

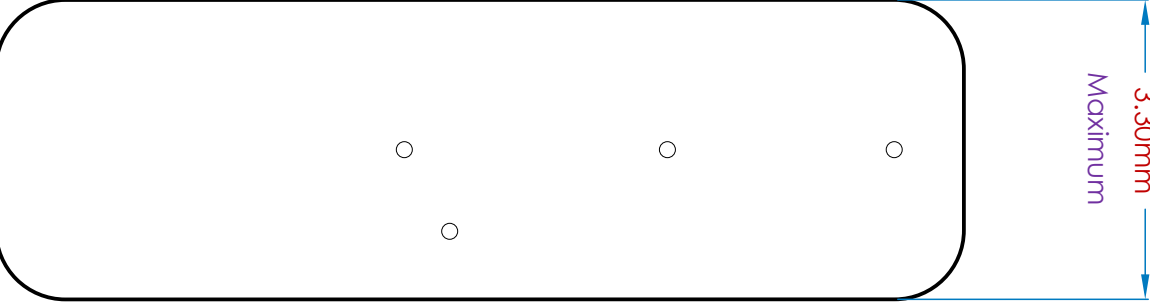
PCB FABRICATION NOTES

1.
- MATERIAL: FR4, RoHS, AT LEAST Tg150C
2.
- ACCEPTANCE CRITERIA PER IPC-600 CLASS 2
3.
- LAYER STACKUP: SEE LAYER STACK LEGEND FOR ESTIMATED COPPER WEIGHT AND DIELECTRIC THICKNESS
4.
- FINISH: ENIG PLATING PER IPC-4552
5.
- HOLES: FINISHED SIZE AS SHOWN
6.
- SOLDERMASK: SMOBC, TWO SIDES, Per IPC-SM-840, GREEN LPI
7.
- SILKSCREEN: NONE
8.
- ROUTING: PLACE TABS/BITES ON SHORT EDGES TO MAINTAIN MAXIMUM WIDTH OF 3.3MM

Layer Stack Legend

	Material	Layer	Thickness	Dielectric Material	Type	Gerber
	Surface Material	Top Solder	0.02mm	Solder Resist	Solder Mask	GTS
	Copper	Top Layer	0.04mm		Signal	GTL
			1.57mm	FR-4	Dielectric	
	Copper	Bottom Layer	0.04mm		Signal	GBL
	Surface Material	Bottom Solder	0.02mm	Solder Resist	Solder Mask	GBS
Total thickness: 1.68mm						

Drill Drawing View (Scale 12:1)

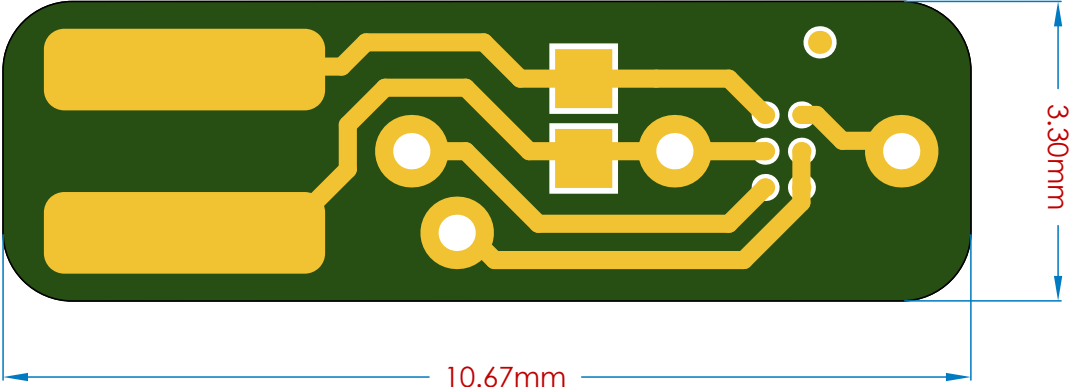


Drill Table

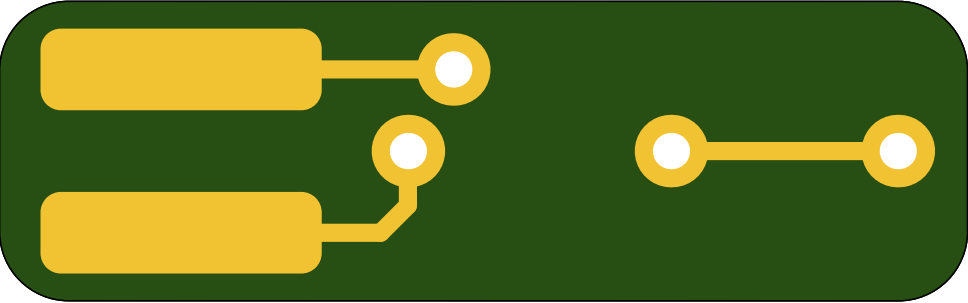
Symbol	Count	Hole Size	Plated	Hole Tolerance
○	4	0.40mm	Plated	
	4 Total			

REVISION	DESCRIPTION	DATE
1	First Version	2022-09-16
1.1	Change C1 from 35V to 10V, layout remains the same - RTH	2023-12-08
2.1	Notes for open sourcing; Rev # to match main PCB - RTH	2025-06-03

Top Layer (Scale 12:1)



Bottom Layer (Scale 12:1)



Licensed under CERN-OHL-P v2
Source package at <https://docs.2to8.cc>

New Horizons

PROJECT
AMS Sensor

REV
2.1

Rod Hinman
808-989-0977
rhinman@nhgh.org
2025-06-03