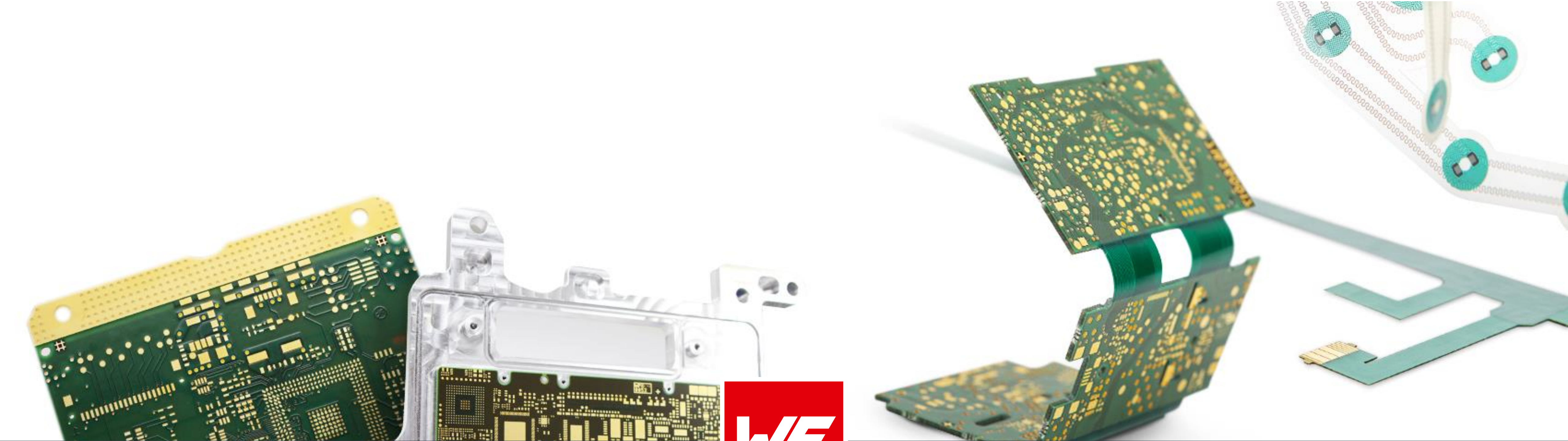


PCB PRODUCTION, PART 4
HIGH DENSITY INTERCONNECT PCBS

Michael Kress

WÜRTH ELEKTRONIK MORE THAN YOU EXPECT

AGENDA

1. High Density Interconnect (HDI) so far: MICROVIA.**hdi**
   - Production processes MICROVIA.hdi
   - Standard Stackups and Design Rules
   - Application examples
   - The limits of MICROVIA.hdi

2. HDI next generation: SLIM.**hdi**
   - Production processes SLIM.hdi
   - Standard Stackups and Design Rules
   - Application examples
   - Advantages and challenges of SLIM.hdi

3. Summary and outlook

Michael Kress
Head of TP plant Rot am See

PCB PRODUCTION, PART 4: HDI

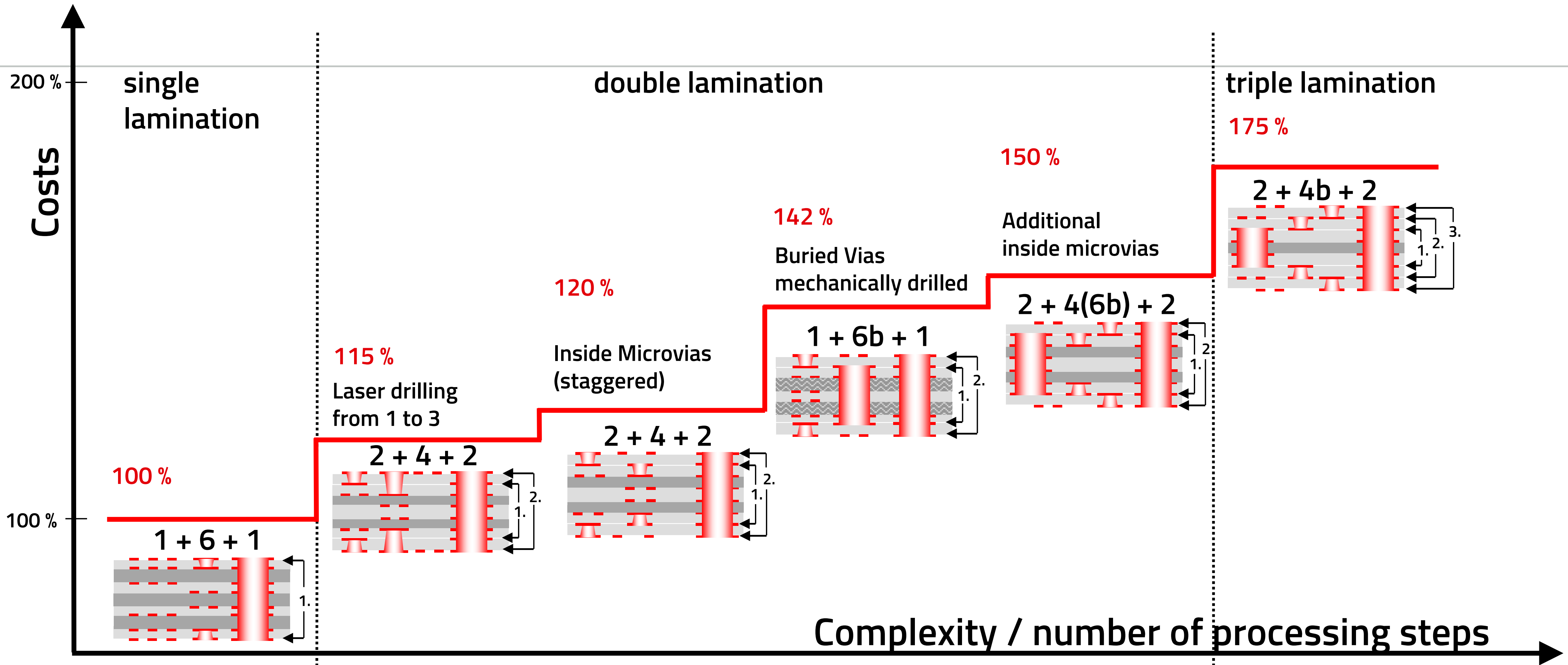
Poll: Mulitple choice with only one correct answer

What is the smallest BGA pitch you are currently using?

