

NOTES: UNLESS OTHERWISE SPECIFIED.

- MATERIAL:
 - FR4 Tg 150 C OR EQUIVALENT.
 - EQUIVALENT MATERIAL SHALL BE RoHS COMPLIANT, HALOGEN FREE AND APPROVED BY PANTS FOR BIRDS LLC.
 - THICKNESS OF INDIVIDUAL COPPER CLAD SHEETS SHALL BE IN AS DEFINED IN STACK-UP.
- ETCH GEOMETRY:
 - MEASURE WIDTH FROM THE BASE OF THE METALIZATION.
 - MINIMUM LINE WIDTH: 0.15 MM OUTER, 0.15 MM INNER LAYERS.
 - 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:
 - 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%)
 - 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.
 - ALL 0.4 MM WIDE/0.32 MM SPACE MICROSTRIP TRACES ON OUTER LAYERS SHALL BE 50 OHMS SINGLE ENDED.
 - ALL 0.26 MM WIDE/0.15 MM SPACE DIFFERENTIAL PAIRS ON OUTER LAYERS SHALL BE 90 OHMS DIFFERENTIAL.
- HOLES:
 - PLATING IN HOLES SHALL BE CONTINUOUS ELECTROLYTIC COPPER WITH 0.025 MM MINIMUM BARREL THICKNESS.
 - MINIMUM FINISHED HOLE SIZE: 0.20 MM
 - HOLE SIZE MEASURED AFTER PLATING.
 - SEE DRILL CHART FOR FINISHED HOLE SIZE AND TOLERANCE.
 - ALL HOLES SHALL BE LOCATED WITHIN 0.08 MM OF TRUE POSITION AS SUPPLIED IN CAD DATA.
 - ALL HOLES < 1.0MM DIAMETER SHALL BE EPOXY FILLED AND CAPPED WITH COPPER.
- SOLDERMASK:
 - SOLDERMASK OVER BARE COPPER (SMOBC) ON PRIMARY AND SECONDARY SIDES USING SUPPLIED ARTWORK IN ACCORDANCE WITH CURRENT REVISION OF IPC-SM-840 TYPE B.
 - COLOR: MATTE BLACK
 - LIQUID PHOTO-IMAGEABLE (LPI) 0.001 MM TO 0.002 MM THICKNESS, HALOGEN FREE
 - NO BLEED-OUT ALLOWED OVER EXPOSED SMD PADS.
 - NO EXPOSED TRACES.
- SILKSCREEN:
 - SILKSCREEN PRIMARY AND SECONDARY SIDE WITH WHITE EPOXY, NON-CONDUCTIVE, NON-NUTRIENT INK.
 - ANY UNSPECIFIED STROKE WIDTH SHALL BE 0.13 MM
 - CLIP SILKSCREEN AWAY FROM ANY EXPOSED METAL.
 - VENDOR DATE CODE, LOGO, UL AND ANY ADDITIONAL MARKING TO BE LOCATED ON THE SECONDARY SIDE.
 - BAG AND TAG ACCEPTABLE FOR PCBs THAT ARE TOO SMALL FOR MARKING.
- STANDARDS:
 - FABRICATE PCB IN ACCORDANCE WITH THE CURRENT REVISION OF IPC-6012, CLASS 2.
 - INTERPRET DIMENSIONS AND TOLERANCES IN ACCORDANCE WITH THE CURRENT REVISION OF ASME Y14.5M.
 - DO NOT SCALE DRAWING.
- FLATNESS:
 - BOW AND TWIST OF ASSEMBLY SUB-PANEL OR SINGULATED PCB SHALL NOT EXCEED .025 MM PER MM.
 - TEST IN ACCORDANCE WITH THE CURRENT REVISION OF IPC-TM-650 2.4.22
- DESTRUCTIVE TESTING:
 - SOLDER SAMPLE PROCESSED THROUGH SOLDERING SHALL BE INCLUDED WITH EACH SHIPMENT.
 - X-OUT PANELS MAY BE USED FOR SOLDER SAMPLE.
- REMOVE ALL BURRS AND BREAK SHARP EDGES R0.003 MIN.
- NON-DESTRUCTIVE EVALUATION:
 - ALL PCBs SHALL PASS 100% ELECTRICAL TEST USING SUPPLIED IPC-356 NETLIST IN ACCORDANCE WITH CURRENT REVISION OF IPC-9252, CLASS 2.
 - CERTIFICATE OF CONFORMANCE SHALL BE SUPPLIED WITH EACH SHIPMENT.
- X-OUTS:
 - X-OUT BOARDS THAT DO NOT MEET ALL SPECIFICATIONS USING PERMANENT MARKING ON BOTH SIDES OF THE AFFECTED PCB.
 - PANELS THAT DO NOT HAVE ANY X-OUTS SHALL BE PACKAGED TOGETHER.
 - PANELS THAT HAVE 2 OR FEWER X-OUTS SHALL BE PACKAGED SEPARATE FROM NON-X-OUT PANELS.
 - PANELS WITH MORE THAN 2 X-OUTS SHALL BE REJECTED.
- PACKAGING REQUIREMENTS:
 - PCBS SHALL BE PACKAGED IN VACUUM SEALED INNER CONTAINERS.
 - OUTER CONTAINERS SHALL BE SUFFICIENT TO PREVENT DAMAGE DURING SHIPPING AND HANDLING.

