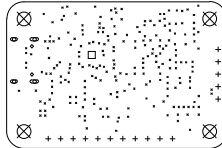


DRILL MAP VIEW



Drill Map:

- 0.300mm / 0.0118" (230 holes)
- 0.600mm / 0.0236" (0 holes + 4 slots)
- 1.000mm / 0.0394" (16 holes)
- 2.000mm / 0.0787" (1 hole)
- 0.650mm / 0.0256" (2 holes) (not plated)
- ⊗ 2.700mm / 0.1063" (4 holes) (not plated)

Layer Name	Type	Material	Thickness (mm)	Color	Epsilon R	Loss Tangent
F.Silkscreen	Top Silk Screen	Liquid Photo	0 mm	White	1	0
F.Paste	Top Solder Paste		0 mm		1	0
F.Mask	Top Solder Mask	Epoxy (JLC)	0.015 mm	Black	3.8	0.02
F.Cu	copper		0.035 mm		1	0
Dielectric 1	prepreg	FR4 (JLC 7628)	0.2104 mm	FR4 natural	4,4	0.02
In1.Cu	copper		0.0152 mm		1	0
Dielectric 2	core	FR4 (JLC Core)	0.4 mm	FR4 natural	4,6	0.02
In2.Cu	copper		0.0152 mm		1	0
Dielectric 3	prepreg	FR4 (JLC 7628)	0.2028 mm	FR4 natural	4,4	0.02
In3.Cu	copper		0.0152 mm		1	0
Dielectric 4	core	FR4 (JLC Core)	0.4 mm	FR4 natural	4,6	0.02
In4.Cu	copper		0.0152 mm		1	0
Dielectric 5	prepreg	FR4 (JLC 7628)	0.2104 mm	FR4 natural	4,4	0.02
B.Cu	copper		0.035 mm		1	0
B.Mask	Bottom Solder Mask	Epoxy (JLC)	0.015 mm	Black	3.8	0.02
B.Paste	Bottom Solder Paste		0 mm		1	0
B.Silkscreen	Bottom Silk Screen	Liquid Photo	0 mm	White	1	0


BOARD CHARACTERISTICS

Copper Layer Count: 6 Board Thickness: 1.5844 mm
Board overall dimensions: 45.0000 mm x 30.0000 mm
Min track/spacing: 0.1500 mm / 0.1500 mm Min hole diameter: 0.3000 mm
Copper Finish: ENIG Impedance Control: Yes
Castellated pads: No Plated Board Edge: No
Edge card connectors: No

ALL 0.3MM VIAS TO BE EPOXY FILLED AND CAPPED WITH COPPER.

NOTES: UNLESS OTHERWISE SPECIFIED.

- MATERIAL:
 - A. FR4 Tg 150 C OR EQUIVALENT.
 - B. EQUIVALENT MATERIAL SHALL BE RoHS COMPLIANT, HALOGEN FREE AND APPROVED BY PANTS FOR BIRDS LLC.
 - C. THICKNESS OF INDIVIDUAL COPPER CLAD SHEETS SHALL BE IN AS DEFINED IN STACK-UP.
- ETCH GEOMETRY:
 - A. MEASURE WIDTH FROM THE BASE OF THE METALIZATION.
 - B. MINIMUM LINE WIDTH: 0.15 MM OUTER, 0.15 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:
 - A. ENIG PLATING PER CURRENT REVISION OF IPC-4552. EXPOSED METAL SHALL HAVE 118-236 MICRO INCHES ELECTROLESS NICKEL AND 2-5 MICRO INCHES GOLD.
- IMPEDANCE (ALL TOLERANCES +/- 10%)
 - A. ALL 0.35 MM WIDE/0.2 MM SPACE PAIRS ON OUTER LAYERS SHALL BE 90 OHMS DIFFERENTIAL.
 - B. ALL 0.6 MM WIDE/0.15 MM SPACE TRACES ON OUTER LAYERS SHALL BE 50 OHMS SINGLE ENDED.
 - C. 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 PANTS FOR BIRDS LLC.
- HOLES:
 - A. PLATING IN HOLES SHALL BE CONTINUOUS ELECTROLYTIC COPPER WITH 0.025 MM MINIMUM BARREL THICKNESS.
 - B. MINIMUM FINISHED HOLE SIZE: 0.20 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.
 - F. ALL HOLES < 1.0MM DIAMETER SHALL BE EPOXY FILLED AND CAPPED WITH COPPER.
- 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 BLACK
 - 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 PCBs THAT ARE TOO SMALL FOR MARKING.
- 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.
- FLATNESS:
 - A. BOW AND TWIST OF ASSEMBLY SUB-PANEL OR SINGULATED PCB SHALL NOT EXCEED .025 MM PER MM.
 - B. TEST IN ACCORDANCE WITH THE CURRENT REVISION OF IPC-TM-650 2.4.22
- DESTRUCTIVE TESTING:
 - A. SOLDER SAMPLE PROCESSED THROUGH SOLDERING SHALL BE INCLUDED WITH EACH SHIPMENT.
 - B. X-OUT PANELS MAY BE USED FOR SOLDER SAMPLE.
- REMOVE ALL BURRS AND BREAK SHARP EDGES R0.003 MIN.
- NON-DESTRUCTIVE EVALUATION:
 - A. ALL PCBs 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 5 OR FEWER X-OUTS SHALL BE PACKAGED SEPARATE FROM NON-X-OUT PANELS.
 - D. PANELS WITH MORE THAN 5 X-OUTS SHALL BE REJECTED.
- PACKAGING REQUIREMENTS:
 - A. PCBs SHALL BE PACKAGED IN VACUUM SEALED INNER CONTAINERS.
 - B. OUTER CONTAINERS SHALL BE SUFFICIENT TO PREVENT DAMAGE DURING SHIPPING AND HANDLING.

	John McNelly - john@pantsforbirds.com Pants for Birds
Sheet: File: adsbee_1090.kicad_pcb	
Title: ADSBee 1090	
Size: B	Date: 2024-06-22
KiCad E.D.A. 8.0.3	Rev: D Id: 1/1