- BGA pitch greater than or equal to 0.8 mm
- BGA pitch greater than or equal to 0.5 mm
- BGA pitch greater than or equal to 0.4 mm
- BGA pitch greater than or equal to 0.3 mm
- BGA pitch less than 0.3 mm

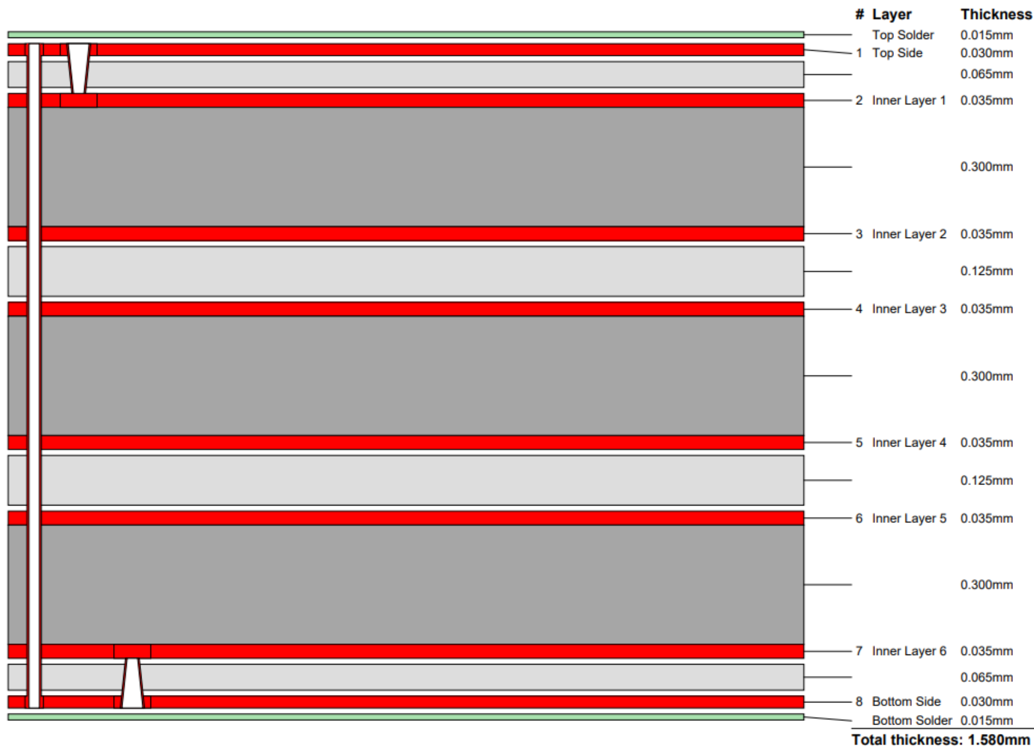
MICROVIA.HDI

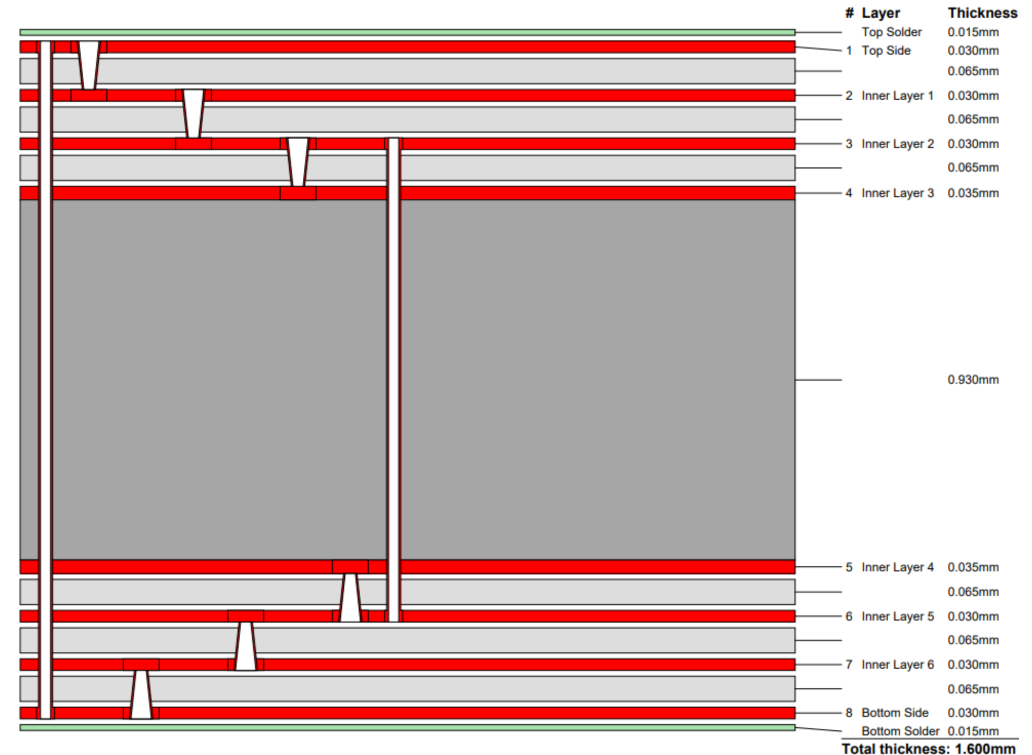
Production processes / cost relations



MICROVIA.HDI

Standard Stackups





- Find many other standard stackups on our Website

MICROVIA.HDI

Design Rules

microvia (for impedance controlled PCBs)
pad ø 325 µm
final ø 100 µm
dielectric thickness 85 – 110 µm

stacked microvia
Cu – filled up to max. 4 layers

standard microvia
pad ø 300 µm
final ø 100 µm
dielectric thickness 58 – 70 µm

outer layer layout
pad ø 550 µm
final ø 200 / 250 µm
distance pad / track ≥ 100 µm
track width ≥ 100 µm
distance track / track ≥ 100 µm
dielectric thickness 58 – 70 µm (layer 1-2)
dielectric thickness 55 – 68 µm (layer 2-3)
core material ≥ 100 µm
prepreg ≥ 100 µm

microvia aspect ratio = 1 : 0.8 (diameter / depth)