Layer Name	Type	Material	Thickness (mm)	Color	Epsilon R	Loss Tangent
F.Silkscreen	Top Silk Screen	Not specified	0 mm	White	1	0
F.Paste	Top Solder Paste		0 mm		1	0
F.Mask	Top Solder Mask	Not specified	0.01 mm	Black	3.3	0
F.Cu	copper		0.035 mm		1	0
Dielectric 1	prepreg	JLC7628	0.2104 mm	FR4 natural	4.4	0
In1.Cu	copper		0.0152 mm		1	0
Dielectric 2	core	JLC Core	1.065 mm	FR4 natural	4.6	0
In2.Cu	copper		0.0152 mm		1	0
Dielectric 3	prepreg	JLC7628	0.2104 mm	FR4 natural	4.4	0
B.Cu	copper		0.035 mm		1	0
B.Mask	Bottom Solder Mask	Not specified	0.01 mm	Black	3.3	0
B.Paste	Bottom Solder Paste		0 mm		1	0
B.Silkscreen	Bottom Silk Screen	Not specified	0 mm	White	1	0

BOARD CHARACTERISTICS

Copper Layer Count:	4	Board Thickness:	1.6062 mm
Board overall dimensions:	62.5 mm x 33 mm		
Min track/spacing:	0.15 mm / 0.15 mm	Min hole diameter:	0.3 mm
Copper Finish:	ENIG	Impedance Control:	Yes
Castellated pads:	No	Plated Board Edge:	No
Edge card connectors:	No		

ALL 0.3MM VIAS MUST BE EPOXY FILLED AND CAPPED.



PCBA: 010250012-C
PCB: 010250013-A
John McNelly - john@pantsforbirds.com
Pants for Birds

