



























### Layer Stack Up Detail for: Holiday r1.PcbDoc

Layer Name	Gerber Document	Dielectric Material
Top Solder Mask	(.GTS)	Solder Resist
Top Layer	(.GTL)	
Mid-Layer 1	(.G1)	FR-4
InternalPlane1	(.GPI)	FR-4
Bottom Layer	(.GBL)	FR-4
Bottom Solder Mask	(.GBS)	Solder Resist

Symbol	Hit Count	Tool Size	Plated	Hole Type
□	367	0.3mm (11.811mil)	PTH	Round
○	42	0.5mm (19.685mil)	PTH	Round
×	32	0.9mm (35.433mil)	PTH	Round
◇	3	1mm (39.37mil)	PTH	Round
⊕	2	1.4mm (55.118mil)	NPTH	Round
▽	1	1.6mm (62.992mil)	NPTH	Round
☆	1	1.8mm (70.866mil)	NPTH	Round
⊗	2	2.3mm (90.551mil)	PTH	Round
■	4	5.6mm (220.472mil)	PTH	Round
454 Total				

BOARD SIZE:	67.00mm x 67.00mm
LAYERS:	4
MATERIAL:	FR4 1.6mm
COPPER:	35um (after plating)
PTH PLATING:	20um (minimum)
FINISH:	ROHS ENIG or HAL ?
OVERLAY:	White, 2 sides
SOLDER MASK:	Green LPI, 2 sides
PROFILING:	V-Groove

FILE NAME:	Holiday r1.PcbDoc
LAYER:	Drill Drawing
PRINT DATE:	7/05/2013