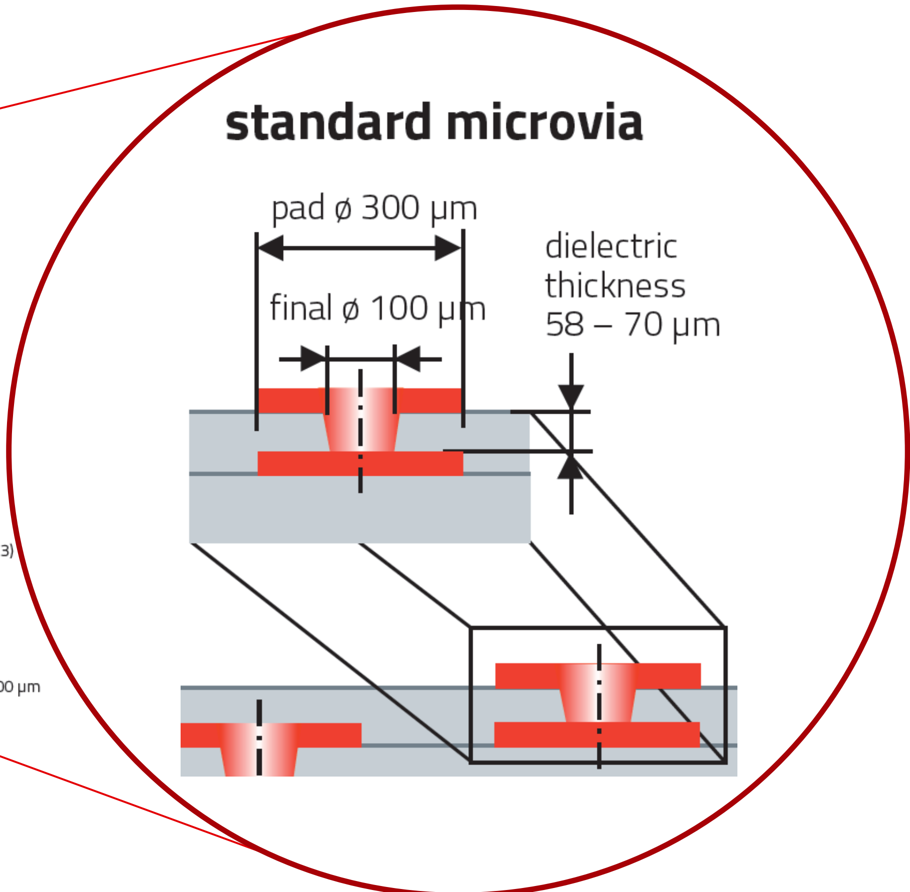
core material
prepreg
buried via
PTH

microvias layer 1 to 3
max. 55 µm
max. 18 µm
max. 50 µm
final ø 125 µm
pad ø 350 µm
inner layer clearance ø 550 µm

pitch ≥ 300 µm
staggered microvia

pitch ≥ 400 µm
pad ø 550 µm inner layer
final ø 200 / 250 µm
pad ø 550 µm
distance pad / pad ≥ 100 µm inner layer
distance track / track ≥ 100 µm inner layer
inner layer track width ≥ 100 µm
distance pad / track ≥ 100 µm inner layer

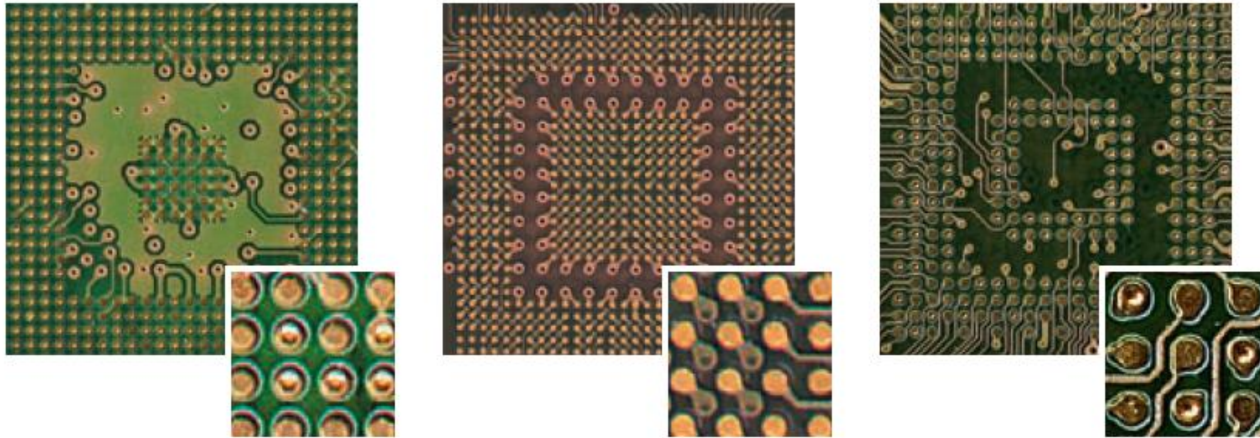
inner layer layout (up to 35 µm Cu thickness)



