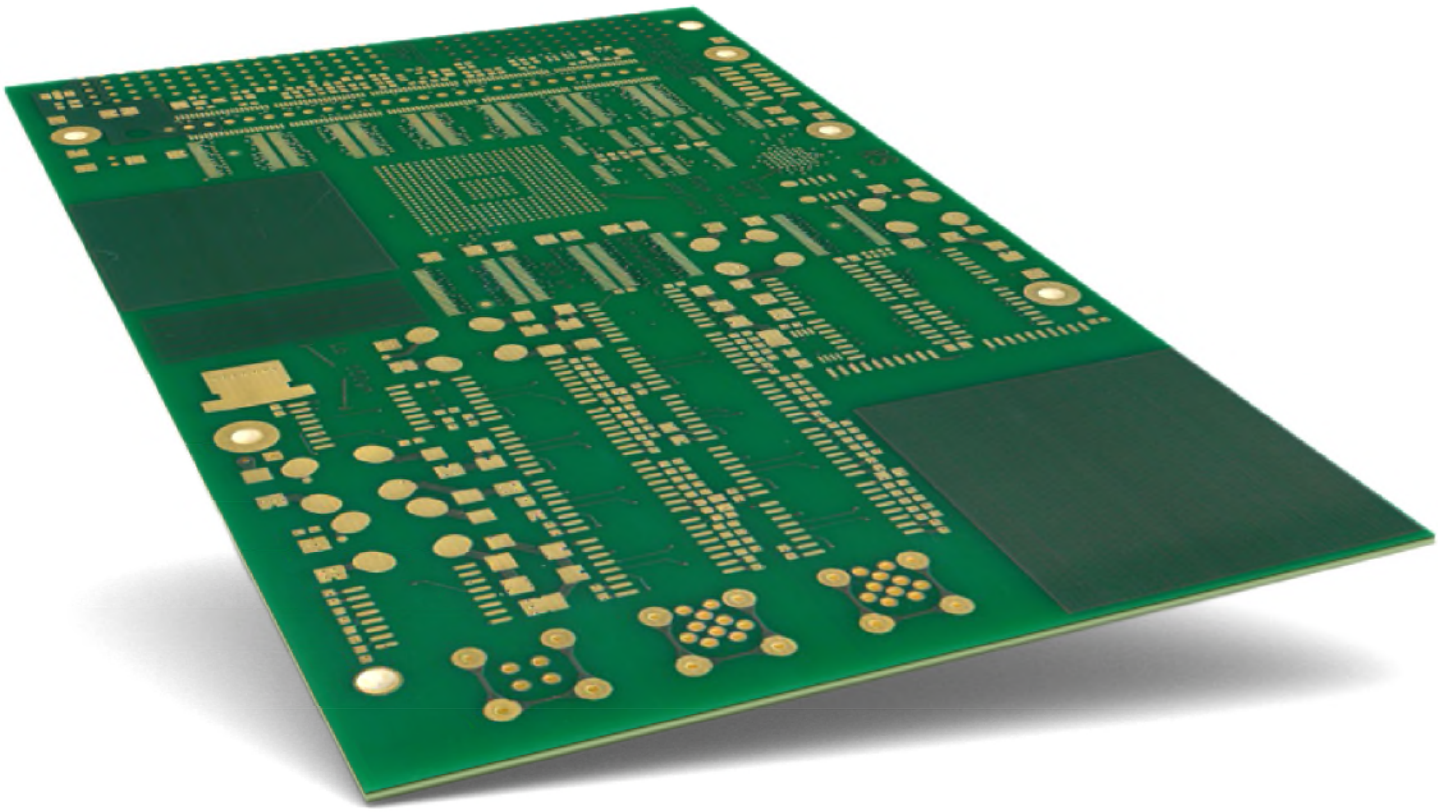
WEdirekt Design Guide

for PCBs from the online shop

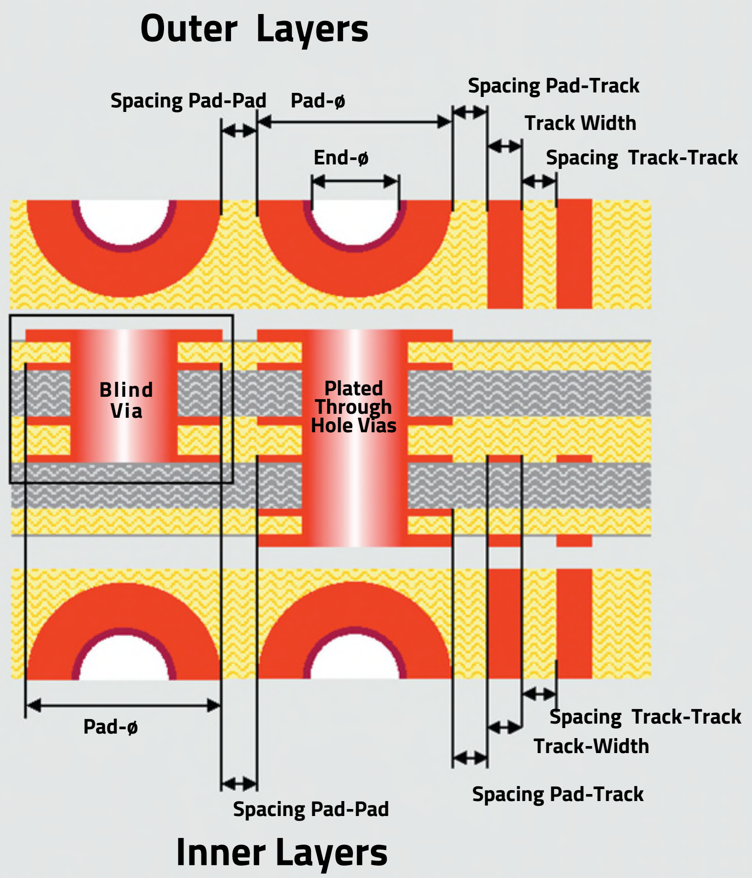


Conductive Pattern

Please note:
Aspect ratio for through-holes 1:10 (ratio of hole depth to drill diameter)

Blind Via Notes:

- Aspect ratio 1:0,8
- Finished diameter ≥ 0,15 mm taking the aspect ratio into consideration
- Layer stack-ups will be created depending on the layout (standard stack-ups are not valid)
- Please send us the blind vias as a separate file in the manufacturing dataset
- Possible surfaces: ENIG and immersion Sn



Outer Layers / Inner Layers Spacing	18 µm Finished Copper	35 µm Finished Copper	70 µm Finished Copper	105 µm Finished Copper
Track-Track	min. 85 µm*	min. 100 µm	min. 192 µm	min. 250 µm
Pad-Track	min. 85 µm*	min. 100 µm	min. 192 µm	min. 250 µm
Pad-Pad	min. 170 µm*	min. 170 µm	min. 192 µm	min. 250 µm
Track Width	min. 85 µm*	min. 100 µm	min. 150 µm	min. 150 µm

*Please note that a finished copper of 18µm is only possible in conjunction with etching technology, i.e. without galvanic metallization

