Rev	Revision note	Date	Sign	Check

[TABLE 1] MANUFACTURING INFORMATION FOR PCB

ElCAD File No. WILCOX.BRD	PCB P/N: M-P-00073-0001 REV-1.0
CUSTOMER: MARIS	

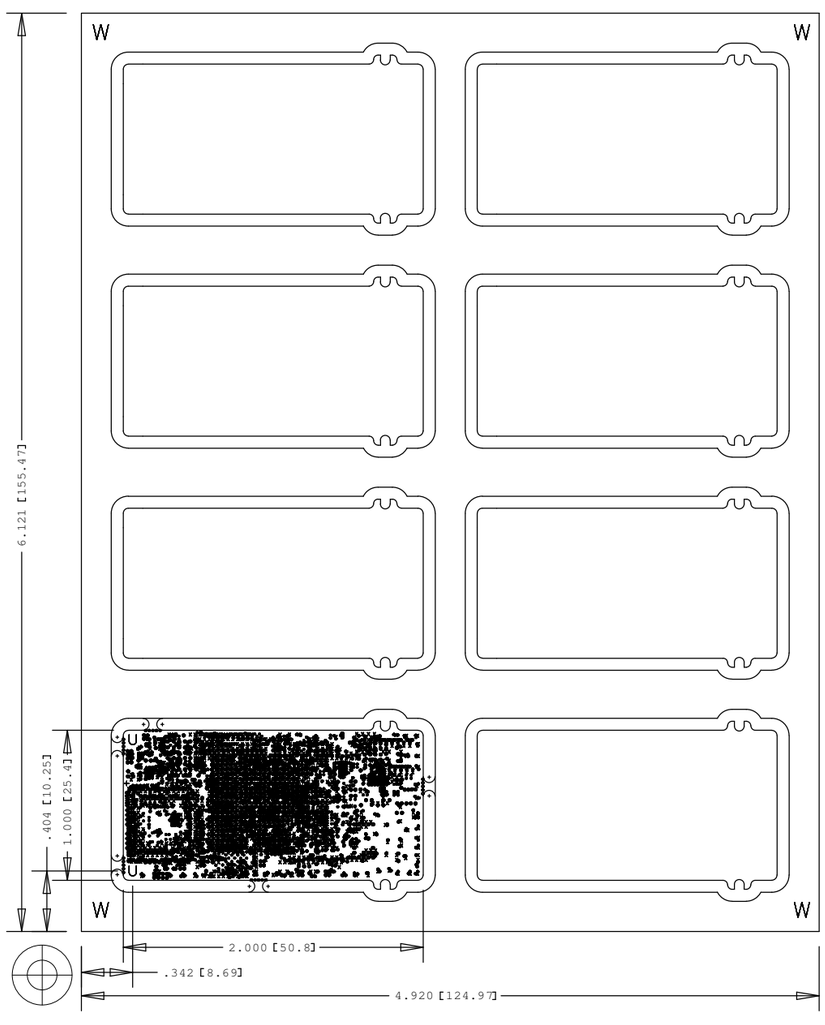
LAYER STRUCTURE (from TOP to BOTTOM)

LYR	DESC.	COPPER [Oz]	DIEL. [Mil]	GERBER FILE	IMPEDANCE			
					SE 50 Ohm	SE 75 Ohm	Diff. 90 Ohm	Diff. 100 Ohm
1.	TOP	1.0[1]	3.0	WILCOX_TOP.LGX	4.5 mil	4.4 mil[2]		
2.	L2-GND	1.0[1]	3.0	WILCOX_L2-GND.LGX				
3.	L3-SIG	1.0[1]	4.0	WILCOX_L3-SIG.LGX	3.3 mil			
4.	L4-SIG	0.5	4.0	WILCOX_L4-SIG.LGX	4.0 mil			
5.	L5-GND	0.5	4.0	WILCOX_L5-GND.LGX				
6.	L6-SIG	0.5	4.0	WILCOX_L6-SIG.LGX	3.2 mil		4/5/4 mil	3.4/6.0/3.4 mil
7.	L7-GND	0.5	4.0	WILCOX_L7-GND.LGX				
8.	L8-SIG	0.5	4.0	WILCOX_L8-SIG.LGX	4.0 mil			
9.	L9-SIG	0.5	4.0	WILCOX_L9-SIG.LGX	4.0 mil			
10.	L10-GND	0.5	4.0	WILCOX_L10-GND.LGX				
11.	L11-SIG	0.5	4.0	WILCOX_L11-SIG.LGX	3.2 mil			3.4/6.0/3.4 mil
12.	L12-GND	0.5	4.0	WILCOX_L12-GND.LGX				
13.	L13-SIG	0.5	4.0	WILCOX_L13-SIG.LGX	4.0 mil			
14.	L14-SIG	1.0[1]	3.0	WILCOX_L14-SIG.LGX	3.3 mil			
15.	L15-GND	1.0[1]	3.0	WILCOX_L15-GND.LGX				
16.	BOTTOM	1.0[1]		WILCOX_-BOTTOM.LGX	4.5 mil	4.4 mil[3]		

