

These design rules apply to:

Anylayer Microvia PCBs made of base material ANSI GPY/42 for substrates and modules or of FR-4.1.

- With 4 to 10 layers, stacked and staggered Microvias.
- Optionally with PTH (Plated Through Hole) for extra charge with restricted design rules.
- Optionally with glued mechanical stiffener (-Ri = Stiffener) or solder carrier (for extra charge).
- No UL-marking. All materials are UL-listed.

Examples:







SLIM.hdi 1-2b-1 Standard: Only Microvias used SLIM.hdi (1-2b-1)PTH Option: Solder carrier, PTH SLIM.hdi (2-2b-2)PTH-Ri Options: Stiffener, PTH

Nomenclature: x = number of sequential build-up copper layers, Ri = Stiffener or solder carrier out of FR-4.0

Layer count	PCB total thickness without Stiffener/solder carrier	Nomenclature	Material
4	≤ 0,35 mm	SLIM.hdi 1-2b-1	FR-4.1 / GPY/42
6	≤ 0,45 mm	SLIM.hdi 2-2b-2	FR-4.1 / GPY/42
8	≤ 0,60 mm	SLIM.hdi 3-2b-3	FR-4.1 / GPY/42
10	≤ 0,58 mm	SLIM.hdi 4-2b-4	GPY/42

Basic instructions

- Please comply with general standards, such as IPC or IEC.
- We will be happy to create the optimal delivery panel for you (best price!).
- ANSI grade GPY/42 is a higher quality base material. It is also cycle-proof like FR4.1, but also offers lower thermal expansion in the x and y axes. The tendency to twist and warp is considerably reduced.
- PTH (Plated Through Hole) should not be used.



Material specifications

Material	Standard	Sheet#	Description	Application, keywords (IPC-4101)	
Base material	IPC-4101	128	ANSI: FR-4.1/128	thermocycle-proof	
			Tg 150 °C	Epoxy / Woven E-Glass	
				High Decomposition Temperature	
				Low Z-axis CTE, Fillers	
				Low Halogen Content	
Base material	IPC-4101	42	ANSI: GPY/42	Semiconductor packages, moduls	
			Tg 260 °C	Low X-/Y-axis CTE, thermocycle-	
			CTE X/Y-axis 4-6 ppm/K	proof, considerably reduced	
				tendency to twist and warp	
				Polyimide / Epoxy / Woven Glass	
				High Reliability, Fillers	
Soldermask	IPC-SM840		green, photosensitive	Standard	
	JIS C 5012				

Standard Stackups

Standard stackups see <u>www.we-online.com/hdi-stackups</u>

SLIM.hdi 1-2b-1								
	PCB Thickness :	0,28 mm +/-0,05	5mm					
	•					Impedance		
Rigid area Thickness	Material description	rigid area		Viatypes	Layer usage	Er	Z[Ohm] / Line / Space	
20	Soldermask photosensitive							
30	9µm copper foil + plating	Top-Layer						
35	Prepreg							
25								
60	Core							
25								
35	Prepreg							
30	9µm copper foil + plating	Bottom-Layer						
20	Soldermask photosensitive							



customer									
pcb name									
WE-number									WÜRTH
engineer									
date									MORE THAN
					SLIM.hdi_substrate	4 -2b-4			YOU EXPECT
			PCB Thickness :	0,58 mm +/-0,0	5mm				
							· · · · ·		
Rigid area Structure	1	Rigid area Thickness	Material description	rigid area		Viatypes	Layer usage	Ter Z[Ohm] / Line / Space	
		20	Soldermask photosensitive						
ы		25	9µm copper foil + plating	Top-Layer					
		30	Prepreg						
L2		25							
		30	Prepreg						
L3		25							
		30	Prepreg						
L4		25							
		30	Prepreg						
L5		25							
		50	Core						
L6		25							
		30	Prepreg						
L7		25							
		30	Prepreg						
L8		25							
		30	Prepreg						
L9		25							
		30	Prepreg						
L10		25	9µm copper foil + plating	Bottom-Layer					
		20	Soldermask photosensitive						

Standard design

- 1. FR4.1-core, optional ANSI GPY/42 core
- 2. Sequential build-up of anylayer pairs with base material
 - a. FR-4.1 prepregs 30 µm / 50 µm / 70 µm
 - b. Option: GPY/42 prepreg 30 µm
- 3. Base Copper thickness 9 µm + electroplating
- 4. Photosensitive solder resist green
- 5. Standard vias are laser drilled microvias as anylayer connections, plating thickness according to IPC-6012
- 6. Outline lasered or milled, smallest milling diameter 1.6 mm. V-scoring not permitted!
- 7. Solderable surface ENIG (electroless Nickel immersion Gold)
- 8. Packaged in ESD shrink wrap



Stackup SLIM.hdi 1-2b-1-Ri

Standard: Microvias only



Symbol	Desicription	Technical Standard	Advanced requirements		
	Line width and spacing $ ightarrow$ microvias only	75 μm / 75 μm			
А	Minimum pad diameter for microvia	225 µm	200 µm		
В	Finished hole diameter of lasered microvia, typical	85 µm	85 µm		
	For all Pad-connections Teardrops are recommended!				
-	Distance copper to outline	≥ 300 µm	≥ 225 µm		
-	Number of copper layers in total	4 to 8			
С	Thickness of core (FR4.1 - TG150, halogenfree, filled)	100 µm	60 µm		
-	Thickness of cold-bonded stiffener made of FR-4.0 material	0.8 mm	1.00 mm – 1.55 mm		
	Thickness of cold-bonded solder carrier made of FR-4.0	0.8 mm	0.8 mm		
-	Thickness of glue for stiffener or solder carrier	50 µm			
W	Minimum bridge width photosensitive solder mask	70 µm	50 µm		
CI	Minimum clearance of copper pad with solder mask, circumferential	40 µm	35 µm		



Cumhal	Decision	Technical	Advanced
Symbol	Desicription	Standard	requirements
	Line width and spacing $ ightarrow$ PTH and Microvias	75 µm / 100 µm	-
A(PTH)	Minimum pad diameter for PTH	450 µm	400 µm
B(PTH)	Finished hole diameter of PTH, typical	200 µm	150 µm
	For all Pad-connections Teardrops are recommended!		
	Non functional / non-used pads do NOT remove!!		
W	Minimum bridge width photosensitive solder mask	70 µm	-
CI	Minimum clearance of copper pad with solder mask, circumferential	40 µm	-

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