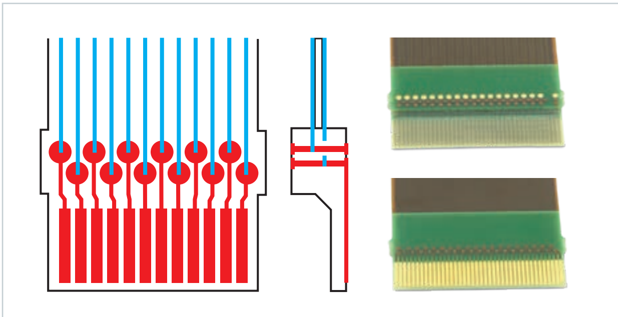
- RIGID.flex:
  components and connectors always on rigid areas
- Contacts for ZIF connectors:
  - Standard thickness 0.3 mm ± 0.05 mm, made by depth milling process
  - Handling aids and latching hooks possible
  - Increased requirement: tolerance +/- 0.03 mm with polyimide stiffener possible after consultation (additional costs!)
  - Outline cut with laser, micro-bridges allow separation without cutting tools







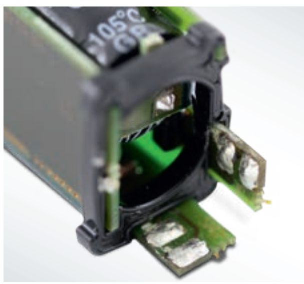
- For RIGID.flex with flex inside, a change to the outside layer by means of vias is recommended near to the ZIF area

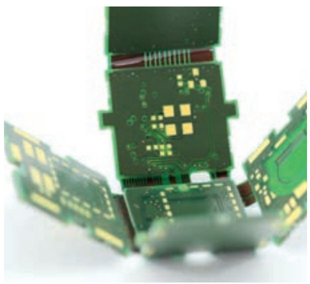


Design bending radii depending on flex thickness

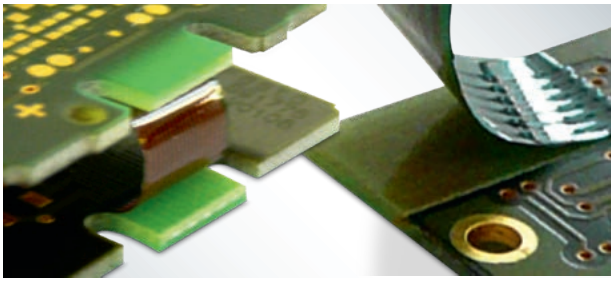
Bending radius [mm]	1	2	3	4	5	6	7	
Flex area 1-layer				thickness x 10				IPC-2223: Use A
Flex area 2-layers				thickness x 10				Flex-to-install
Flex area 4-layers				thickness x 20				
SEMI.flex								
BEND.flex								

- RIGID.flex: use "key and slot" design or plastic holding frames

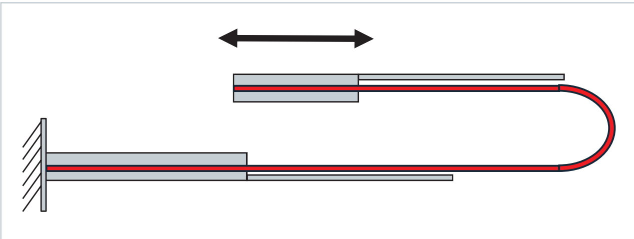




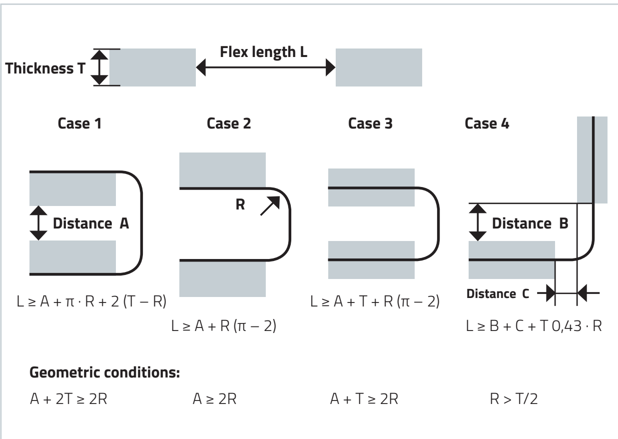
- "Flex Lift-off" – Technique
  Attention: NO copper layout under the flex
  and NO vias allowed!