NOTES: [1] - COPPER THICKNESS AFTER PLATING

NOTES: [2] -REFERENCE TO LAYER 3

NOTES: [3] -REFERENCE TO LAYER 14



MicroVia structure

1.VIA BETWEEN LAYERS 1-2
2.VIA BETWEEN LAYERS 2-3
3.STACK VIA MADE OF 1-2 and 2-3
4.VIA BETWEEN LAYERS 3-14
5.STACK VIA MADE OF 14-15 and 15-16
6.VIA BETWEEN LAYERS 14-15
7.VIA BETWEEN LAYERS 15-16

DRILL CHART: TOP to L2-GND

ALL UNITS ARE IN MILS

FIGURE	SIZE	TOLERANCE	PLATED	QTY
•	4.0	+1.0/-3.0	PLATED	1517

TOTAL HOLES: 1517

DRILL CHART: L2-GND to L3-SIG

ALL UNITS ARE IN MILS

FIGURE	SIZE	TOLERANCE	PLATED	QTY
•	4.0	+1.0/-3.0	PLATED	998

TOTAL HOLES: 998

DRILL CHART: L3-SIG to L14-SIG

ALL UNITS ARE IN MILS

FIGURE	SIZE	TOLERANCE	PLATED	QTY
×	8.0	+2.0/-2.0	PLATED	1135

TOTAL HOLES: 1135

DRILL CHART: L14-SIG to L15-GND

ALL UNITS ARE IN MILS

FIGURE	SIZE	TOLERANCE	PLATED	QTY
•	4.0	+1.0/-3.0	PLATED	830

TOTAL HOLES: 830

DRILL CHART: L15-GND to BOTTOM

ALL UNITS ARE IN MILS

FIGURE	SIZE	TOLERANCE	PLATED	QTY
•	4.0	+1.0/-3.0	PLATED	1052
•	4.0	+1.0/-3.0	PLATED	3

TOTAL HOLES: 1055

DRILL CHART: TOP to BOTTOM

ALL UNITS ARE IN MILS

FIGURE	SIZE	TOLERANCE	PLATED	QTY
+	10.0	+2.0/-0.0	PLATED	1
∘	16.0	+4.0/-0.0	NON-PLATED	25
U	65.0	+4.0/-0.0	NON-PLATED	2
W	98.0	+4.0/-4.0	NON-PLATED	4

TOTAL HOLES: 32

[TABLE 2] NOTES: UNLESS OTHERWISE SPECIFIED

APPLICABLE STANDARDS/SPECIFICATIONS: ASME Y14.5M-1994, DIMENSIONS AND TOLERANCES.
PRINTED CIRCUIT BOARD (PCB) DESCRIPTION :
RIGID HDI 16 LAYERS BOARD.
PCB NAME: WILCOX P/N M-P-00073-0001 REV-1.0
FINAL BOARD THICKNESS 1.8mm +/-10%.