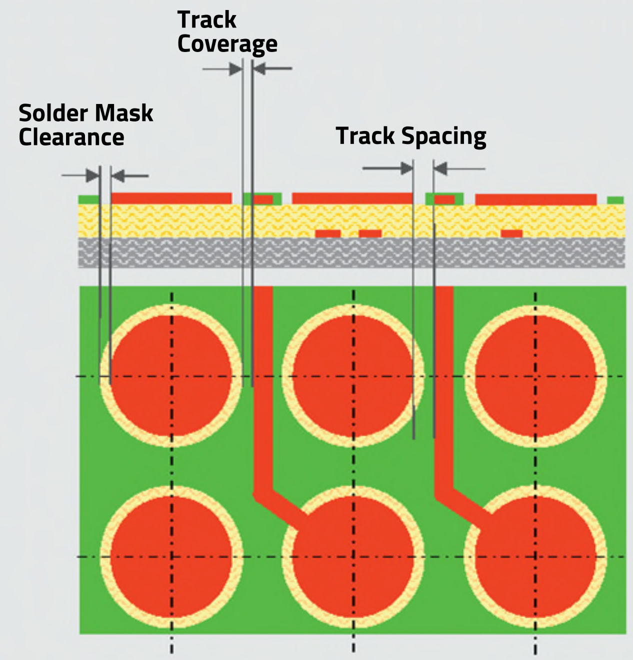
Plated Through Hole Vias*

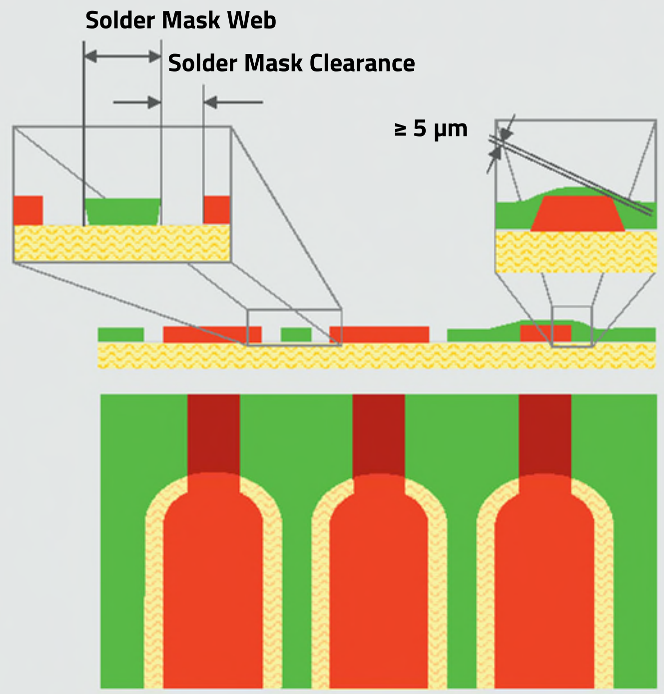
Pad-Ø	Drill Tool	End-Ø	Tolerance	Copper Clearance on Inner Layers	Solder Mask Clearance
0.60 mm	0.40 mm	**0.25 mm**	+0.10/-0.05 mm	≥ 0.80 mm	≥ 0.40 mm
0.55 mm	0.35 mm	**0.20 mm**		≥ 0.75 mm	≥ 0.35 mm
0.50 mm	0.30 mm	**0.15 mm**		≥ 0.70 mm	≥ 0.45 mm
0.45 mm	0.25 mm	**0.10 mm**		≥ 0.65 mm	≥ 0.40 mm

*Please note that an annular ring of 100 µm is only possible with a finished copper of max. 35 µm and up to 12 layers

Solder Mask and Silkscreen

Solder Mask





Solder Mask	
Clearance	**Track Coverage**
≥ 50 µm	50 µm
Solder Mask Web	**Solder Mask Clearance**
≥ 70 µm	See chart on page 2

Information about the Thickness of our Solder Mask	
Thickness on Base Material	**Thickness on Tracks**
20-45 µm	10-25 µm
Thickness on Edge of Tracks	
≥ 5 µm	

Silkscreen Design Parameter	Copper Thickness ≤ 80 µm	Copper Thickness > 80 µm
Line Width	≥ 80 µm	≥ 80 µm
Font Height	1.00 mm	1.50 mm
Distance to Solder Mask Opening	≥ 100 µm	≥ 100 µm

Gold Connector Definition

In general: Connector contacts must always be arranged in one line (no offset to the rear)

Chamfering

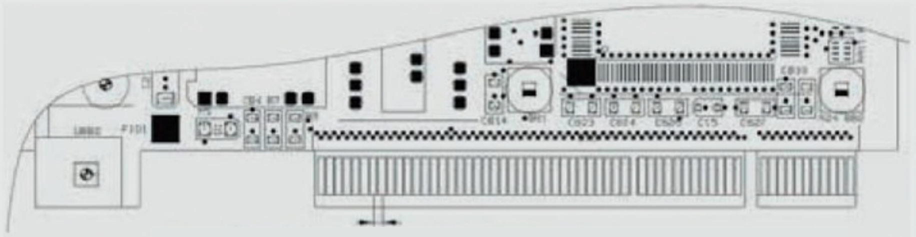
You can choose between 20° PCI and 45° ISA. TOP and Bottom is always chamfered during this process

Note: the depth of chamfer is based on the thickness of the PCB. The depth at 20° PCI and 45° ISA applies to a material thickness of 1,55 mm.

Electroplated Gold

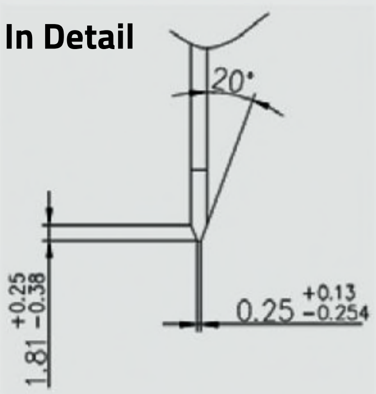
We usually produce electroplated gold in combination with connector contacts. A complate surface plating is not possible.

