









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
○	373	11.811mil (0.3mm)	PTH	Round
□	42	19.685mil (0.5mm)	PTH	Round
✕	32	35.433mil (0.9mm)	PTH	Round
◇	3	39.37mil (1mm)	PTH	Round
⊕	2	55.118mil (1.4mm)	NPTH	Round
▽	1	62.992mil (1.6mm)	NPTH	Round
☆	1	70.866mil (1.8mm)	NPTH	Round
⊗	2	90.551mil (2.3mm)	PTH	Round
■	4	220.472mil (5.6mm)	PTH	Round
460 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 r2.PcbDoc
LAYER:	Drill Drawing
PRINT DATE:	11/06/2013