Download here

MICROVIA.HDI

The limits of MICROVIA.**hdi** - BGA-Pitch 0.50 mm

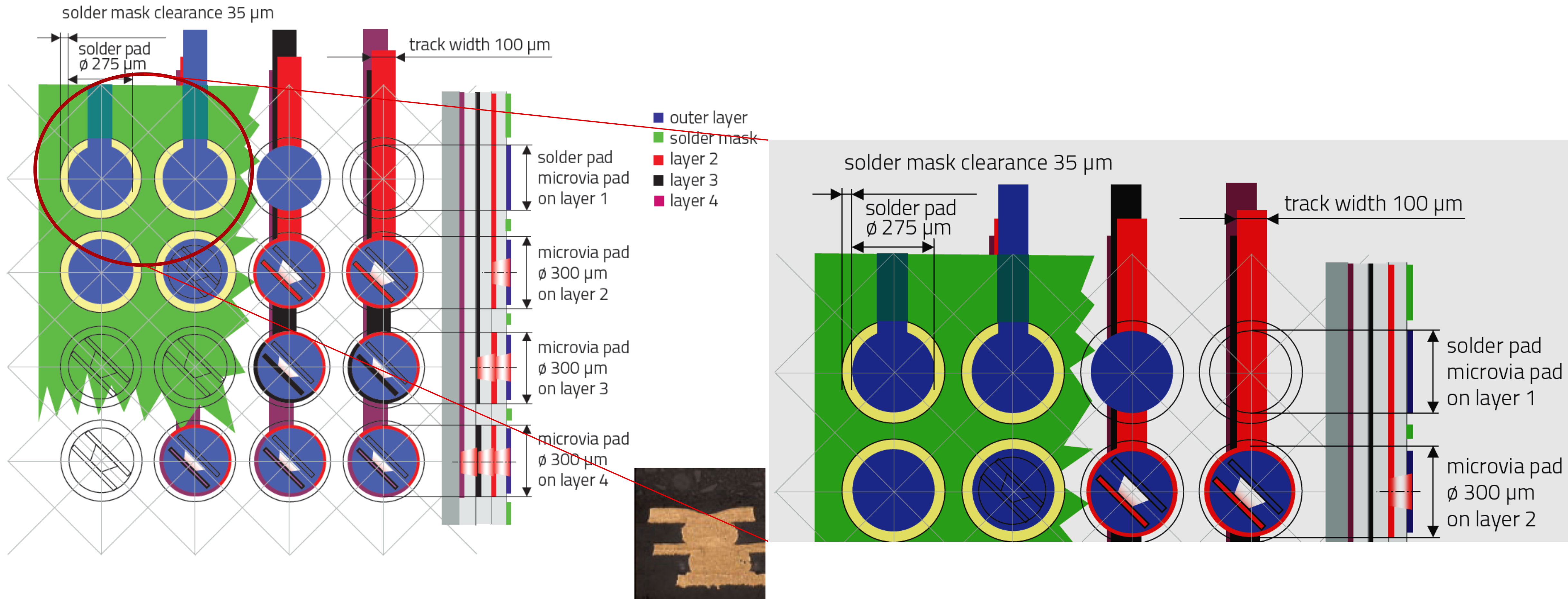


Var. 1: Via in pad   Var. 2: Dogbone   Var. 3: Via in pad

	Var. 1	Var. 2	Var. 3
BGA solder pad	300 – 330 µm	240 µm	275 µm
Solder mask clearance	50 µm	40 µm	35 µm
Microvia pad outer layers	≥ 300 µm	275 µm	275 µm
Microvia pad inner layers	275 µm	275 µm	275 µm
Track width / spacing outer layers	≥ 100 µm	80 / 90 µm	75 µm
Track width / spacing inner layers	75 µm	75 µm	75 µm

MICROVIA.HDI

The limits of MICROVIA.**hdi** - BGA-Pitch 0.40 mm



MICROVIA.HDI

The limits of MICROVIA.**hdi**

- Limit soldermask - WHY?
  - min. web width = 70 µm
  - min. distance solder mask web to pad edge = 35 µm

In total: Pad edge to pad edge min.140 µm, see sketch on the right side

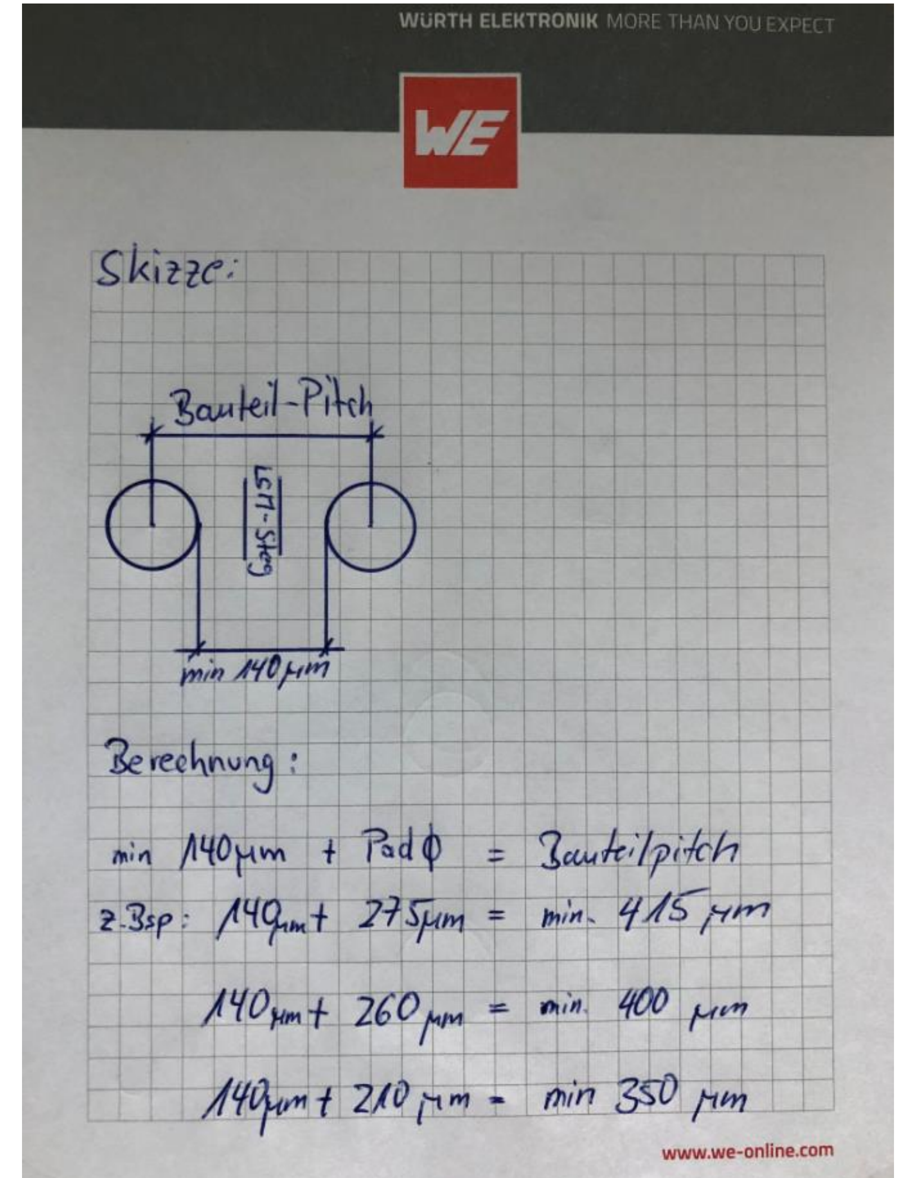
In our design example for 0.40 mm BGA pitch this means:

