ELCAD File No. NPn_HDSDI_R2.BRD	MANUFACTURING INFORMATION FOR PCB
CUSTOMER: MARIS	P/N: M-P-00079-0002 REV-2.0

LAYER STRUCTURE (from TOP to BOTTOM)

LYR	DESCRIPTION	COPPER [Oz]	DIELECTRIC [Mil]	LAYER SYM	GERBER FILE	IMPEDANCE
1.	COMPONENT SIDE	0.5+0.5		CS	xxx_CS.LGX	SE=5 MIL 50 Ohm
2.	POWER PLANE	1.0		PP	xxx_PP.LGX	
3.	INTERNAL 1	0.5		I1	xxx_I1.LGX	SE=5 MIL 50 Ohm
4.	INTERNAL 2	0.5		I2	xxx_I2.LGX	SE=5 MIL 50 Ohm
5.	GROUND PLANE	1.0		GP	xxx_GP.LGX	
6.	PRINT SIDE	0.5+0.5		PS	xxx_PS.LGX	SE=5 MIL 50 Ohm SE=10 MIL 75 Ohm Ref.to INT2

IMPORTANT: SPACING BETWEEN INT1 AND INT2 SHOULD BE MAXIMUM

SILKSCREEN AND SOLDERMASK

SILKSCREEN TOP= FILE: XXX_ST.LGX (WHITE)
SILKSCREEN BOT= FILE: XXX_SB.LGX (WHITE)
SOLDERMASK TOP= FILE: XXX_MT.LGX (GREEN)
SOLDERMASK BOT= FILE: XXX_MB.LGX (GREEN)

DRILL AND PCB OUTLINE

DRILL GRAPHIC FILE= FILE: xxx_MH.LGX
DRILL TAPE FILE= FILE: xxx-1-6.DRL
DRILL (N/C) TAPE ORIGIN = DATABASE 0,0
BOARD OUTLINE SUPPLIED IN FILE: xxx_BO.LGX
THE FILE SUPPLIES FOR MILLING PURPOSE ONLY

FILE DESCRIPTION IN xxx-M.TXT
GERBER FORMAT RS-274-X (M,N = 5,5)

