

1. Used material – S1000H

2. Number of layers – 4

3. Board thickness – 1.0mm +/-10%

4. Thickness of outer layer (Cu) – 43um include copper plating

5. Thickness of inner layers (Cu) – 18um

6. Solder Mask – Two sides, Green

7. Silkscreen – No

8. Type of coating – ENIG

9. Edge Connector – Hard Gold Au 0,2–0,5um/Ni 5um

10. Electrical Test on both side (use Adapter_MPCle_to_M2_E.ipc file)

11. Min dia of plated hole – 0.2mm

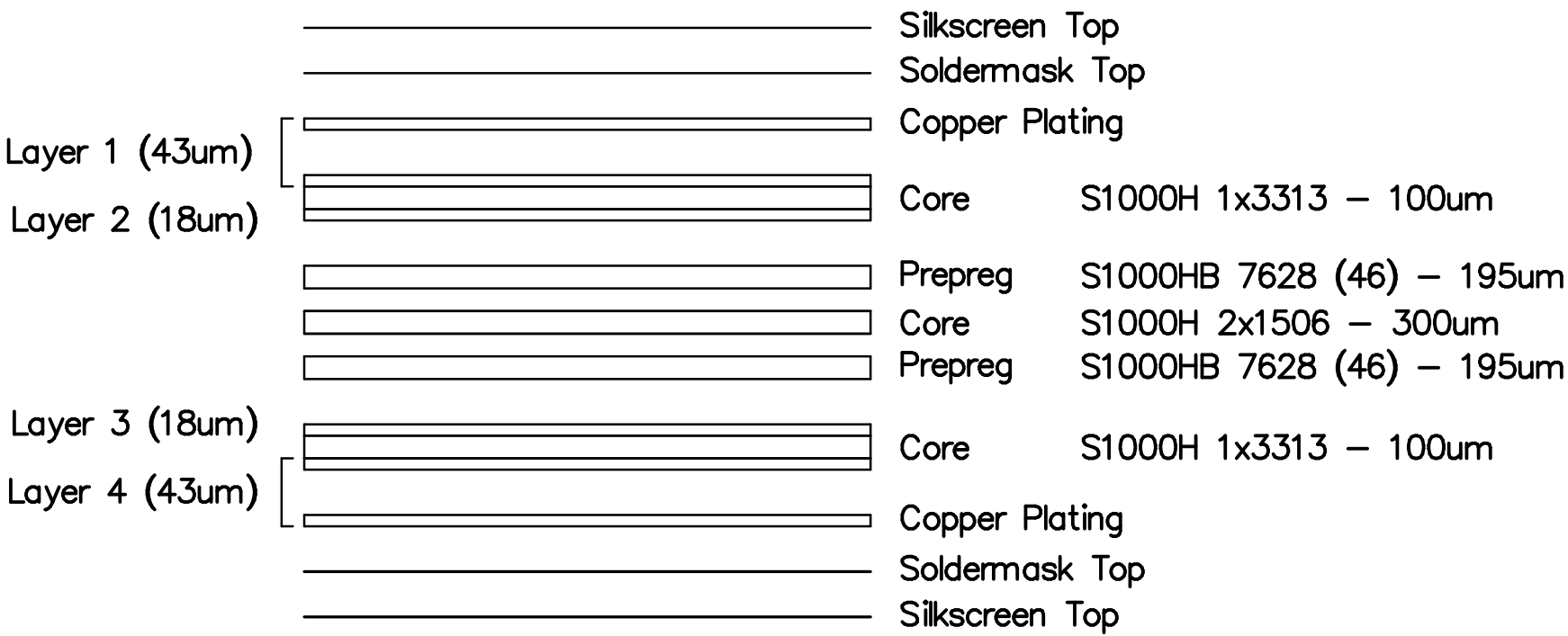
12. Min trace width/clearance – 0.1mm/0.1mm

13. Impedance control – Yes, see details

14. Impedance tolerance – 10%

15. Board dimensions – 50.95mm x 30mm

Stack-Up



Impedance Requirement Table

Type	Layer	Ref. Layer		Trace Width	Trace Separation	Ground Spacing	Impedance
		Upper	Lower				
Differential	1	—	2	0.150	—	0.300	50 Ohm
Differential	4	—	3	0.135	0.155	—	90 Ohm
Differential	1	—	2	0.110	0.170	—	100 Ohm

Symbol	Hole Size	Plated	Drill Layer Pair	Count
⊕	0.200mm	PTH	01_Top – 04_Bottom	585
□	1.100mm	NPTH	01_Top – 04_Bottom	1
○	1.600mm	NPTH	01_Top – 04_Bottom	1
▽	2.600mm	PTH	01_Top – 04_Bottom	3
✕	3.200mm	NPTH	01_Top – 04_Bottom	2