- maximum possible solder mask web at the narrow point:

  400 µm (pitch) - 275 µm (pad) - 2x (35 µm (solder mask clearance))

  = 50 µm solder mask web

Solder Mask		
	Standard	**Advanced**
Clearance	**≥ 50 µm**	35 µm
Coverage	**50 µm**	40 µm
Solder mask web	**≥ 70 µm**	–
Via-opening	final diameter +0,25 mm	



PCB PRODUCTION, PART 4: HDI

Poll: Mulitple choice with only one correct answer

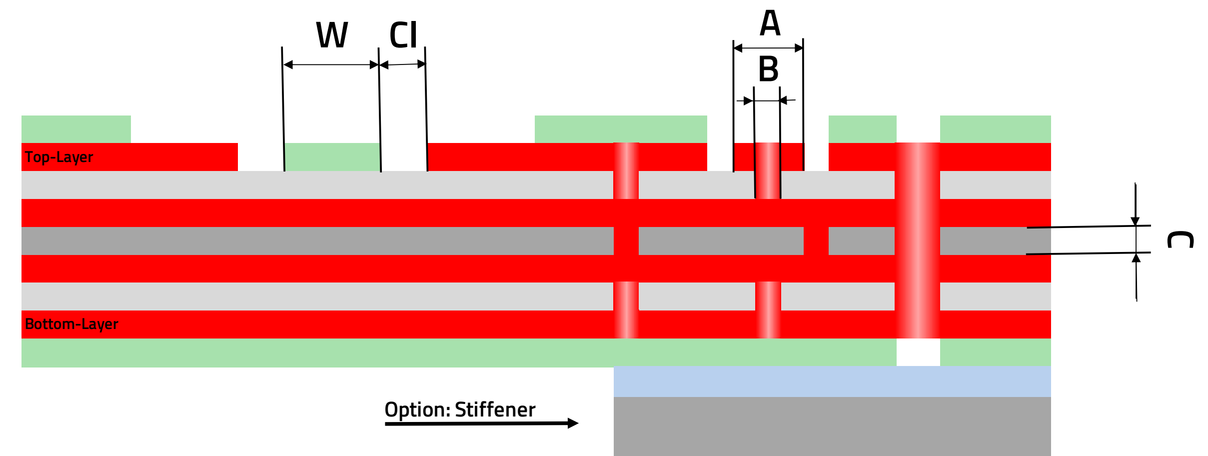
How much would reducing the pad Ø from 0.275 mm to 0.225 mm help in the layout?

- Helps very strongly (enables completely new solutions)
- Helps strongly
- Helps less
- Does not help at all
  (then we ask for additional explanation in the question field)

HDI NEXT GENERATION: SLIM.HDI

Profile SLIM.**hdi**?

- Anylayer-Microvia-Technology
- Very thin, rigid FR-4.1 materials (Tg150 °C, low halogen, filled) ultra-thin stackup
- Laser drilled microvias ∅ 85 µm in pad ∅ 200 µm
- Very thin copper layer thicknesses on all layers
- Optimal for routing of finest BGA components
- 75 µm structures
- Options
  - Impedance defined design
  - Stiffener
  - Solder carrier



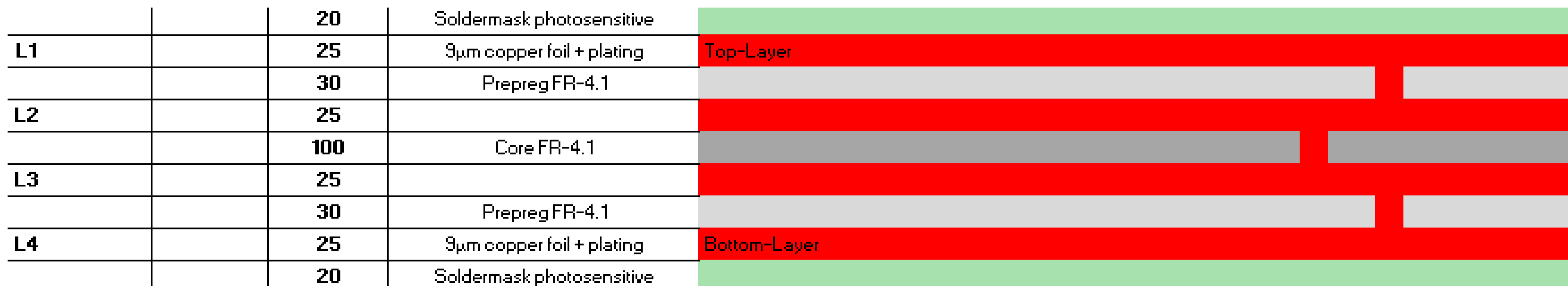
SLIM.HDI

Production processes Anylayer Microvia Technology SLIM.**hdi** 1-2b-1

- Inner layer production of core with laser drilled microvias L2 - L3 + copperfilling
- inner layer etching up to max. 25 µm copper thickness

Layer		Thickness	Material
L2		25	
		100	Core FR-4.1
L3		25	

- Pressing to 4-layer multilayer
- Laser drilling microvias Top - L2 and Bot - L3 with subsequent copperfilling
- Etching of outer layers up to max. 35 µm copper thickness (nominal 25 µm)
- Outer layer fabrication with solder mask and final surface finish



Layer		Thickness	Material	
		20	Soldermask photosensitive	
L1		25	9µm copper foil + plating	Top-Layer
		30	Prepreg FR-4.1	
L2		25		
		100	Core FR-4.1	
L3		25		
		30	Prepreg FR-4.1	
L4		25	9µm copper foil + plating	Bottom-Layer
		20	Soldermask photosensitive	

SLIM.HDI

Standard Stackups

SLIM.hdi ML4

PCB Thickness : 0,30 mm +/-0,05mm

Rigid area Structure		Rigid area Thickness	Material description	rigid area		Viatypes	Layer usage
		20	Soldermask photosensitive				
L1		25	9µm startup copper	Top-Layer			
		30	Prepreg HTG				
L2		25					
		100	FR4 HTG				
L3		25					
		30	Prepreg HTG				
L4		25	9µm startup copper	Bottom-Layer			
		20	Soldermask photosensitive				