PCI 20°

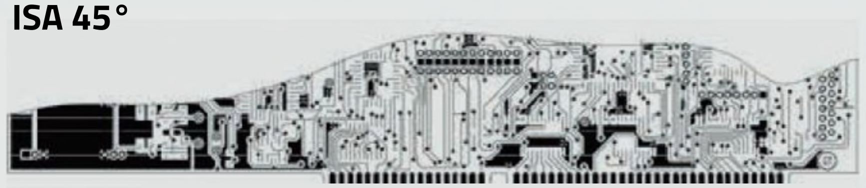


Distance from one connector contact to another: min. 1.27 mm

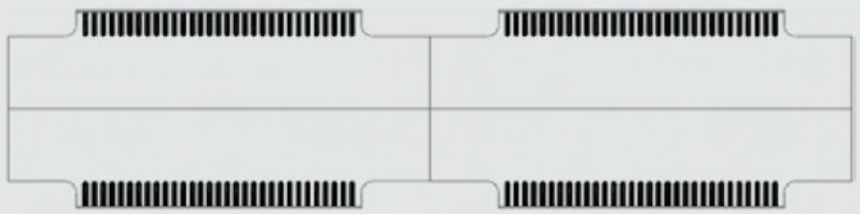
In Detail



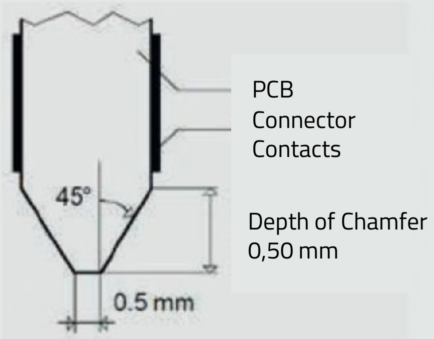
ISA 45°



Distance from one connector contact to another: min. 2.54 mm



In Detail



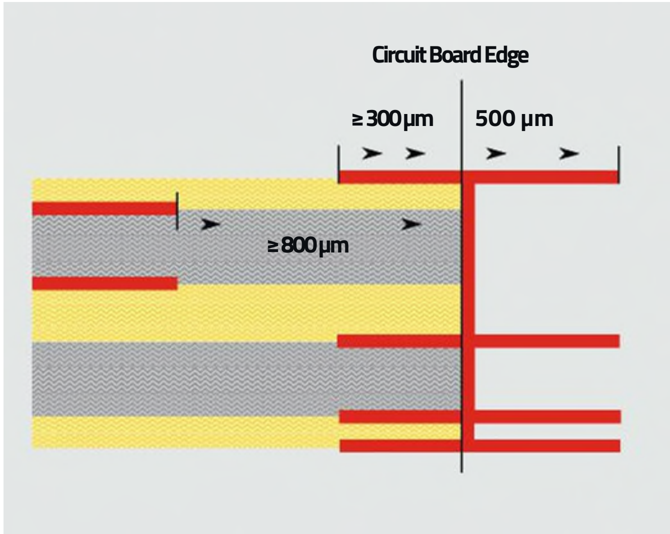
Edge Plating and Plugged Vias

Edge Plating

We offer edge plating for the outer edges of your PCBs. We kindly ask you to follow the design parameters to ensure a flawless production:

In your layout data, the circuit board edge must be marked with 500 µm of protruding copper to be edge plated. A connection of at least 300 µm must also be defined.

Layers that are not meant to be connected should have a clearance of at least 800 µm on the outer contour.

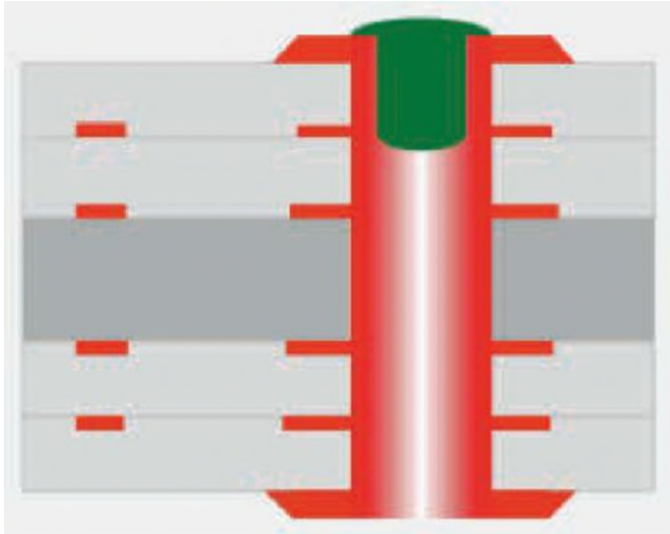


Plugged Vias

Plugged Via (according to IPC 4761 Typ III-a)

Clearance to neighbouring Solderable Surface when Plugging

Finished Diameter	Plugged Via Mask	Clearance between Mask and Solderable Surface
≤ 0,15 mm	0,40 mm	0,15 mm
≤ 0,25 mm	0,50 mm	0,15 mm
0,30 mm – 0,55 mm	End-Ø + 0,35 mm	0,15 mm
≤ 0,65 mm	End-Ø + 0,45 mm	0,15 mm



Remark for Vias in Solder Mask

Samples (Rigid PCBs)	Vias are always opened in the Solder Mask
HDI Microvia	Laser Vias will be covered with Solder Mask (depending on Specification)

