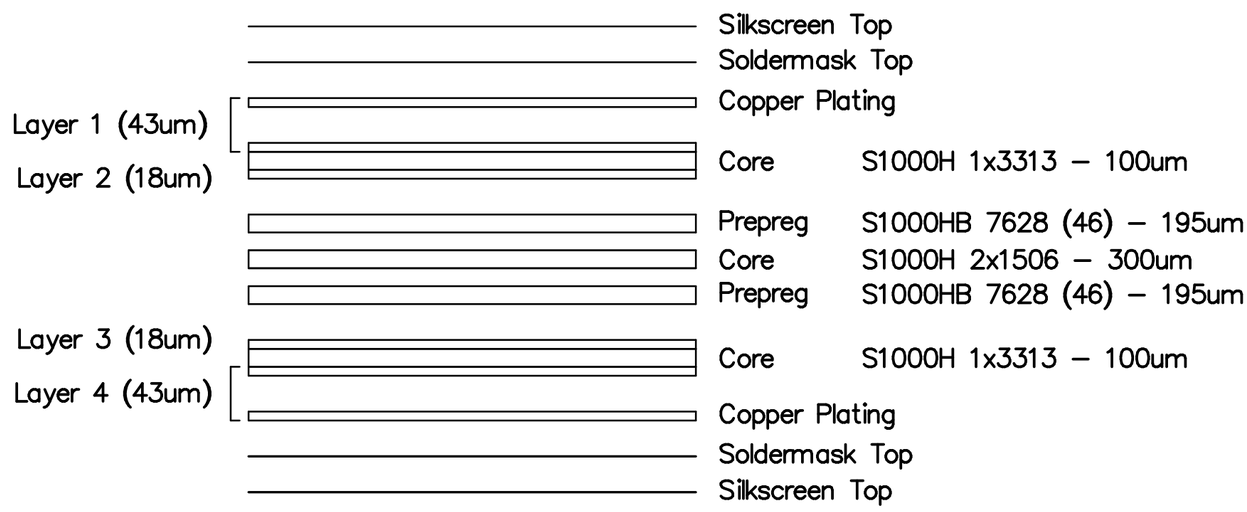
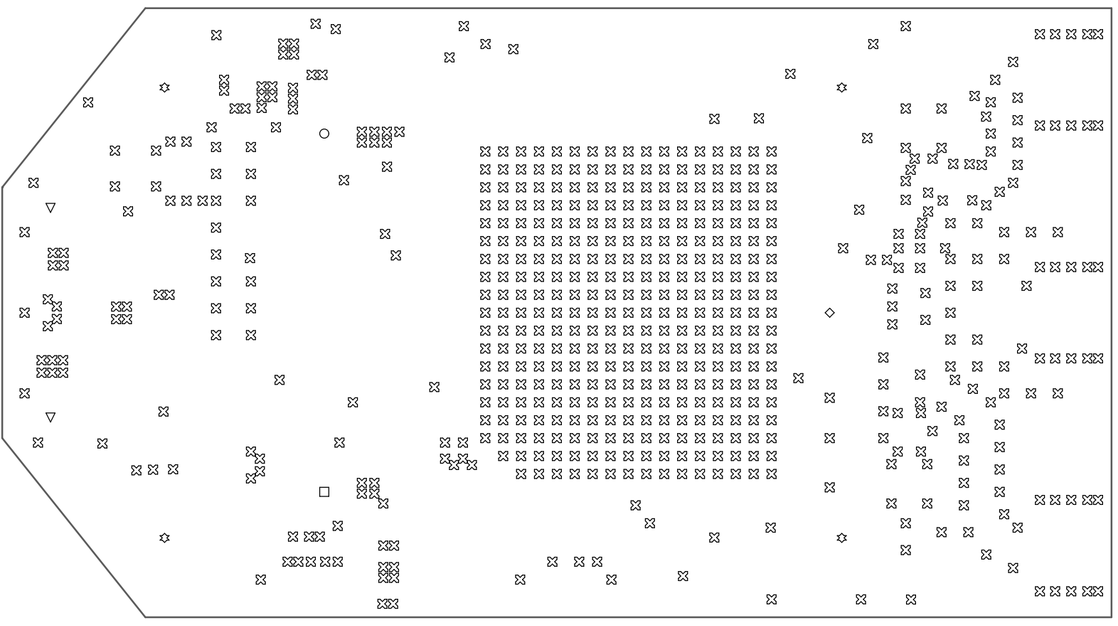
1. Used material – S1000H
2. Number of layers – 4
3. Board thickness – 1.0mm +/–10%
4. Thickness of outer layer (Cu) – 43um include copper plating
5. Thickness of inner layers (Cu) – 18um
6. Solder Mask – Two sides, Green
7. Silkscreen – One side, White
8. Type of coating – ENIG
9. Electrical Test on both side (use Adapter_USB3_to_M2_E.ipc file)
10. Min dia of plated hole – 0.2mm
11. Min trace width/clearance – 0.1mm/0.1mm
12. Impedance control – Yes, see details
13. Impedance tolerance – 10%
14. Board dimensions – 60mm x 34mm

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	608
▽	0.600mm	PTH	01_Top – 04_Bottom	2
□	1.100mm	NPTH	01_Top – 04_Bottom	1
○	1.600mm	NPTH	01_Top – 04_Bottom	1
◇	2.600mm	PTH	01_Top – 04_Bottom	1
✡	3.200mm	NPTH	01_Top – 04_Bottom	4

Slot definitions : Routed Path Length = Calculated from tool start centre position to tool end centre position.
Hole Length = Routed Path Length + Tool Size = Slot length as defined in the PCB layout