SLIM.hdi ML8

PCB Thickness : 0,60 mm +/-0,05mm

Rigid area Structure		Rigid area Thickness	Material description	rigid area		Viatypes	Layer usage
		20	Soldermask photosensitive				
L1		25	9µm startup copper	Top-Layer			
		30	Prepreg HTG				
L2		25					
		30	Prepreg HTG				
L3		25					
		30	Prepreg HTG				
L4		25					
		100	FR4 HTG				
L5		25					
		30	Prepreg HTG				
L6		30					
		100	Prepreg HTG				
L7		25					
		30	Prepreg HTG				
L8		25	9µm startup copper	Bottom-Layer			
		20	Soldermask photosensitive				

- FR-4.1 Materials
  (Tg150 °C, low halogen, filled)
- Core material selection:
  - 0.06 mm, starting copper 12 µm
  - 0.10 mm, starting copper 12 µm
- Final thickness by number of layers

layer count	total thickness
4 layers 1-2b-1	≤ 0.35 mm
6 layers 2-2b-2	≤ 0.45 mm
8 layers 3-2b-3	≤ 0.60 mm

WE

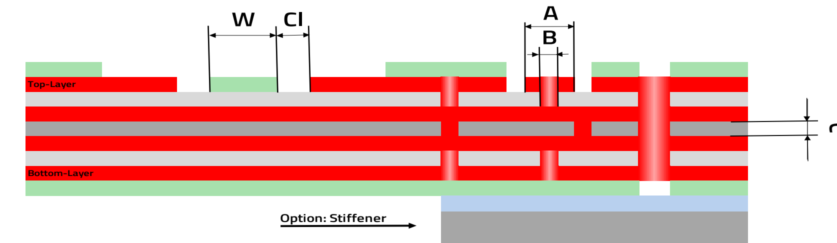
SLIM.HDI

Design Rules

Symbol	Desicription	Technical Standard	Advanced requirements
	Line widths and spacing	75µm/75µm only Microvias	75µm/100µm with PTH
A	Minimum pad diameter for microvia (for PTH)	225µm (-)	200µm (400µm)
B	Finished hole diameter of lasered microvia	85µm	85µm
-	Distance copper to outline	≥300µm	≥225µm
-	Number of copper layers in total	4 bis 8	
C	Thickness of core (FR4.1 - TG150, halogenfree, filled)	100µm	60µm
-	Thickness of cold-bonded stiffener made of FR-4.0 material	0,8 mm	1,00mm – 1,55mm
	Thickness of cold-bonded solder carrier made of FR-4.0	0,8mm	0,8mm
-	Thickness of glue for stiffener or solder carrier	50µm	
W	Minimum bridge width photosensitive solder mask	70µm	50µm
Cl	Minimum clearance of copper pad with solder mask, circumferential	40µm	35µm