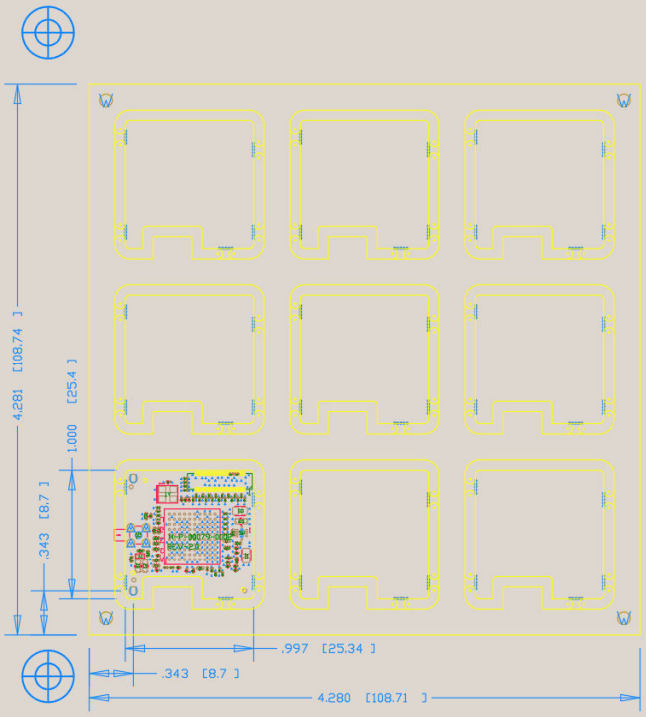
NOTES:
UNLESS OTHERWISE SPECIFIED:
1. TOTAL THICKNESS 1.2 mm +/- 10% AFTER PLATING
2. MATERIAL: FR4 Tg=170 DEG.C MINIMUM
3. FINISHING: IMMERSION GOLD
FINISH: IMMERSION GOLD 2-4 MICRO INCH THICKNESS,
OVER ELECTROLESS NICKEL 150-170 MICRO INCH THICKNESS
ON EXPOSED COPPER.
4. VIAS CONNECTED TO PLANE ARE DIRECT CONNECTION
5. HOLE DIAMETER IS AFTER PLATING
6. FINISHED HOLE SIZE TOLERANCE: +/-0.003"
7. VIA HOLE SIZE MAY BE MODIFIED BY MANUFACTURER
8. UNUSED VIA MAY REMOVED AS PRODUCTION NEEDS
9. PCB MFR SHOULD CLEAN SILKSCREEN BY SOLDER MASK
10. IN CASE OF CONFLICT BETWEEN MECHANICAL DIMENSIONS AND BOARD OUTLINE, GERBER FILE xxxxBO.LGX IS GOVERN
11. THE MANUFACTURER MAY WILL ADD TEAR DROPS IN ALL SIGNAL LAYERS IF NEED.
12. THE MANUFACTURER WILL ADD MANUFACTURER CODE, (FSCM) DATE AND UL-CODE IN P.S.ONLY.
13. ALL DIMENSIONS ARE IN INCH (mm)
14. FINISH BOARD SHALL MEET THE PERFORMANCE REQUIREMENTS OF IPC-6012 CLASS 2.
15. REMOVE ALL BURRS AND BREAK SHARP EDGES .015 INCH MAXIMUM.
16. BOARD BOW, WARP OR TWIST SHALL NOT EXCEED +/- 0.0075 PER INCH MEASURED IN ACCORDANCE WITH IPC-TM-650, METHOD 2.4.22.
17. CONTROLLED IMPEDANCE +/- 10% ☒ YES ☐ NO
18. PERFORM VCUT AS PER VIEW:A ☐ YES ☒ NO
19. DO NESTING (BREAKAWAY) AS PER DRAWING ☒ YES ☐ NO
20. RoHS COMPLIANCE:
20.1 RoHS COMPLIANT MARK/STAMP ON EACH PCB FRAME
20.2 RoHS COMPLIANT ON EACH PCB SEALED BAG/PACKAGE
20.3 IN ALL PACKAGING LEVELS AND SHIPPING DOCUMENTS IT SHOULD BE MARKED WITH A VISIBLE RoHS COMPLIANT MARK OR LABEL
20.4 C.O.C.SHOULD INCLUDE A DECLARATION THAT THE PCB'S ARE RoHS COMPLIANT

ELECTRICAL TEST

ELECTRICAL TEST FILE: XXXXX.IPC (IPC356D FORMAT)
IF FILE XXXXX.IPC IS MISSING PLEASE DO NOT PRODUCE THIS BOARD AND CALL ELCAD

GERBER FILES NAME CONVENTION: XXXXXYY.LGX
YY: LAYER CODE
XXXXX: BOARD NAME

Artwork by ELCAD PCB Ltd Tel: Office: 972-8-941-7274, 972-8-941-7284
CP: 972-522-560-866, e-mail: elcad@zahav.net.il



LAYER STACK-UP AND SCREEN RETURN SIDE)
NPn_HDSDI_R2.BRD 22-AUG-2016

DRILL CHART: TOP to BOTTOM
ALL UNITS ARE IN MILS

FIGURE	SIZE	TOLERANCE	PLATED	QTY
▪	8.0	+2.0/-4.0	PLATED	3
·	9.8	+2.0/-4.0	PLATED	200
⊘	20.0	+2.0/-2.0	PLATED	1
△	33.0	+2.0/-2.0	PLATED	4
▪	16.0	+4.0/-0.0	NON-PLATED	225
∪	65.0	+4.0/-0.0	NON-PLATED	2
W	98.0	+4.0/-4.0	NON-PLATED	4

TOTAL HOLES: 439