



Drill Map:

- 0.200mm / 0.0079" (5 holes)
- 0.254mm / 0.0100" (39 holes)
- 0.400mm / 0.0157" (15 holes)
- 0.508mm / 0.0200" (90 holes)
- 1.000mm / 0.0394" (9 holes)
- 1.020mm / 0.0402" (40 holes)
- 1.067mm / 0.0420" (21 holes)
- 1.800mm / 0.0709" (2 holes) (not plated)
- 2.500mm / 0.0984" (4 holes) (not plated)
- 3.400mm / 0.1339" (8 holes) (not plated)

- NOTES: UNLESS OTHERWISE SPECIFIED.
- STANDARDS:
 - A. FABRICATE PCB IN ACCORDANCE WITH THE CURRENT REVISION OF IPC-6012, CLASS 2.
 - B. INTERPRET DIMENSIONS AND TOLERANCES IN ACCORDANCE WITH THE CURRENT REVISION OF ASME Y14.5M.
 - C. DO NOT SCALE DRAWING.
 - MATERIAL:
 - A. FR4 Tg 180 C OR EQUIVALENT.
 - B. EQUIVALENT MATERIAL SHALL BE RoHS COMPLIANT, HALOGEN FREE AND APPROVED BY YourCo.
 - C. THICKNESS OF INDIVIDUAL COPPER CLAD SHEETS SHALL BE IN AS DEFINED IN STACK-UP. SEE DETAIL A.
 - FLATNESS:
 - A. BOW AND TWIST OF ASSEMBLY SUB-PANEL OR SINGULATED PWB SHALL NOT EXCEED .025 MM PER MM.
 - B. TEST IN ACCORDANCE WITH THE CURRENT REVISION OF IPC-TM-650 2.4.22
 - ETCH GEOMETRY:
 - A. MEASURE WIDTH FROM THE BASE OF THE METALLIZATION.
 - B. MINIMUM LINE WIDTH: 0.nn MM OUTER, 0.nn MM INNER LAYERS.
 - C. FINISHED LINE WIDTH AND TERMINAL AREA SHALL NOT DEVIATE FROM THE 1-TO-1 MASTER PATTERN IMAGE BY MORE THAN +/- 0.025 MM OR 20%, WHICHEVER IS LESS.
 - SURFACE FINISH: (SELECT APPROPRIATE FINISH(ES))
 - A. ENIG PLATING IN ACCORDANCE WITH CURRENT REVISION OF IPC4556. EXPOSED METAL SHALL HAVE 118-236 MICRO INCHES ELECTROLESS NICKEL, 2-6 MICRO INCHES ELECTROLESS PALLADIUM, AND 1.2MICRO INCHES GOLD.
 - B. ENIG PLATING PER CURRENT REVISION OF IPC-4552. EXPOSED METAL SHALL HAVE 118-236 MICRO INCHES ELECTROLESS NICKEL AND 2-5 MICRO INCHES GOLD.
 - DESTRUCTIVE TESTING:
 - A. MICRO SECTION SAMPLE AND REPORT SHALL BE PROVIDED TO YourCo DESIGN ENGINEERING.
 - B. SOLDER SAMPLE PROCESSED THROUGH LEAD-FREE SOLDERING SHALL BE INCLUDED WITH EACH SHIPMENT.
 - C. X-OUT PANELS MAY BE USED FOR SOLDER SAMPLE.
 - HOLES:
 - A. PLATING IN HOLES SHALL BE CONTINUOUS ELECTROLYTIC COPPER WITH 0.025 MM MINIMUM BARREL THICKNESS.
 - B. MINIMUM FINISHED HOLE SIZE: 0.nn MM
 - C. HOLE SIZE MEASURED AFTER PLATING.
 - D. SEE DRILL CHART FOR FINISHED HOLE SIZE AND TOLERANCE.
 - E. ALL HOLES SHALL BE LOCATED WITHIN 0.08 MM OF TRUE POSITION AS SUPPLIED IN CAD DATA.
 - SOLDERMASK:
 - A. SOLDERMASK OVER BARE COPPER (SMOBC) ON PRIMARY AND SECONDARY SIDES USING SUPPLIED ARTWORK IN ACCORDANCE WITH CURRENT REVISION OF IPC-SM-840 TYPE B
 - B. COLOR: MATTE GREEN
 - C. LIQUID PHOTO-IMAGEABLE (LPI) 0.001 MM TO 0.002 MM THICKNESS, HALOGEN FREE
 - D. NO BLEED-OUT ALLOWED OVER EXPOSED SMD PADS.
 - E. NO EXPOSED TRACES.
 - SILKSCREEN:
 - A. SILKSCREEN PRIMARY AND SECONDARY SIDE WITH WHITE EPOXY, NON-CONDUCTIVE, NON-NUTRIENT INK.
 - B. ANY UNSPECIFIED STROKE WIDTH SHALL BE 0.13 MM
 - C. CLIP SILKSCREEN AWAY FROM ANY EXPOSED METAL.
 - D. VENDOR DATE CODE, LOGO, UL AND ANY ADDITIONAL MARKING TO BE LOCATED ON THE SECONDARY SIDE.
 - E. BAG AND TAG ACCEPTABLE FOR PWBS THAT ARE TOO SMALL FOR MARKING.
 - REMOVE ALL BURRS AND BREAK SHARP EDGES R0.003 MIN.
 - NON-DESTRUCTIVE EVALUATION:
 - A. ALL PWBS SHALL PASS 100% ELECTRICAL TEST USING SUPPLIED IPC-356 NETLIST IN ACCORDANCE WITH CURRENT REVISION OF IPC-9252, CLASS 2.
 - B. CERTIFICATE OF CONFORMANCE SHALL BE SUPPLIED WITH EACH SHIPMENT.
 - X-OUTS:
 - A. X-OUT BOARDS THAT DO NOT MEET ALL SPECIFICATIONS USING PERMANENT MARKING ON BOTH SIDES OF THE AFFECTED PCB.
 - B. PANELS THAT DO NOT HAVE ANY X-OUTS SHALL BE PACKAGED TOGETHER.
 - C. PANELS THAT HAVE n OR FEWER X-OUTS SHALL BE PACKAGED SEPARATE FROM NON-X-OUT PANELS.
 - D. PANELS WITH MORE THAN n X-OUTS SHALL BE REJECTED.
 - PACKAGING REQUIREMENTS:
 - A. PWBS SHALL BE PACKAGED IN VACUUM SEALED INNER CONTAINERS.
 - B. OUTER CONTAINERS SHALL BE SUFFICIENT TO PREVENT DAMAGE DURING SHIPPING AND HANDLING.
 - IMPEDANCE (ALL TOLERANCES +/- 10%)
 - A. ALL 0.nn MM WIDE TRACES ON OUTER LAYERS SHALL BE 50 OHMS.
 - B. ALL 0.nn MM WIDE/0.nn MM SPACE PAIRS ON OUTER LAYERS SHALL BE 90 OHMS.
 - C. ALL 0.nn MM WIDE/0.nn MM SPACE PAIRS ON INNER LAYERS SHALL BE 90 OHMS.
 - D. VENDOR MAY ADJUST DESIGN GEOMETRIES UP TO +/-20% TO ACHIEVE TARGET IMPEDANCE. ADJUSTMENTS BEYOND 20% OF LINE WIDTH, SPACING OR DIELECTRIC THICKNESS SHALL REQUIRE APPROVAL FROM YourCo ENGINEERING.

