BOARD BUILD SPECIFICATION FOR H13174v1

BMG Contact:		**Tel:**	
Design Contact:		**Tel:**	
Delivery Address:			

PCB Size (mm): X x Y	25.5 x 18	**Thickness (mm): +/- 10%**	1
Panel Size (mm): X x Y	116.5 x 94	**Board Material:**	FR4
PCBs per Panel:	9	**Number of Layers:**	4
Panelisation Details:	Refer to Electronic Data (Mechanical Layer 7)		

Build Sequence from Top to Bottom

Description	ODB++ Layer	Gerber Extension	Details	Thickness (µm)
Top Silkscreen	topoverlay	.GTO	WHITE	n/a
Top Solder Mask	topsolder	.GTS	BLUE	n/a
Top Layer	top	.GTL		18u + plating
PrePreg 3 x 1080				210u
Mid Layer 1	mid1	.G1		18u finished
FR4 Core				To suit thickness
Mid Layer 2	Mid 2	.G2		18u finished
PrePreg 3 x 1080				210u
Bottom Layer	bottom	.GBL		18u + plating
Bottom Solder Mask	bottomsolder	.GBS	BLUE	n/a

Drill Information (per PCB)

Layer	ODB++ Layer	Excellon Extension	Number of sizes	Count
Top to Bottom	drill	.TXT	4	150
		Total	4	150
Type of via holes:	Through – 0.3mm pad, 0.15mm hole			

Design Rules

	mm	Comments
Minimum Track Width	0.1	
Minimum Track Gap	0.1	
Minimum BGA Pad Diameter	n/a	
Minimum Hole Diameter (finished)	0.15	
Minimum Annular Ring	0.075	
Minimum Clearance on Solder Resist	0.05	
Minimum Line width on Silkscreen	0.15	

Finish

Immersion Tin	Immersion Silver	Immersion Ni Au	Hard Gold	Other
		Y		

Specific Requirements

	Y/N	Comments
U.L Approval	Y	In Solder resist away from tracks and pads
Date / Manufacturers code	Y	In Solder resist away from tracks and pads
Half-plated holes	Y	
Plated slots	N	
Flat Pads (Filled uVias in CSP/BGA Pad)	N	
Edge chamfer	N	
Peelable mask	N	
Impedance test required	N	

Additional Information

Boards to be bare board tested against netlist.
If ODB data is available, please use in preference to Gerber data.
Return ‘step and repeat’ Gerber paste mask files ASAP on receipt of order (Include panel outline).
Manufacture In Accordance with IPC-A-600 / IPC-6012 Class 2.
PCB’s to be free of Burrs.
Data must not be modified other than to allow for manufacturing process tolerances.
Copper Balancing can be added to panel frame if required (Do not copper balance the PCB).
If design has 250um diameter BGA pads, ensure these are a minimum of 240um diameter after processing.

Nets GND and ANTENNA shorted together at component ANT1

390um Traces on Layer 1 with reference to Plane on Layer 2 to be 50R +/-10%