

TABLE 2

NOTES: UNLESS OTHERWISE SPECIFIED

APPLICABLE STANDARDS/SPECIFICATIONS: ASME Y14.5M-1994, DIMENSIONS AND TOLERANCES.

PRINTED CIRCUIT BOARD (PCB) DESCRIPTION :

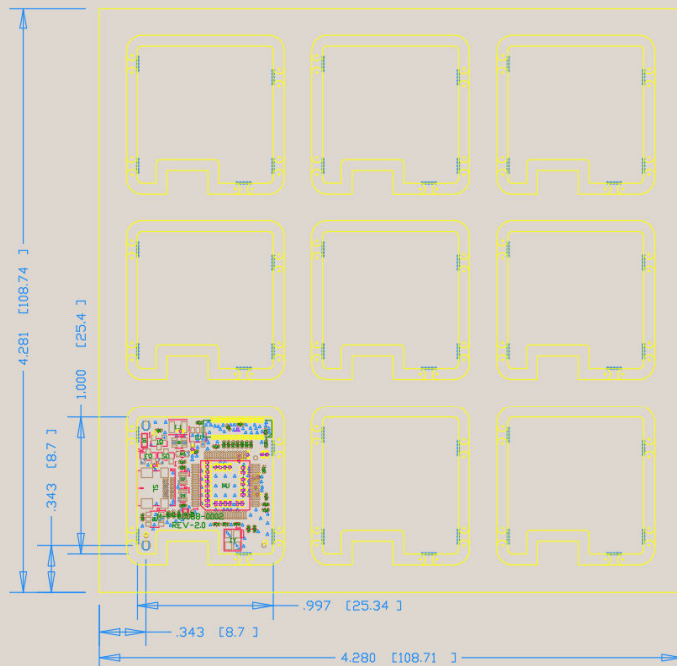
RIGID 6 LAYERS BOARD.

PCB NAME: NP-N\_MEZZ\_HDMI\_R2.BRD P/N M-P-00088-0002 REV-2.0

FINAL BOARD THICKNESS 1.2mm +/-10%.

- MILLIMETERS (mm) ARE THE CONTROLLING DIMENSIONS FOR THE DRAWINGS AND SUPPLIED DATA. INCH DIMENSIONS ARE FOR REFERENCE ONLY.
- 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.
- MATERIALS<UNLESS OTHERWISE SPECIFIED>
  - LAMINATE SHALL BE PER IPC-2221 PAR 4.3; IN ACCORDANCE WITH IPC-4101/24 , LAMINATED SHEET, FLAME RESISTANT (MEETING UL 94V-0). T<sub>g</sub> RATING: 170 DEG MIN.
  - PREPREG SHALL BE PER IPC-2221 PAR 4.2; IN ACCORDANCE WITH IPC-4101/24 , PREPREG (B-STAGE), FLAME RESISTANT (MEETING UL 94V-0). T<sub>g</sub> RATING: 170 DEG MIN.
  - REFER TO STACKUP\_LDVC\_REV4.XLSX FILE FOR COPPER FOIL WEIGHTS AND DIALECTRIC THICKNESS.
  - LASER VIAS 5-6 AND PTH VIAS SHALL BE 100% FILLED AND CAPPED FROM BOTH SIDES. BURIED VIAS SHALL BE 100% FILLED WITHOUT CAPPING. FILLING MATERIAL SHALL BE WITH SPECIALLY FORMULATED HOLE PLUGGING MATERIAL. PREFERRED MATERIAL TAIYO THP-100DX1 OR PETERS. FILLING WITH SOLDERMASK OR RESIN IS NOT ALLOWED.
- 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").
- FINISH:
  - 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.
  - APPLY LIQUID PHOTO IMAGEABLE SOLDER MASK PER IPC-SM-840, CLASS H, TO BOTH SIDES OF THE BOARD OVER BARE COPPER.
  - VIA PLUG : N/A.
  - SILKSCREEN SHALL BE WHITE, PERMANENT, ORGANIC, NON-CONDUCTIVE INK. THERE SHALL BE NO SILKSCREEN ON ANY SOLDERABLE COMPONENT PAD.
- MARKING:
  - UL LOGO, MANUFACTURER'S IDENTIFICATION AND DATE CODE LETTER SHALL BE ADDED ON TOP OR BOTTOM SILK.
- TEST REQUIREMENTS:
  - 100 % NETLIST ELECTRICAL VERIFICATION USING CUSTOMER SUPPLIED IPC-D-356A NETLIST FOR OPENS AND SHORTS. ALL NETS SHALL BE ACCESSED SIMULTANEDUSLY OR AS OTHERWISE MUTUALLY AGREED UPON.
- TOLERANCES<UNLESS OTHERWISE SPECIFIED>
  - WARP OR TWIST OF BOARD SHALL NOT EXCEED 0.75 % PER IPC-TM-650 , METHOD 2.4.22 .
  - CONDUCTOR WIDTHS AND SPACING SHALL BE WITHIN +/- 0.0125 (.0005") OF GERBER DATA.
  - REMOVE ALL BURRS AND BREAK SHARP EDGES 0.4 (.015") MAXIMUM.
  - SURFACE MOUNT PAD PLATING MUST BE FLAT TO A MAXIMUM OF 0.08 (.003") ABOVE BOARD SURFACE.
  - 0.8 (.032") MAXIMUM RADIUS ON ANY INSIDE CORNERS.
  - HOLE SIZE TOLERANCE SHALL BE : +0.05/-0.05. (UNLESS OTHERWISE SPECIFIED).
  - GENERAL TOLERANCE: +/- .01 .
  - ANGLE TOLERANCE SHALL BE +/- 2 DEGREES.
  - MINIMUM WALL THICKNESS OF 0.001 INCH TO ALL COPPER PLATED HOLES.
- CONTROLLED IMPEDANCE:
  - PLEASE SEE TABLE 1
- THEIVING:
  - SUPPLIER SHALL NOT ADD THIEVING TO COMPENSATE FOR HIGH COPPER DENSITY AREAS ON THIS DESIGN PRIOR TO CUSTOMER APPROVAL.
- PLATE EDGE CONNECTOR : N/A
- GENERAL NOTES:
  - PACKAGING: EACH 10 BOARDS/PANELS SHOULD BE PACKGED SEPARATLY.
  - DRAWING IS VIEWED FROM COMPONENT SIDE.
  - GERBER FILES IN 274-X FORMAT.
  - DRILL BOARD USING NC DRILL FILE(S) SUPPLIED (USING EXCELLON FORMAT).
  - ALL PRODUCT LOT SHOULD BE DELIVERED WITH C.O.C.
  - IN CASE OF PANEL DESIGN - PLEASE DUPLICATE PASTE FILES AND SEND TO CUSTOMER.
  - IN CASE OF PANEL DESIGN - DO NOT SEPARATE BOARDS FROM PANEL.