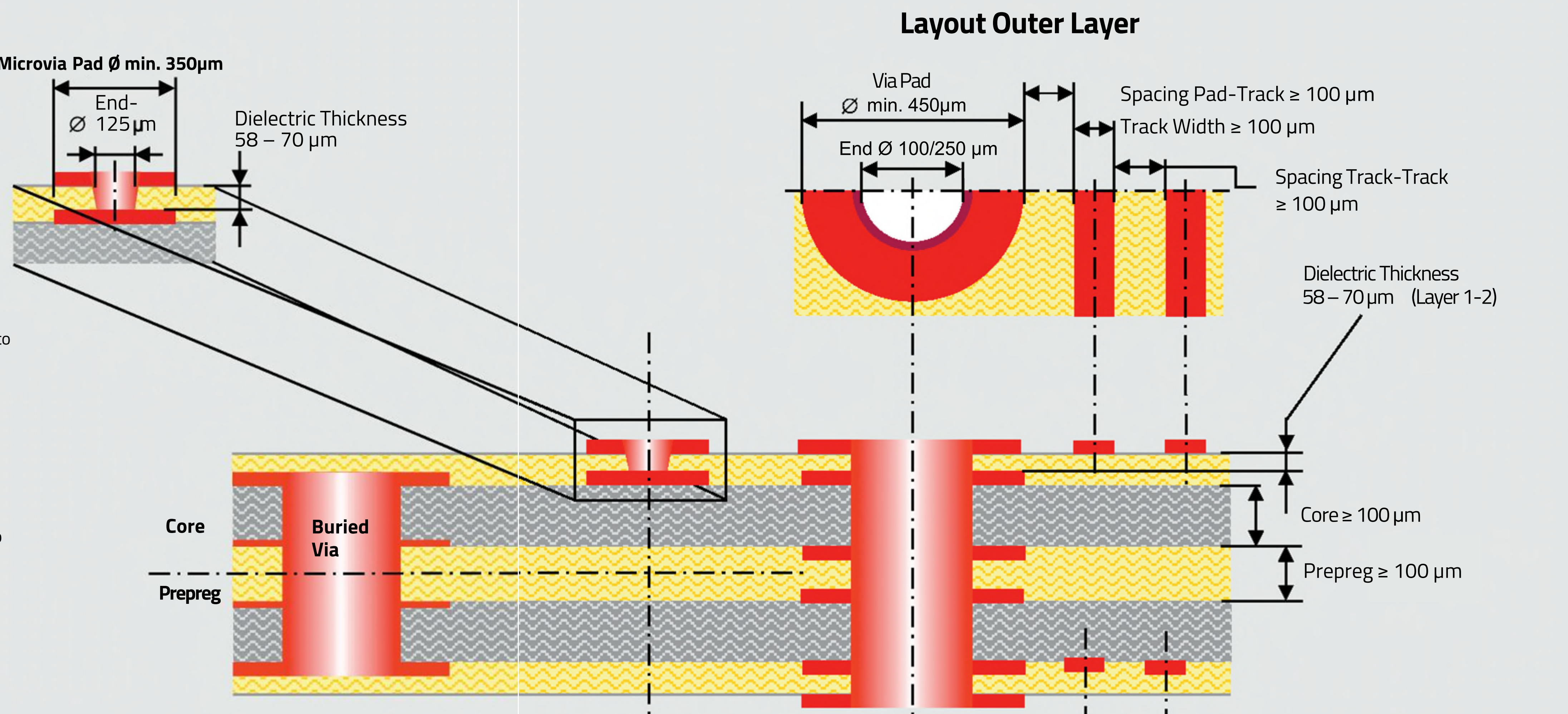
HDI Microvia
Standard Design Rules

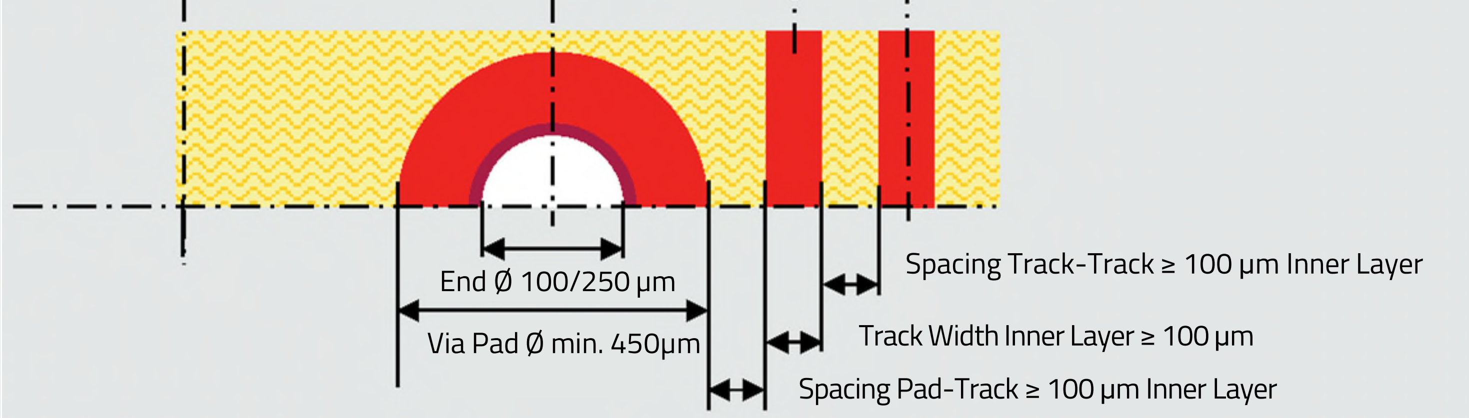
WÜRTH ELEKTRONIK
MORE THAN YOU EXPECT

Microvia
Aspect Ratio = 1 : 0.8
(Diameter / Depth)

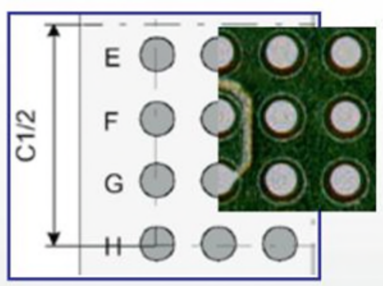
Achtung:

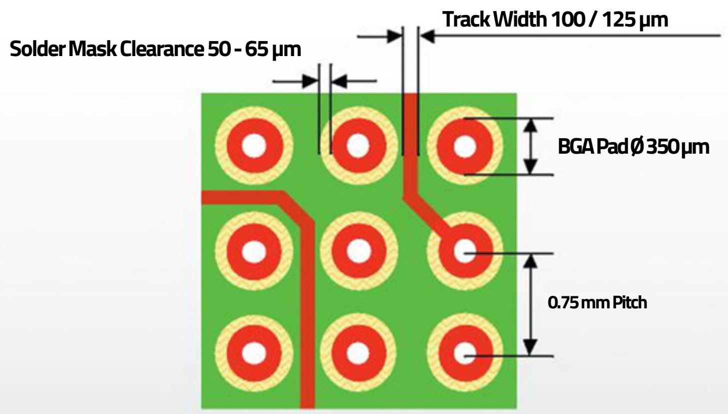
- Layer stack-ups are not selectable
- Microvias only possible from TOP and / or Bottom to the neighbouring inner layer
- 35 µm outer and inner layers according IPC Class 2
- For track width and spacing please see WEdirekt specification www.wedirekt.com
- Microvias are not filled
- Microvias pads are always changed to 350 µm
- Min. BGA Pitch spacing 750 µm
- Aspect ratio for buried vias 1:10 (ratio hole depth to drill diameter)
- Buried vias must be delivered as separate drill file
- Buried vias only through **all** inner layers possible





