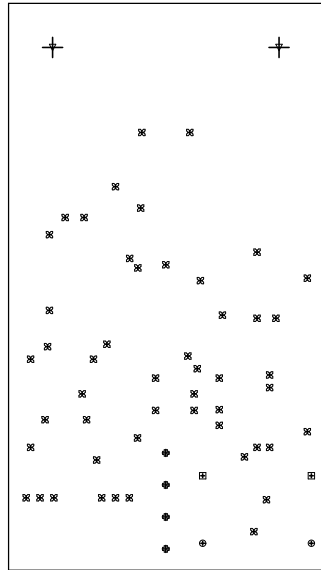
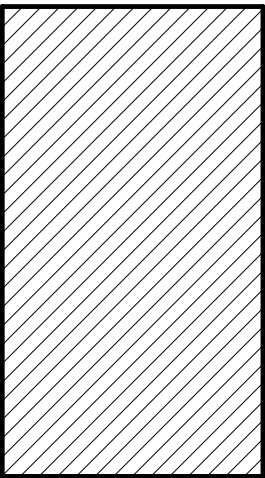


Layer	Name	Material	Thickness	Constant	Gerber	Board Layer Stack
	Top Overlay				GTO	
	Top Solder	Solder Resist	0.010mm	3.5	GTS	
1	Top Layer		0.035mm		GTL	
	Dielectric1	FR-4 (7628)	0.180mm	4.2		
	Dielectric 2	FR-4 (7628)	0.180mm	4.2		
2	MidLayer1		0.035mm		G1	
	Core		0.710mm	4.2		
3	MidLayer2		0.035mm		G2	
	Dielectric 3	FR-4 (7628)	0.180mm	4.2		
	Dielectric 4	FR-4 (7628)	0.180mm	4.2		
4	Bottom Layer		0.035mm		GBL	
	Bottom Solder	Solder Resist	0.010mm	3.5	GBS	
	Bottom Overlay				GBO	

Total board thickness: 1.590mm



Symbol	Count	Hole Size	Hole Length	Plated	Hole Type	Drill Layer Pair	Via/Pad	Pad Shape
⊠	50	0.300mm (11.81mil)	-	PTH	Round	Top Layer - Bottom Layer	Via	Rounded
□	2	0.600mm (23.62mil)	1.600mm (62.99mil)	PTH	Slot	Top Layer - Bottom Layer	Pad	Rounded
○	2	0.600mm (23.62mil)	2.100mm (82.68mil)	PTH	Slot	Top Layer - Bottom Layer	Pad	Rounded
⊕	4	1.100mm (43.31mil)	-	PTH	Round	Top Layer - Bottom Layer	Pad	(Mixed)
▽	2	3.200mm (125.98mil)	-	NPTH	Round	Top Layer - Bottom Layer	Pad	Rounded
	60 Total							

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