Layer Name	Type	Material	Thickness (mm)	Color	Epsilon R	Loss Tangent
F.Silkscreen	Top Silk Screen	Not specified	0 mm	Not specified	1	0
F.Paste	Top Solder Paste		0 mm		1	0
F.Mask	Top Solder Mask	Not specified	0.01 mm	Not specified	3.3	0
F.Cu	copper		0.035 mm		1	0
Dielectric	core	FR4	0.508 mm	Not specified	4.5	0.02
In1.Cu	copper		0.035 mm		1	0
Dielectric	pregreg	FR4	0.508 mm	Not specified	4.5	0.02
In2.Cu	copper		0.035 mm		1	0
Dielectric	core	FR4	0.508 mm	Not specified	4.5	0.02
B.Cu	copper		0.035 mm		1	0
B.Mask	Bottom Solder Mask	Not specified	0.01 mm	Not specified	3.3	0
B.Paste	Bottom Solder Paste		0 mm		1	0
B.Silkscreen	Bottom Silk Screen	Not specified	0 mm	Not specified	1	0



Sheet:
File: PI-Power-Board.kicad_pcb

Title:

Size: B Date: Rev:
KiCad E.D.A. kicad-cli 7.0.10-7.0.10-ubuntu22.04.1 Id: 1/1