- 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

Stack-Up

Layer 1 (43um)

Layer 2 (18um)

Layer 3 (18um)

Layer 4 (43um)

Silkscreen Top Soldermask Top

Copper Plating

Copper Plating

Soldermask Top Silkscreen Top

S1000H 1x3313 - 100um

S1000H 2x1506 - 300um

S1000H 1x3313 - 100um

S1000HB 7628 (46) - 195um

S1000HB 7628 (46) - 195um

Core

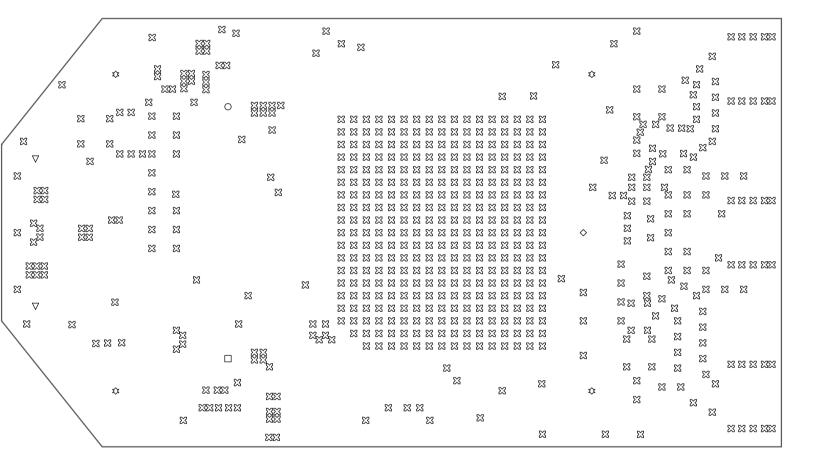
Core

Core

Prepreg

Prepreg

- 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



Impedance Requirement Table

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

Symbol	Hole Size	Plated	Drill Layer Pair	Count
\boxtimes	0.200mm	PTH	01_Top - 04_Bottom	608
∇	0.600mm	PTH	01_Top - 04_Bottom	2
	1.100mm	NPTH	01_Top - 04_Bottom	1
0	1.600mm	NPTH	01_Top - 04_Bottom	1
\Diamond	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