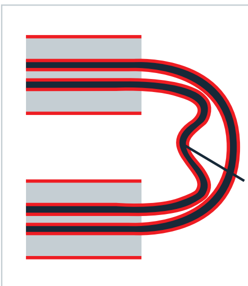
- Lift-off technique with contact and roll-off surface ideal for continuous bending loads



- Calculation of the flex length

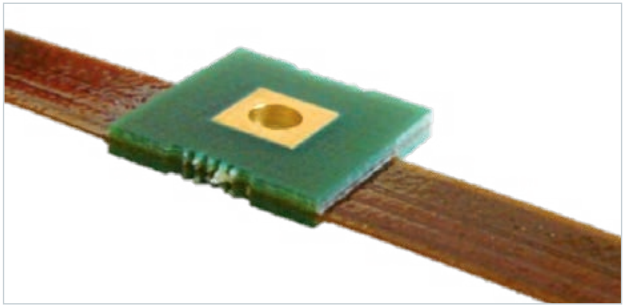


- With short flexible areas, four bonded flex layers are easier to bend than 2+2 flex layers with airgap

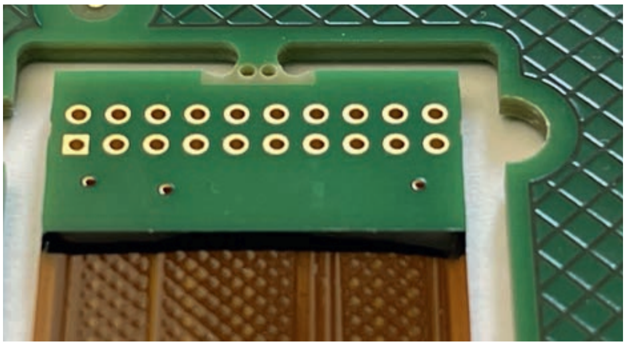


Air gap designs must have a sufficient length of the flex area, we recommend at least 50 mm.

- Fixings: For attachments of flexible areas in RIGID.flex, use small integral rigid areas
  - with metallized or non-metallized holes for fastening to the housing
  - as a bending aid and for fastening bent / folded flex areas.



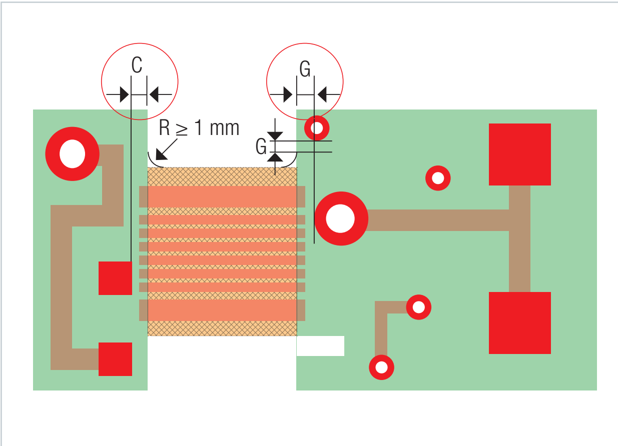
- Option adhesive fillets for stress relief according to IPC-2223 5.2.9



8 LAYOUT AND ROUTING

SPECIAL FEATURES AND RECOMMENDATIONS FOR RIGID.FLEX PCBS

- Use extra mechanical layer to describe flex and rigid areas in EDA tool



- Do not use vias in bending area
- Use teardrops
- Do not remove non functional / non used pads on the flexible layers
- Copper areas on all layers must be hatched to allow drying. For more information, please visit **www.we-online.com/dryingspecification**
- Impedance Design:
  - Adjustment of the layout parameters in the flex area to the changed dielectric values is necessary
  - Hatching of the reference layer directly above or below differential transmission lines can be omitted, but not in the entire reference layer
  - For further notes and explanations see chapter 6: RIGID.flex with impedance defined stackups, Impedance measurement for RIGID.flex
- Flexible and rigid-flex PCBs must be dried before assembly. Further information can be found at **www.we-online.com/dryingprocess**

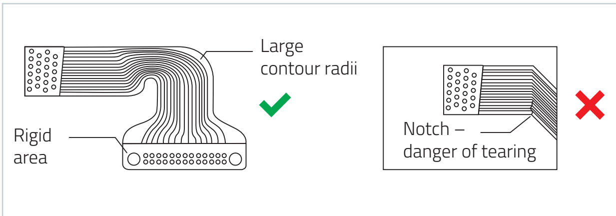
DESIGN RULES

Our Design Rules cover all important parameters you need to make your project successful. Basically, the rules for line widths, distances, via and pad sizes as well as for the solder mask apply, which you will find in our BASIC Design Rules (**www.we-online.com/designrulesbasic**).