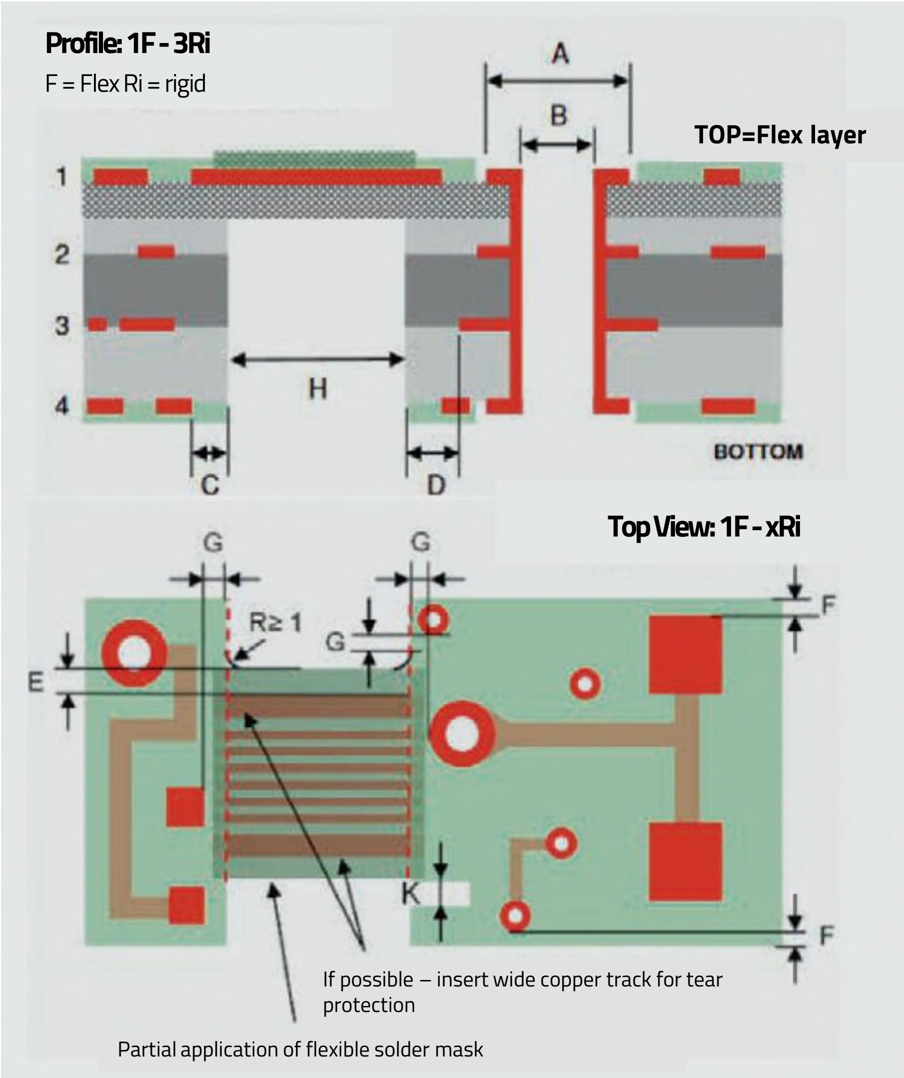
BGA 0.75 mm Pitch





Design Rules
Flex-rigid 1F-xRi

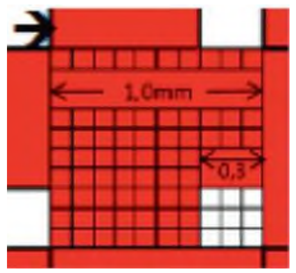
Application according to IPC 2223 Use A: Flex-to-install Marking according to UL94 and UL796 possible



	Symbol Description	Technical Standard
	Line Widths and Spacings	Please see page 2
A	Minimum Via Pad Diameter – Teardrops recommended	Please see page 2
B	Final Diameter of Through Hole Vias	Please see page 2
C	Spacing Cu – Outer Layer to Flex-rigid Transition (Bottom)	≥ 300 µm
D	Spacing Cu – Inner Layer to Flex-rigid Transition	≥ 500 µm
E	Distance of Track to the Flexible Contour	≥ 300 µm
F	Spacing exposed Cu – outside of Flex-rigid Transition	≥ 300 µm
G	Flexible Solder Mask: Spacing exposed Cu to Flex-rigid Transition (Top)	≥ 1000 µm
H	Length of the Flex Area	≥ 5 mm
K	Minimum Recess Width directly at the Flex Area	1,6 mm
"K"	Outline Manufacturing of Flex Area: No Scoring permitted!	
"ZIF"	ZIF Contacts Thickness Tolerance	± 0,05 mm

Basic Information:

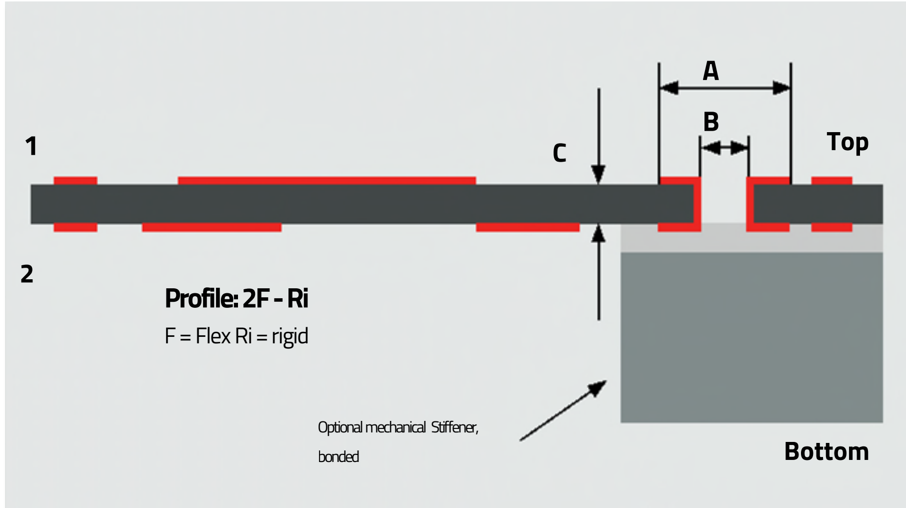
- Please consider the general standards, such as IPC or IEC.
- Lift-of-areas – attention: NO copper layout below the flex and NO vias permitted!
- Flex-rigid circuit boards must be dried before they are assembled and soldered.
- For the drying, copper openings in ground or reference layer are needed.
- Recommendation for Copper openings: 0.30 mm per 1 mm copper length (up to 70 µm Cu thickness):