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

DOWNLOAD Design Rules here



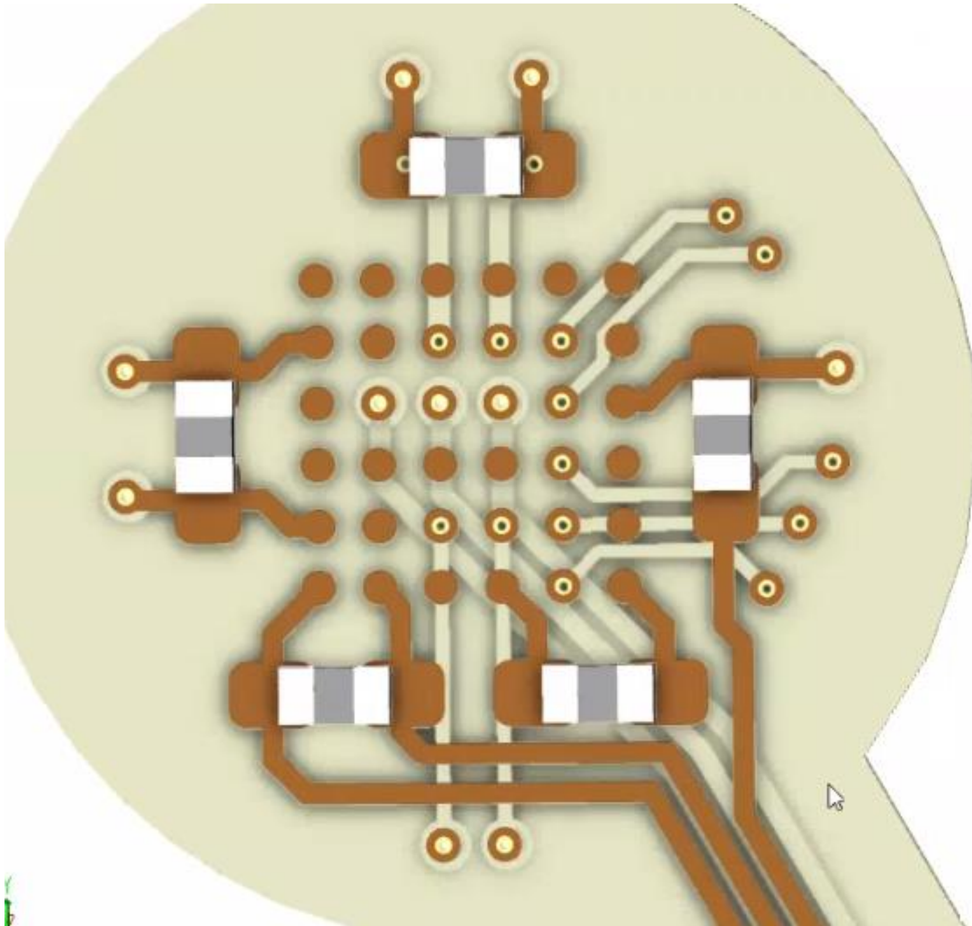
- Land Pad acc. IPC

0201	N/A	0.4 mm [9.84 mil]	0.45 mm [15.7 mil]
01005	N/A	0.200 mm [7.87 mil]	0.300 mm [11.81 mil]

→ Microvia-in Pad possible

SLIM.HDI

EDA View BGA 0.35 mm



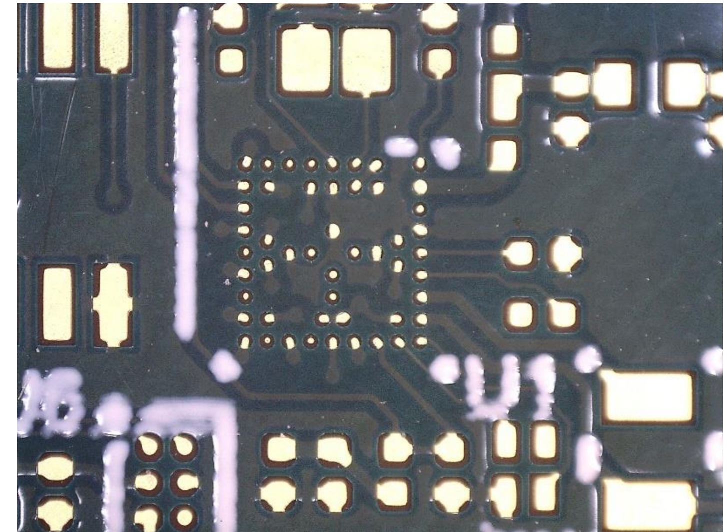
SLIM.HDI

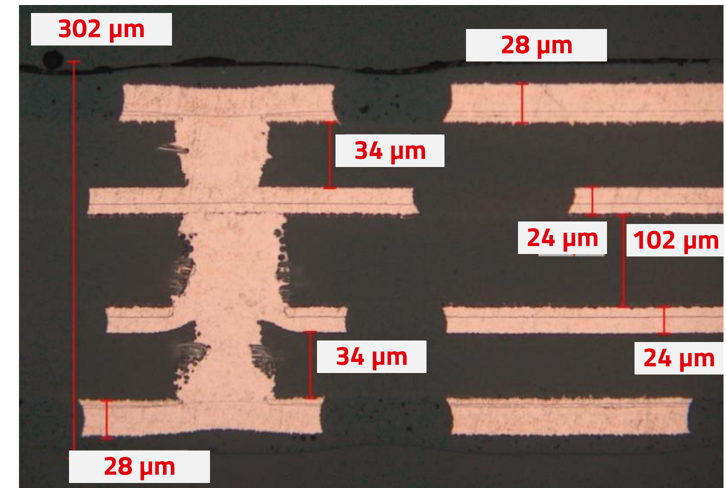
Design Parameters BGA 0.35 mm with solder mask

Comparison Design Rules SLIM.**flex** / SLIM.**hdi**:

- **BGA pad diameter Ø:** 210 µm / 225 µm
- **Solder mask web:** 70 µm / 55 µm
- **Solder mask clearance:** 35 µm / 35 µm
- **Laser drilled Microvia Ø:** 85 µm
- **Lines / spaces:** 75 µm

Only microvia-in-pad technology including copperfilling possible!



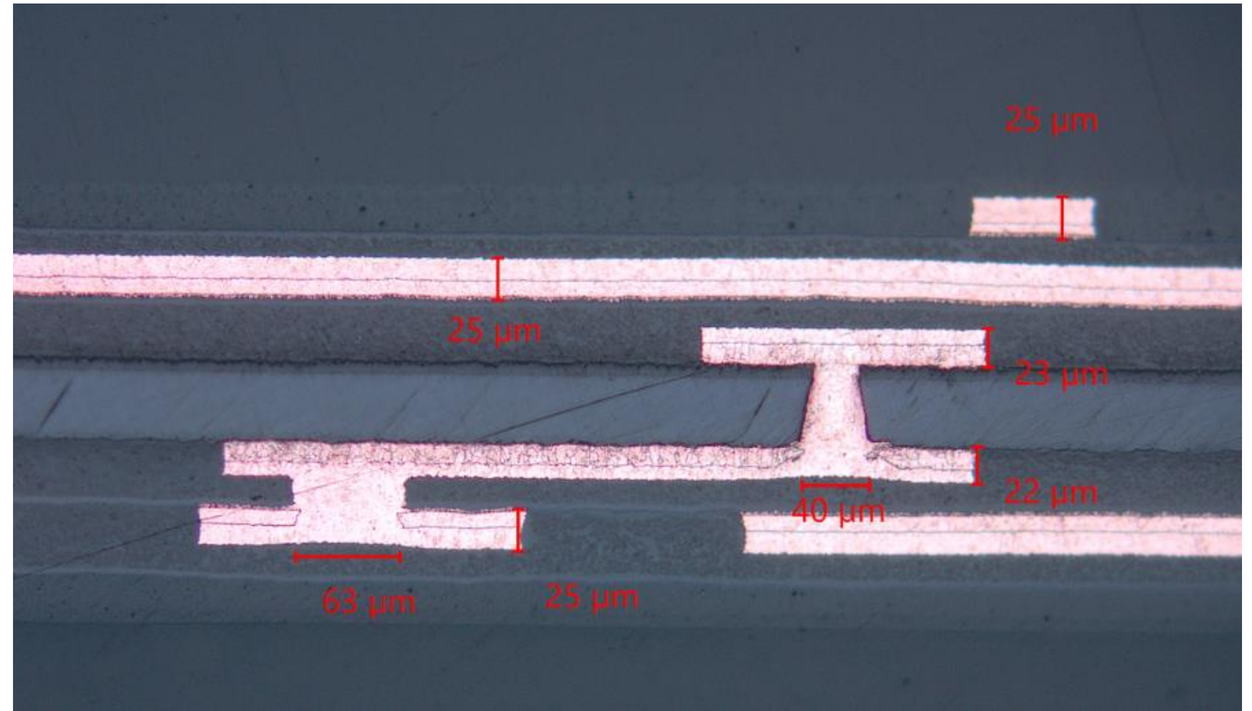


SLIM.HDI

Design Parameters BGA 0.30 mm pitch with solder mask, Technology SLIM.**flex**

Design Rules only applicable for SLIM.**flex**:

- **BGA pad diameter Ø:** 180 µm
- **Solder mask web:** 50 µm
- **Solder mask clearance:** 35 µm
- **Laser drilled Microvia Ø:** 60 µm
- **Lines / spaces:** 75 µm



Only microvia-in-pad technology including copperfilling possible!