Based on these rules, the sectional design rules for

- PURE.flex **(www.we-online.com/designrulespureflex)**
- SEMI.flex **(www.we-online.com/designrulessemiflex)**
- RIGID.flex **(www.we-online.com/designrulesrigidflex)**
- SLIM.flex **(www.we-online.com/designrulesslimflex)**
- STRETCH.flex **(www.we-online.com/designrulesstretchflex)**

LAYOUT / ROUTING IN THE BENDING AREA



- Line routing parallel and perpendicular to the bending line
- Right-angled line routing at the flex-rigid transition
- No vias in the bending area on flex PCBs
- No change in width or direction of lines in the bending area
- Use round line shapes and large contour radii
- Distribute conductors evenly, fill copper-free areas
- Offset conductors on TOP/bottom when routing on both sides (prevent "I-beam" effect)
- Always provide full-surface copper reference layers with copper openings to improve flexibility and dryability in both flex and rigid areas

Regard distances of drill holes and SMD pads to the flex-rigid transition, see Design Rules Würth Elektronik

!

!

For IPC-2223 "Use B" (dynamic bending load) and "Use D" (UL listing) applications, please consult our specialists!

9 THE DOCUMENTATION FOR THE RIGID.FLEX PRINTED CIRCUIT BOARD

- Include all applicable items from the project checklist (Chapter 4). As a matter of principle, please ensure that no over-specification with exaggerated safety margins is formulated. Beware of copying specifications from other projects, this usually leads to excessive costs. With regard to UL, a UL94 V-1 specification is almost always sufficient.
- Define what is leading: drawing or data. Electronic data (Gerber/ODB++, CAD data) already contain all geometric dimensions.
- No over-specification: a good drawing shows the flat projection of the circuit (2D), a 3D view of the installation situation and only critical dimensions. Every redundant dimension on the drawing must be compared with the electronic data and, if there are discrepancies, will inevitably lead to clarifications, delays and additional work resp. costs.
- Specify materials as generally as possible, e.g. according to IPC specification sheets. Do not dictate material designations of supplier X with material Y.
- Adhesive layers should generally not be dimensioned, but only the total thickness, copper thicknesses and necessary dielectric thicknesses (e.g. due to impedance or insulation requirements).
- General specifications for the creation of an optimal delivery panel and possible positions for webs. In the flex area, laser cutting offers a preferred alternative. A delivery array specified in the inquiry can then be replaced by a more favorable one and leads to a more favorable price due to a better utilization.

10 FURTHER PROCESSING

DRYING:

As a matter of principle, printed circuit boards already have a certain moisture content directly after production due to the manufacturing processes and ambient conditions. Thus, without special treatment, they are never free of moisture that has diffused into the dielectric materials. Over the storage period, the moisture content can increase further under unfavorable storage conditions. To avoid damage during soldering processes, it may be necessary to dry the PCBs in a suitable oven immediately before assembly or to store them in a dry atmosphere for a longer period (dry storage cabinet).
For flexible and rigid-flexible PCBs, this drying is mandatory before soldering!

The necessary drying parameters depend on the PCB design (copper areas), the drying equipment and the arrangement of the PCBs in it, the PCB material, the soldering process and the soldering parameters, the latter if necessary also multiple and combined from reflow, wave and partial soldering technology. These drying parameters must be determined and verified by the PCB assembler. Particular importance must also be attached to logistics and, in particular, waiting times, since the dried PCBs absorb moisture again from the environment. Packaging the PCBs in a moisture barrier bag (MBB) is not sufficient for the supply chain and storage.

PANEL SEPARATION NOTES:

Please avoid introducing stresses during the separation process by using appropriate separation processes to avoid damaging sensitive components as well as the PCB. For flexible and rigid-flex PCBs, the flex material must be cut at the lands before breaking to avoid delamination in the PCB. Predetermined breaking holes alone are not sufficient for this purpose! Please ask for these options, we will be pleased to make you an offer with a delivery array proposal.