- Flex-to-install bending radius: bending requirement according to IPC-2223
- 1 copper layer: Bending radius of at least 10 x total thickness (IPC-2223 Section 5.2.4.2)
- For use under more demanding conditions, please contact us

Design Rules
PURE.flex xF and
PURE.Flex with Stiffener xF – Ri

Application according to IPC 2223 Use A: Flex-to-install Without UL-Marking



Legende

Symbol	Description	Standard Demand
A	Minimum Via / Pad Diameter	Via Diameter B + 400 µm
B	Via Diameter	≥ 250 µm
-	Track Width	≥ 85 µm
-	Track Spacing	≥ 85 µm
-	Copper Clearance to Board Edge	≥ 300 µm
-	Number (x) of Copper Layers (xF)	1-2
-	Copper Thickness: see Layer Stack-Ups	18 µm or 35 µm
C	Thickness of Flexible Material (Polyimide	50 µm
	Thickness of Cold-Bonded Stiffener made of FR4 Material	0,15 mm
	Thickness of Glue for Stiffener	50 µm
-	Total PCB Thickness: see Layer Stack-Ups	120 µm (1F), 170 µm (2F), 300 µm (PURE.flex with Stiffener)
-	Bending Radius	3 mm
-	Maximum Number of Bending Cycles (in consideration of the Bending Radius)	100
	Solderable Surfaces	ENIG, Immersion Sn
	Important: Please do not place any Vias in the Bending Area	

Special Features regarding PURE.flex and PURE.Flex with Stiffener in the Delivery Panel:

- The distance between the individual PCBs in the delivery panel has to be ≥ 8.00 mm
- Circumferential frame of ≥ 7,50 mm is required
- In general, copper free areas in the frame of the delivery panel will be filled with copper balancing on top and bottom. This prevents warping of your PCB.
- The frame of the delivery panel and the complete bach of 1F stack-ups are generally coated with flexible solder mask..

Worth Knowing

WÜRTH ELEKTRONIK
MORE THAN YOU EXPECT

UL-Marking

The marking is introduced in the silkscreen or solder mask (unless another location is specified)

UL-Marking is not possible in the following cases:

- Exposed copper (without surface protection)
- On PURE.flex and PURE.flex with Stiffener PCBs

The following Data Sheets can be found on www.wedirekt.com

- Layer Stack-Ups
- Material Datasheets
- Solder Mask Datasheets
- DRU Files

Tolerances/Mechanical

Holes and Tolerances

Holes	Tolerances
Plated-through Holes	+0,10 / -0,05 mm
Non-plated-through Holes	+0,10 / -0,10 mm
Hole to Hole drilled in one run	+0,05 / -0,05 mm

Routing/Scoring and Tolerances

Routing an Scoring	Tolerances
Routing an Scoring	according to DIN EN ISO 2768 middle
Outline to non-plated Hole, Contour routed	+0,10 / -0,10 mm
Outline to non-plated Hole, Contour scored	+0,15 / -0,15 mm

Drill to Contour and Tolerances

Drill to Contour	Tolerances
Contour routed (0,50 – 6,00 mm)	+0,10/ -0,10 mm
Contour scored (0,50 – 6,00 mm)	+0,15/ -0,15 mm
Contour routed/scored (6,00 – 30,00 mm)	+0,20/ -0,20 mm
Contour routed/scored (≥ 30,00 mm)	+0,30/ -0,30 mm
Copper Image to Drill	+0,10/ -0,10 mm

Other Design Parameters

Copper Image	Routing	Scoring
Distance Copper to Outline	≥ 0.25 mm	≥ 0.45 mm For PCB Thickness 1.55 mm
Copper Clearance to Non-plated through Hole	≥ 0.25 mm circumferential	

Advantages at a Glance

- Orders with instant price calculation 24/7
- Prototypes up to 16 layer, no tooling costs
- High industrial quality in all established technologies
- Express production from 3 working days
- Production according to IPC A-600 Class 2
- The right stencil to your PCB
- 50 % discount on excess parts
- 5% discount on repeat orders
- Qualified service team
- Personal loyalty discount, based on the value of the previous year's turnover at WEdirekt (net, excluding shipping costs).

**Any questions?
Feel free to contact us**

E-Mail: info@wedirekt.de
Hotline: +9 755 388807-333

WEdirekt
c/o Würth Elektronik GmbH & Co. KG
Rudolf-Diesel-Str. 10
74585 Rot am See / Germany