WÜRTH ELEKTRONIK
MORE THAN YOU EXPECT

Design Rules

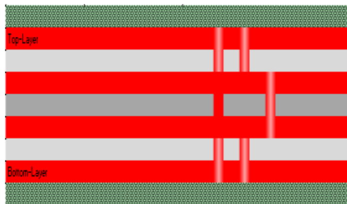
SLIM.**hdi** x-2b-x and (x-2b-x)PTH

These design rules apply to:

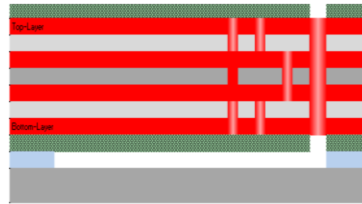
HDI anylayer Printed Circuit Boards with 4 to 8 layers, stacked and staggered Microvias

- 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.

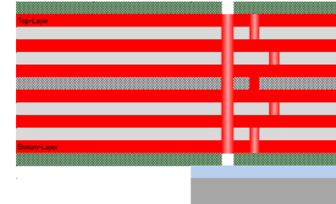
Examples:



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 sequentiel build-up copper layers, Ri = Stiffener or solder carrier out of FR4

Layer count	PCB total thickness without Stiffener/solder carrier	Description
4	≤ 0,35 mm	SLIM.hdi 1-2b-1
6	≤ 0,45 mm	SLIM.hdi 2-2b-2
8	≤ 0,60 mm	SLIM.hdi 3-2b-3

<u>Basic instructions</u>

- Please comply with general standards, such as IPC or IEC.
- We will be happy to create the optimal delivery panel for you (best price!).

Design Rules

SLIM.**hdi** x-2b-x and (x-2b-x)PTH

Material specifications

Material	Standard	Spec. sheet	Description	Application
Base material	IPC-4101	128	FR-4.1 Tg150 °C	thermocycle-proof, halogen free, filled, low CTE(z)
Soldermask	IPC-SM840 JIS C 5012		green, photosensitive	Standard

Standard Stackups

Standard stackups see www.we-online.com/hdi-stackups

customer	
pcb name	
WE-number	
engineer	
date	

WÜRTH ELEKTRONIK MORE THAN YOU EXPECT

SLIM.hdi 1-2b-1

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

Rigid area Structure	Rigid area Thickness	Material description	rigid area	Viatypes	Layer usage	Impedance Er	Impedance Z[Ohm] / Line / Space
	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					

Standard design

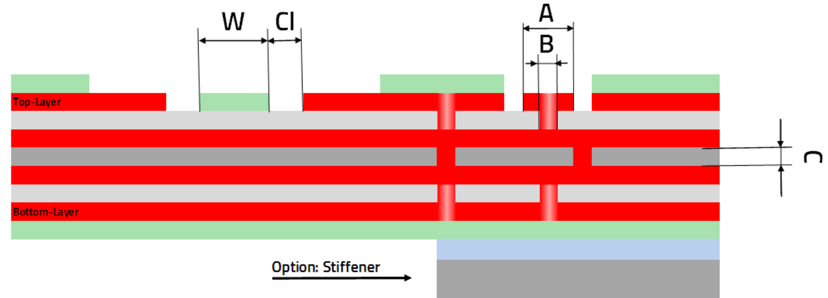
1. FR4.1-core, sequential build-up of anylayer pairs with prepregs 30 µm / 50 µm / 70 µm
2. Base Copper thickness inner layers 17 µm, exterior layers 9 µm + electroplating
3. Photosensitive solder resist green
4. Standard vias are laser drilled microvias as anylayer connections, plating thickness according to IPC-6012
5. Outline lasered or milled, smallest milling diameter 1.6 mm. V-scoring not permitted!
6. Solderable surface ENIG (electroless Nickel – immersion Gold)
7. Packaged in ESD shrink wrap

Design Rules

SLIM.**hdi** x-2b-x and (x-2b-x)PTH

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

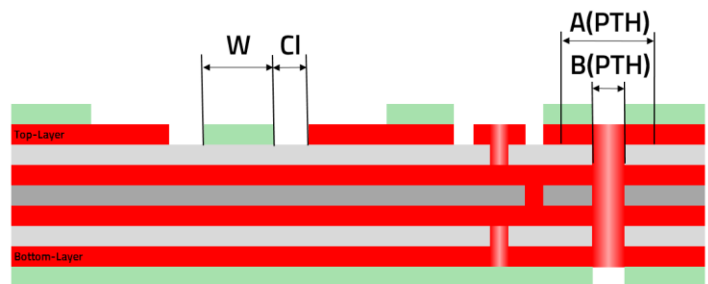
Standard: Microvias only



Symbol	Desicription	Technical Standard	Advanced requirements
	Line width and spacing → microvias only	75 µm / 75 µm	
A	Minimum pad diameter for microvia	225 µm	200 µm
B	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	
C	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
Cl	Minimum clearance of copper pad with solder mask, circumferential	40 µm	35 µm

Stackup SLIM.**hdi (1-2b-1)PTH**

Option: Microvias and PTH
only deviating parameters



Symbol	Desicription	Technical Standard	Advanced requirements
	Line width and spacing → 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	-
Cl	Minimum clearance of copper pad with solder mask, circumferential	40 µm	-

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