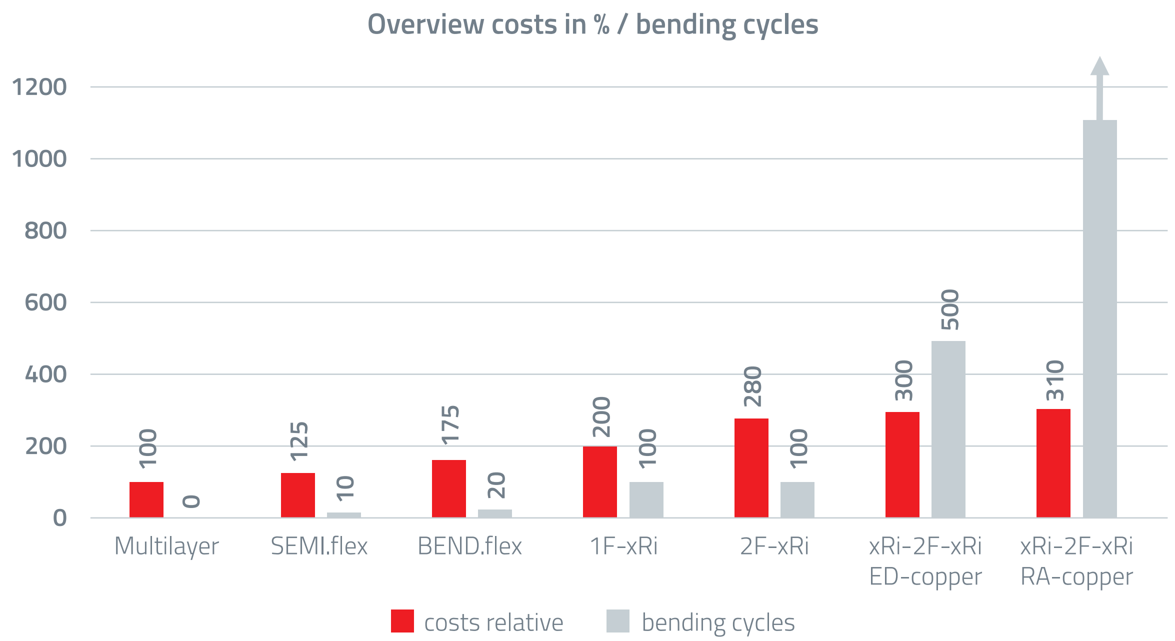
Under no circumstances should you use side cutters for snipping off webs!

! **For further information on the subject of "drying", please refer to**
www.we-online.com/dryingspecification
www.we-online.com/dryingprocess

11 COSTS, SYSTEM COSTS

Consideration of cost should take place primarily according to the decision to provide reliable electrophysical function of the printed circuit board. Design and cost are directly related. It is essential to distinguish between PCB costs and system costs. For a product, only the system costs are relevant, whereas for PCB buyers (only) the PCB costs are relevant. Thus, a possible conflict of interest is already inherent in the system if the performance and cost targets are not defined before the start of a development.

FLEXIBILITY AND COSTS IN RELATION – AN ASSESSMENT



System costs include not only PCB costs but also costs for components and assembly, installation, testing and commissioning, and quality costs. For the overall system, it can be favorable to use a higher-quality PCB technology and thereby achieve higher miniaturization, performance and reliability at the same or even lower system costs. This can often be seen, for example, in the use of RIGID.flex.

The following is a list of factors that influence PCB costs.

- PCB size and delivery array
  Here, early coordination with the PCB manufacturer on the optimal design of the delivery array can bring immense savings.
  Better utilization can also be achieved through modified nesting, narrower spacing and panel frame and / or alternative contour machining.
- Material selection and stackup
  Whenever possible, the use of standard stackups should be aimed for. This helps to avoid errors in the stackup and the use of exotic materials or material thicknesses. Standard materials are in stock and the PCB can usually be delivered more quickly.
  Approvals must not be forgotten here. For example, it can happen that no UL listing is available for designs that deviate from the standard and therefore no UL marking is possible.
- Design Structures
  The finer the structures, the lower the yield in production. This increases the costs. Here, too, the (economical) standard should be used as long as possible.
- Mechanical processing
  Small drills are more expensive and shorter. They can thus drill fewer plates in the stack and wear out faster. Laser drilling is cheaper for many vias. Complex milling contours with small diameters can significantly increase board costs.

- Multiple pressings are expensive because they require virtually the entire production run to be repeated several times. It is worth discussing via strategies and stackups with the PCB manufacturer at an early stage.
- Other cost drivers are edge metallization, via filling and thick gold layers.
- Dealing with X-out:
  A certain amount of bad parts should be allowed in multiple-delivery arrays. Too strict a delivery specification can jeopardize profitability and delivery capability, not to mention the unnecessary burden on the environment from wasted resources.

PCB costs and the design influence on the same cannot be generalized. However, it helps to know the trends. One way to clarify these trends, or even to compare design options, is to request comparative quotes from your preferred PCB manufacturer. We will be happy to do so, if we also get regular orders from you.

And finally, some good advice again:
Talk to your PCB manufacturer early and regularly and profit from his experience.
In Europe, this is still possible.

!

Hotline to our "FLEXperts":
Phone +49 7940 946-FLEX (3539)
flex@we-online.com
www.we-online.com/flex

Würth Elektronik GmbH & Co. KG
Circuit Board Technology
Salzstr. 21 · 74676 Niedernhall · Germany

Version 09.2023 / 504720 EN