Sheet:
File: adsbee_m1090_eval_board.kicad_pcb

Title: ADSBee m1090 Eval Board


Size: A Date: 2026-05-09

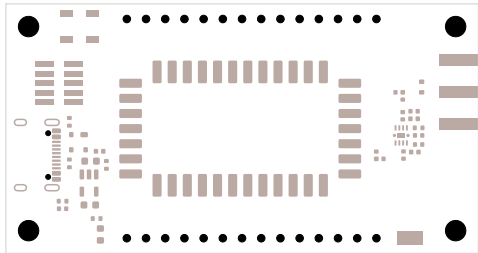
KiCad E.D.A. 10.0.1

Rev: C

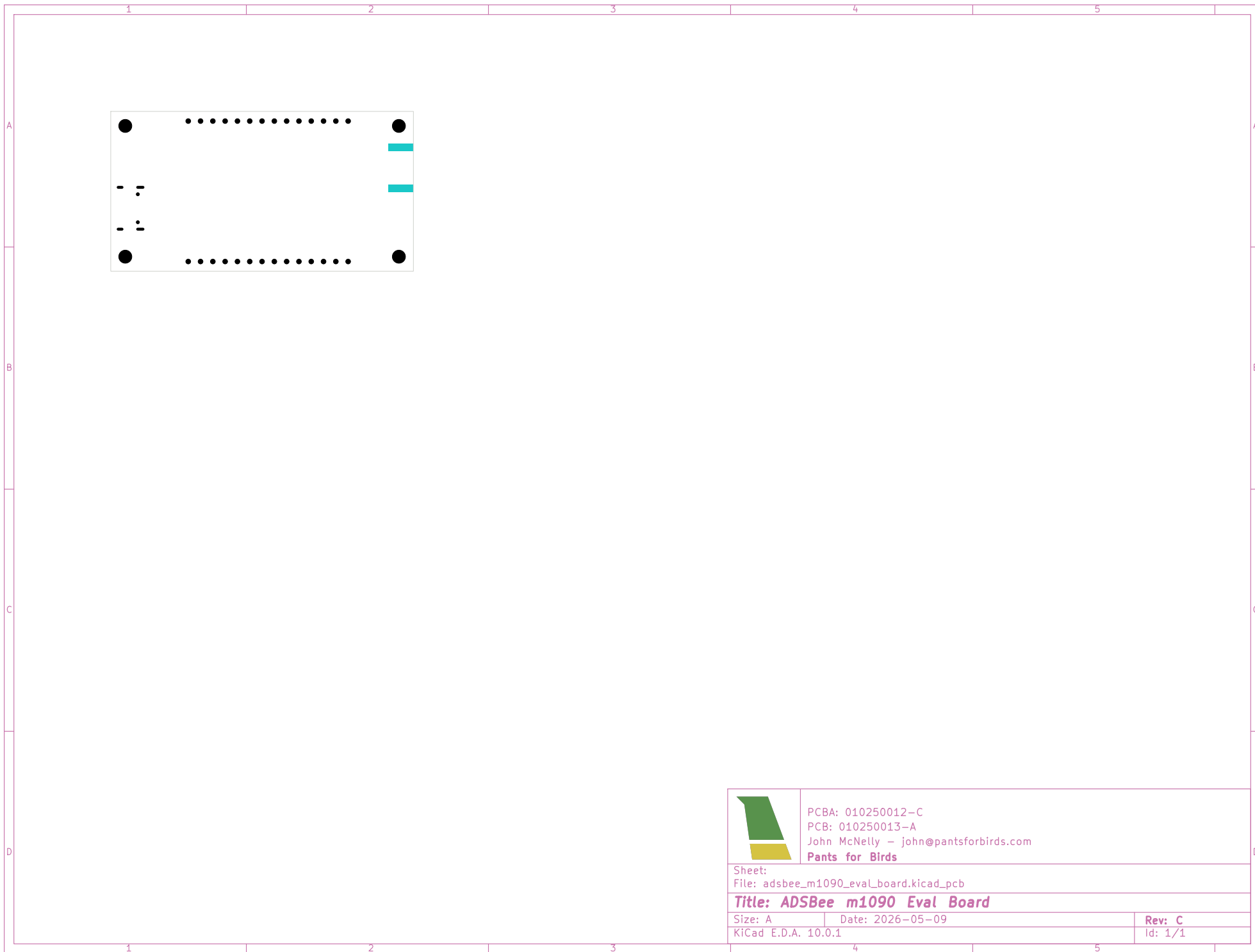
Id: 1/1



		PCBA: 010250012-C PCB: 010250013-A John McNelly – john@pantsforbirds.com Pants for Birds	
Sheet: File: adsbee_m1090_eval_board.kicad_pcb			
Title: ADSBee m1090 Eval Board			
Size: A		Date: 2026-05-09	
KiCad E.D.A. 10.0.1		Rev: C Id: 1/1	
4		5	