TABLE 1							MANUFACTURING INFORMATION FOR PCB		
EICAD File No. NP-N_MEZZ_HDMI_R2.BRD				PCB P/N: M-P-00088-0002 REV-2.0					
CUSTOMER: MARIS									
							LAYER STRUCTURE (from TOP to BOTTOM)		
L.YR	DESC.	COPPER [Oz]	DIEL. [mil]	GERBER FILE	IMPEDANCE				
					SE 50 Ohm	DIFF. 100 Ohm			
1.	TOP	0.333+Plating		XXXXX_TOP.LGX	5.0 mil				
2.	L2-PWR	1.0		XXXXX_L2-PWR.LGX					
3.	L3-SIG	0.5		XXXXX_L3-SIG.LGX	4.0 mil				
4.	L4-SIG	0.5		XXXXX_L4-SIG.LGX	4.0 mil				
5.	L5-GND	1.0		XXXXX_L5-GND.LGX		5.1/8/5.1 mil Ref to Layer 4			
6.	BOT	0.333+Plating		XXXXX_BOT.LGX	5.0 mil				



DRILL CHART: L5-GND to BOTTOM				
ALL UNITS ARE IN MILS				
FIGURE	SIZE	TOLERANCE	PLATED	QTY
*	4.0	+2.0/-4.0	PLATED	44
TOTAL HOLES:				44

DRILL CHART: TOP to BOTTOM				
ALL UNITS ARE IN MILS				
FIGURE	SIZE	TOLERANCE	PLATED	QTY
*	10.0	+2.0/-4.0	PLATED	112
*	16.0	+4.0/-0.0	NON-PLATED	225
U	65.0	+4.0/-0.0	NON-PLATED	2
TOTAL HOLES:				339

Elcad PCB Ltd. TEL: 08-941-7274		LAYOUT DESIGNER	
Email: elcad@zahavnet.il		ELCAD	
CUSTOMER:		Maris	
PROJECT		NEPTUN MEZZ HDMI M-P-00088-0002 Rev-2.0	
LAYER	NAME	NP-N_MEZZ_HDMI_R2_SILK.LGX	
	DESCRIPTION	DRILL-DWG	
DATE: 21-AUG-2016		ORDER NO:	ELCAD NO: NP-P_MEZZ_HDMI_R2.BRD

PROPERTY OF Maris  
AFTER USE RETURN TO Maris