Symbol	Count	Hole Size	Plated	Hole Type	Drill Layer Pair	Via/Pad	Pad Shape	Template
0	1	0.406mm (16.00mil)	PTH	Round	Top Layer - Bottom Layer	Via	Rounded	v81h41m0mx0
	2	2.400mm (94.49mil)	NPTH	Round	Top Layer - Bottom Layer	Pad	Rounded	c10hn240
$\nabla$	2	3.000mm (118.11mil)	NPTH	Round	Top Layer - Bottom Layer	Pad	Rounded	c10hn300
O	6	0.860mm (33.86mil)	PTH	Round	Top Layer - Bottom Layer	Pad	(Mixed)	(Mixed)
Ħ	51	0.305mm (12.00mil)	PTH	Round	Top Layer - Bottom Layer	Via	Rounded	v61h30m0mx0
❖	60	0.254mm (10.00mil)	PTH	Round	Top Layer - Bottom Layer	Via	Rounded	v51h25m0mx0
	122 Total							



## ABRICATION NOTES:

- 1. LATEST VERSION OF ALL REFERENCED IPC SPECIFICATIONS TO BE USED.
- 2. ALL MULTILAYER PC BOARDS MUST BE PRODUCED IN ACCORDANCE IPC-A-600, CLASS 2.
- 3. INSPECT ALL MULTILAYER PC BOARDS IN ACCORDANCE WITH IPC-6012, CLASS 2.
- 4. MATERIAL: THIN LAMINATE, FR4 COPPER CLAD 1 OZ PER LAYER (EXTERNAL LAYERS) 1.6 MM +/-10% THICK, COLOR GREEN,
- MINIMUM CTI (PER IEC 60112 USING SOLUTION A) = 175.
- 5. FABRICATE BOARDS USING THE GERBER FILES CONTAINED IN FABRICATION FOLDER.
- 6. DRILL DATA ARE CONTAINED IN THE FABRICATION FOLDER.
- 7. ALL HOLE DIAMETERS MUST BE PER DRILL CHART +/-0.003 INCHES TOLERANCE, UNLESS OTHERWISE SPECIFIED.
- 8. PLATED-THRU HOLES TO HAVE COPPER WALL THICKNESS NOT LESS THAN 0.001 INCHES.
- THICKNESS TO BE DETERMINED BY IPC-6012, CLASS 2.
- 9. SURFACE FINISH PROCESS MUST BE ENIG.
- 10. SILKSCREEN BOTH SIDES WITH WHITE EPOXY NON-CONDUCTIVE INK. SILKSCREEN MAY BE TRIMMED OFF ANY SOLDERED ENTITY.
- 11. SOLDER MASK BOTH SIDES WITH LIQUID PHOTO IMAGEABLE SOLDER MASK (LPI), MEETING IPC-SM-840, CLASS 2, HIGH PERFORMANCE SPECIFICATIONS. FINISH: SOLDER MASK OVER BARE COPPER (SMOBC).
- 12. WARP AND TWIST SHALL BE LESS THAN 0.0075 INCH PER INCH. INSPECT PER IPC-TM-650, 2.4.22.
- 13. ALL BOARDS MUST BE DATE CODED USING A FOUR DIGIT CODE, YEAR FOLLOWED BY WEEK (e.g. 0449 = 49TH WEEK OF 2004).
- 14. ALL DIMENSIONS AND HOLE DIAMETERS APPLY TO THE FINISHED BOARD.
- 15. PADS MUST BE FINISHED TO WITHIN +/-0.0015 INCHES OF THE MINOR DIMENSION (PAD WIDTH)
- AND +/-0.002 INCHES OF THE MAJOR DIMENSION (PAD LENGTH).
- 16. FIDUCIALS MUST BE FREE OF ANY MARKINGS.
- 17. BARE BOARD TO BE ELECTRICALLY TESTED IN ACCORDANCE WITH IPC-ET-652 AND MARKED AS TESTED.
- 18. BOARDS SHALL BE UL-RECOGNIZED AND UL-MARKED. UL FIRE RETARDANT RATING 94V-0 OR HIGHER.
- 19. ROHS: ALL MATERIALS MUST BE COMPLIANT WITH THE ROHS DIRECTIVE.
- 20. EACH PCB MUST BE INDIVIDUALLY BAGGED.

Layer	Name	Material	Thickness	Constant	Board Layer Stack
1	Top Overlay				
2	Top Solder	Solder Resist	0.40mil	3.5	
3	Top Layer	Copper	1.40mil		
4	Dielectric 1	FR-4	59.39mil	4.2	
5	Bottom Layer	Copper	1.40mil		
6	Bottom solder	Solder Resist	0.40mil	3.5	
7	Bottom Overlay				

Total Pcb Thickness = 1.6MM +/- 10%

Finished cu Thickness: 1 0Z