PCB PRODUCTION, PART 4: HDI

Poll: Mulitple choice with only one correct answer

Have you recently processed PCBs with a total thickness of less than 0.50 mm?

- Yes, happens more often lately
- Yes, but they are still isolated cases
- No, has not been an issue lately

CHALLENGES IN THE ASSEMBLY OF SLIM.HDI

Very thin stackups → handling in the assembly process difficult

Solution 1

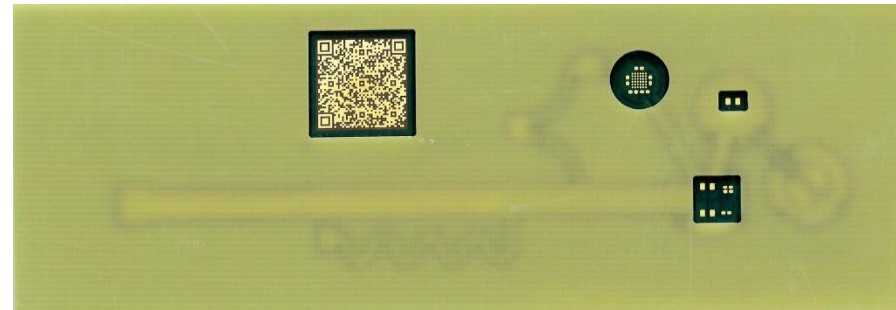
- FR4 solder carrier 0.80 mm

- This results in single-sided assembly!

  What to do if 2-sided assembly becomes necessary?

Solution 2

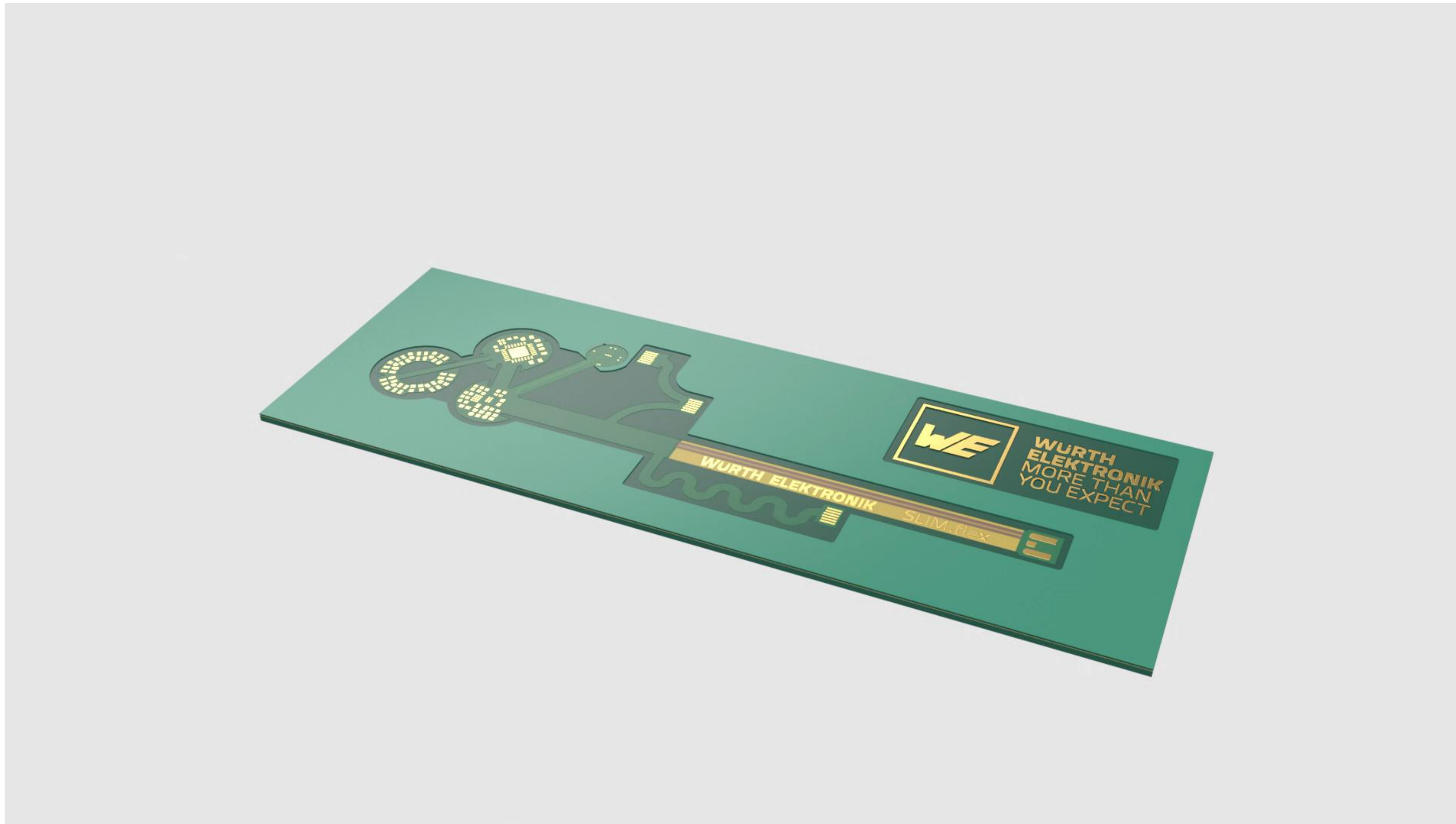
- Opening of the FR4 solder carrier
  Solder paste printing via step stencil / dispenser technology.



- Further option:
  FR4 reinforcement 0.1 - 0.15 mm only in the delivery panel

SOLDER CARRIER

Short Film about the System Solution PCB on FR4 Solder Carrier



SUMMARY

HDI = MICROVIA.**hdi** & SLIM.**hdi**

- Limits for standard MICROVIA.**hdi** design: BGA component pitch 0.40 mm
- From BGA component pitch 0.35 mm: SLIM.**hdi** offers finer design parameters
- From BGA component pitch 0.30 mm: SLIM.**flex** technology
- Challenges in the assembly process with PCB thicknesses ≤ 0.50 mm
  - Solder carriers for the assembly process, talk to us

Outlook:

SLIM.**hdi (with rigid materials) for BGA component pitch 0.30 mm in development**

THANKS FOR YOUR ATTENTION

Basics of printed circuit board production
High Density Interconnect (HDI)

WE