























Layer Stack Up Detail for: Holiday r1.PcbDoc

Layer Name	Gerber Document	Dielectric Material
Top Solder Mask	<.GTS>	Solder Resist
Top Layer	<.GTL>	FR-4
Mid-Layer 1	<.G1>	FR-4
InternalPlane1	<.GP1>	FR-4
Bottom Layer	<.GBL>	FR-4
Bottom Solder Mask	<.GBS>	Solder Resist

Symbol	Hit Count	Tool Size	Plated	Hole Type
○	374	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
▽	1	220.472mil (5.6mm)	NPTH	Round
○	3	220.472mil (5.6mm)	PTH	Round
461 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 r3.PcbDoc
LAYER:	Drill Drawing
PRINT DATE:	14/02/2014