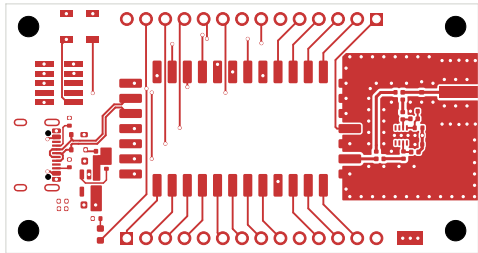


	PCBA: 010250012-C	
	PCB: 010250013-A	
John McNelly - john@pantsforbirds.com		
Pants for Birds		
Sheet:		
File: adsbee_m1090_eval_board.kicad_pcb		
Title: ADSBee m1090 Eval Board		
Size: A	Date: 2026-05-09	Rev: C
KiCad E.D.A. 10.0.1		Id: 1/1



PCBA: 010250012-C
PCB: 010250013-A
John McNelly - john@pantsforbirds.com
Pants for Birds

Sheet: File: adsbee_m1090_eval_board.kicad_pcb		
Title: ADSBee m1090 Eval Board		
Size: A	Date: 2026-05-09	Rev: C
KiCad E.D.A. 10.0.1		Id: 1/1



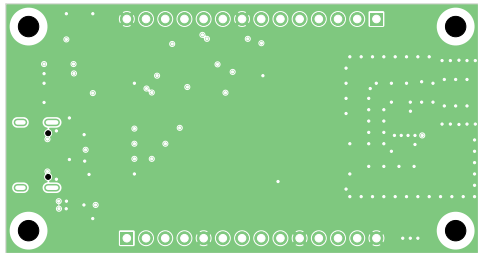
PCBA: 010250012-C
PCB: 010250013-A
John McNelly - john@pantsforbirds.com
Pants for Birds

Sheet:
File: adsbee_m1090_eval_board.kicad_pcb

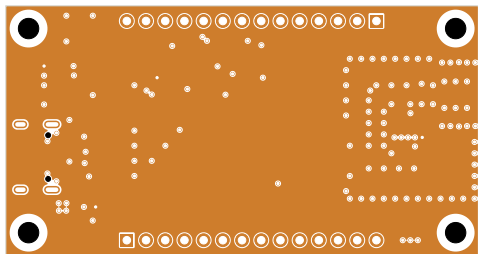
Title: ADSBee m1090 Eval Board

Size: A Date: 2026-05-09
KiCad E.D.A. 10.0.1

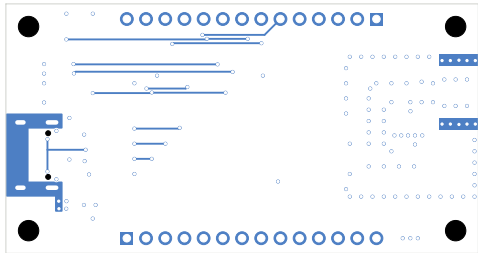
Rev: C
Id: 1/1



	PCBA: 010250012-C		
	PCB: 010250013-A		
	John McNelly - john@pantsforbirds.com		
	Pants for Birds		
Sheet:			
File: adsbee_m1090_eval_board.kicad_pcb			
Title: ADSBee m1090 Eval Board			
Size: A		Date: 2026-05-09	
KiCad E.D.A. 10.0.1		Rev: C	
		Id: 1/1	



		PCBA: 010250012-C PCB: 010250013-A John McNelly – john@pantsforbirds.com Pants for Birds	
Sheet: File: adsbee_m1090_eval_board.kicad_pcb			
Title: ADSBee m1090 Eval Board			
Size: A		Date: 2026-05-09	
KiCad E.D.A. 10.0.1		Rev: C Id: 1/1	
4		5	



	PCBA: 010250012-C		
	PCB: 010250013-A		
	John McNelly - john@pantsforbirds.com		
	Pants for Birds		
Sheet:			
File: adsbee_m1090_eval_board.kicad_pcb			
Title: ADSBee m1090 Eval Board			
Size: A		Date: 2026-05-09	
KiCad E.D.A. 10.0.1		Rev: C	
		Id: 1/1	