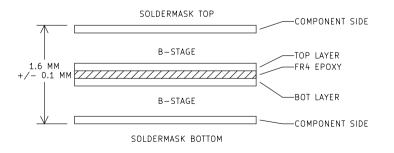
70.000 mm → 5.000 mm 5.000 mm k П П 5.000 8 3 Ē ĸ, 0 0 0 0 0 0 0 0 Ē 5.000 mm 5.000 mm

Drill Map:

- · 0.50mm / 0.020" (25 holes) · 0.81mm / 0.032" (8 holes) · 0.90mm / 0.035" (2 holes) (not plated) □ 3.00mm / 0.118" (4 holes) (not plated)

LAYER DETAIL



GENERAL NOTES

- 1. FABRICATE PER ANSI/IPC-1-600, IPC-0F-605, IPC-4101, IPC-4552 AND IPC-SM-840 SPECIFICATIONS. MUST USE ROHS-COMPLIANT MATERIALS.
- 2. MATERIAL: BASE MATERIAL LAMINATED EPOXY GLASS FR-4, NOM .062", COLOUR NATURAL. NOM 1 OZ COPER WEIGHT, THICKNESS .070" MAX AFTER PLATING AND FINISHING, PREFER ROHS COMPLIANT PER IPC-4101 SLASH SHEETS #26 OR #83 OR #98 WITH MINIMUM TG 135 DEGREE C OR HIGHER. TD 300 DEGREE C OR HIGHER AND FLAME RATED UL 94V-0.
- 3. SOLDERMASK BOTH SIDES OF BOARD OVER BARE COPPER WITH MATERIAL PER ANSI/IPC-SM-840, COLOR SHALL BE GREEN.
- 4. APPLY SOLDER PLATING WITH HOT-AIR LEVELLING TO EXPOSE COPPER BOTH SIDES. LEAD-FREE SOLDER REQUIRED.
- 5. INDICATED AREA ON BOTTOM SIDE AVAILABLE FOR APPLICATION OF PCB MANUFACTURER LOGO. UL MARKING, AND DATE-CODE. USE OF OTHER LOCATION TO BE APPROVED BY PURCHASER.
- 6. APPLY SILKSCREEN TO TOP SIDE OF BOARD USING NON-CONDUCTIVE WHITE EPOXY INK.
- 7. SEE FABRICATION FILES WITH SAME TITLE OR DRAWING NUMBER, AND REVISION, AS THIS DRAWING,
- 8. DIMNESONAL TOLERANCES ARE: .XX=+/-.01; .XXX=+/-.005.
- 9. OUTLINE DEFINED IN SEPARATE GERBER FILE. DIMENSIONS OF CIRCUMSCRIBED RECTANGLE SHOWN ON THIS DWG FOR REF ONLY.
- 10. SEE SEPERATE DRILL FILE FOR HOLE LOCATIONS. SELECTED HOLE LOCATIONS SHOWN ON THIS DWG FOR REF
- 11. HOLE LOCATIONS SPECIFIED IN SEPERATE DRILL FILE TAKE PRECEDENCE OVER THIS DWG AND ARTWORK. CONTACT PURCHASER TO SEOLVE DIMENSION CONFLICT BETWEEN DWG AND ARTWORK.
- 12. HOLE SIZES ARE SPECIFIED AS FINAL DIMENSIONS AFTER PLATING AND FINISHING. UNLESS OTHERWISE SPECIFIED ALL HOLES TO BE PLATED.
- 13. VENDOR TO PLATE HOLES AND EXPOSED PADS. VENDOR TO SPECIFY TYPE OF PLATING MATERIAL TO BE USED WHEN ACCEPTING ORDER.
- 14. AFTER ASSEMBLY TRIM SOLDER FILLETS ADN COMPONENT LEADS TO 0.06" (MAX) BEYONG BOARD SURFACE ON BOTTOM SIDE.
- 15. DESIGN GEOMETRY MINIMUM FEATURE SIZES:

TRACE WIDTH; ANULUS	0.20 MM	TRACE-TO-TRACE: TRACE-TO-PAD	0.20 MM
HOLE-TO-HOLE	0.25 MM	PAD-TO-PAD	0.20 MM
SILKSCREEN LINE	0.20 MM	SILKSCREEN-TO-BARE COPPER	0.20 MM
MINIMUM HOLE SIZE (PLATED)	0.50 MM	MAXIMUM HOLE SIZE (UNPLATED)	0.81 MM
BOARD EDGE-TO-COPPER ´	0.30 MM	,	

Drawn by: Andrew Li Project: Instrumentation Buoy File: power-board.kicad_pcb Title: Power Board Size: A4 Date: 2020-04-25 Rev: C KiCad E.D.A. kicad (5.1.5-0-10_14) ld: 1/1

		1