1. INCH DIMENSIONS ARE THE CONTROLLING DIMENSIONS FOR THE DRAWINGS AND SUPPLIED DATA. MILLIMETERS DIMENSIONS ARE FOR REFERENCE ONLY.
2. FABRICATE PCB IN ACCORDANCE WITH IPC-6012, CLASS 2; PER IPC-6011 USING CUSTOMER SUPPLIED DATA FILES. WHENEVER IS APPROPRIATE WILL BE INSPECTED WITH REFERENCE TO PUBLICATION IPC-A-600-E CLASS 2.
3. MATERIALS:(UNLESS OTHERWISE SPECIFIED)
   A. LAMINATE SHALL BE PER IPC-2221 PAR 4.3; IN ACCORDANCE WITH IPC-4101/24, LAMINATED SHEET, FLAME RESISTANT (MEETING UL 94V-0). Tg RATING: 170 DEG MIN.
   B. PREPREG SHALL BE PER IPC-2221 PAR 4.2; IN ACCORDANCE WITH IPC-4101/24, PREPREG (B-STAGE), FLAME RESISTANT (MEETING UL 94V-0). Tg RATING: 170 DEG MIN.
   C. REFER TO STACKUP mLDVC-STACKUP-14-AUG-14.XLS FILE FOR : FOR COPPER FOIL WEIGHTS AND DIALECTRIC THICKNESS.
   D. LASER VIAS 1-2, 15-16 AND PTH VIAS SHALL BE 100% FILLED AND CAPPED FROM BOTH SIDES. BURRIED VIAS SHALL BE 100% FILLED WITHOUT CAPPING. FILLING MATERIAL SHALL BE WITH SPECIALLY FORMULATED HOLE PLUGGING MATERIAL. PREFFERED MATERIAL TAIYO THP-100DX1 OR PETERS. FILLING WITH SOLDERMASK OR RESIN IS NOT ALLOWED.
4. ALL HOLES SHALL BE LOCATED WITHIN +/- 0.05 (.002") (RADIAL ERROR) OF TRUE POSITION. LAYER-TO-LAYER REGISTRATION SHALL BE WITHIN +/- 0.05 (.002"). ADD TEARDROPS AS REQUIRED SO THAT ALL HOLES SURROUNDED BY LAND HAVE A MINIMUM ANNULAR RING OF 0.05 (.002").
5. FINISH:
   A. ALL EXPOSED CONDUCTIVE PATTERN AREAS NOT COVERED WITH SOLDERMASK OR OTHER PLATING SHALL BE PLATED WITH IMMERSION GOLD 0.08 - 0.23 Micron OVER ELECTROLESS NICKEL 2.5 - 5 Micron (NiAu) OVER COPPER.
   B. APPLY LIQUID PHOTO IMAGEABLE SOLDER MASK PER IPC-SM-840, CLASS H, TO BOTH SIDES OF THE BOARD OVER BARE COPPER.
   C. VIA PLUG : N/A.
   D. SILKSCREEN SHALL BE WHITE, PERMANENT, ORGANIC, NON-CONDUCTIVE INK. THERE SHALL BE NO SILKSCREEN ON ANY SOLDERABLE COMPONENT PAD.
6. MARKING:
   A. UL LOGO, MANUFACTURER'S IDENTIFICATION AND DATE CODE LETTER SHALL BE ADDED ON TOP OR BOTTOM SILK.
7. TEST REQUIREMENTS:
   A. 100 % NETLIST ELECTRICAL VERIFICATION USING CUSTOMER SUPPLIED IPC-D-356A NETLIST FOR OPENS AND SHORTS. ALL NETS SHALL BE ACCESSED SIMULTANEOUSLY OR AS OTHERWISE MUTUALLY AGREED UPON.
8. TOLERANCES:(UNLESS OTHERWISE SPECIFIED)
   A. WARP OR TWIST OF BOARD SHALL NOT EXCEED 0.75 % PER IPC-TM-650, METHOD 2.4.22.
   B. CONDUCTOR WIDTHS AND SPACING SHALL BE WITHIN +/- 0.0125 (.0005") OF GERBER DATA.
   C. REMOVE ALL BURRS AND BREAK SHARP EDGES 0.4 (.015") MAXIMUM.
   D. SURFACE MOUNT PAD PLATING MUST BE FLAT TO A MAXIMUM OF 0.08 (.003") ABOVE BOARD SURFACE.
   E. 0.8 (0.032") MAXIMUM RADIUS ON ANY INSIDE CORNERS.
   F. HOLE SIZE TOLERANCE SHALL BE : +0.05/-0.05. (UNLESS OTHERWISE SPECIFIED.
   G. GENERAL TOLERANCE: +/- 0.1.
   H. ANGLE TOLERANCE SHALL BE +/- 2 DEGREES.
   I. MINIMUM WALL THICKNESS OF 0.001 INCH TO ALL COPPER PLATED HOLES.
9. CONTROLLED IMPEDANCE: AS PER TABLE 1
10. THEIVING:
    A. SUPPLIER SHALL NOT ADD THIEVING TO COMPENSATE FOR HIGH COPPER DENSITY AREAS ON THIS DESIGN PRIOR TO CUSTOMER APRROVAL.
11. PLATE EDGE CONNECTOR : N/A
12. GENERAL NOTES:
    A. PACKAGING: EACH 10 BOARDS/PANELS SHOULD BE PACKGED SEPARATLY.
    B. DRAWING IS VIEWED FROM COMPONENT SIDE.
    C. GERBER FILES IN 274-X FORMAT.
    D. DRILL BOARD USING NC DRILL FILE(S) SUPPLIED (USING EXCELLON FORMAT).
    E. ALL PRODUCT LOT SHOULD BE DELIVERD WITH C.O.C.
    F. IN CASE OF PANEL DESIGN - PLEASE DUPLICATE PASTE FILES AND SEND TO CUSTOMER.
    G. IN CASE OF PANEL DESIGN - DO NOT SEPATATE BOARDS FROM PANEL.

Elcad PCB Ltd. TEL: 08-941-7274
Email: elcad @ zahav.net.il

LAYOUT DESIGNER ELCAD			
CUSTOMER:	Maris		
PROJECT	WILCOX M-P-00073-0001 Rev.1.0		
LAYER	FILE NAME :	WILCOX_FAB-DWG.LGX	
	DESCRIPTION	FAB-DWG.LGX	
DATE: 22-03-15	ORDER NO:	ELCAD NO: WILCOX	

PROPERTY OF Maris
AFTER USE RETURN TO Maris