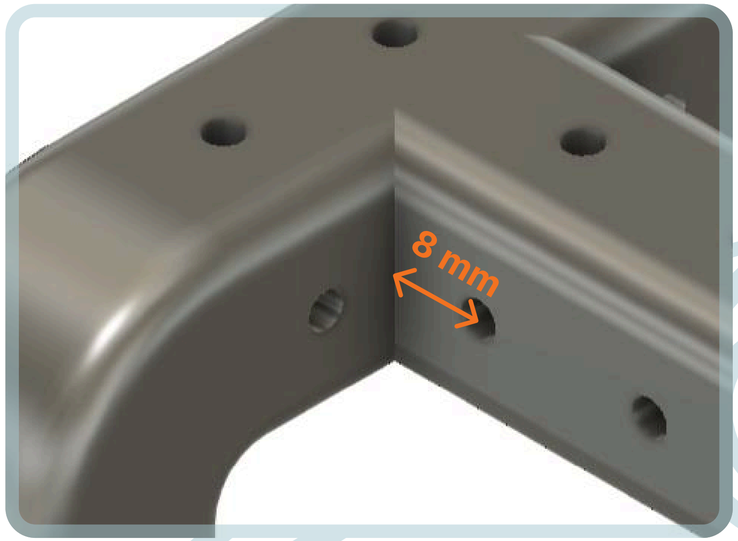
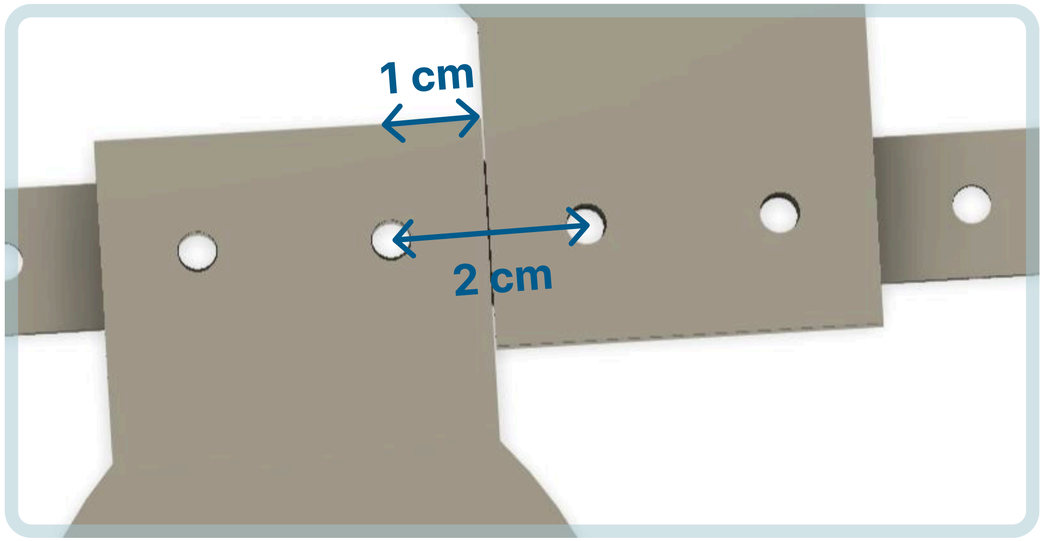
How-To Design and Print Sensor Mounts

How-To Design Sensor Mounts

Distances



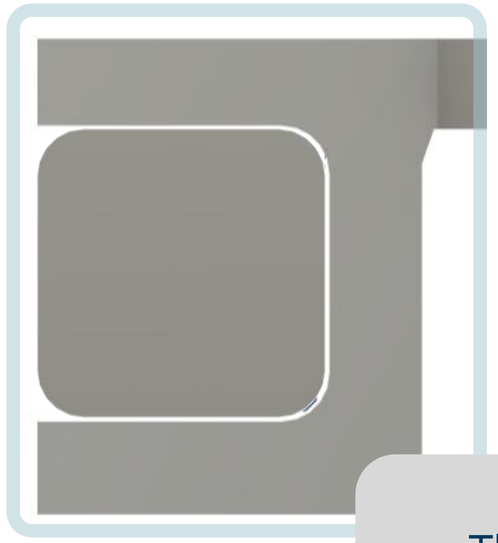
The space between a hole and a corner is less than 1 cm due to the thickness of the walls, be mindful of this if designing for parts that might need to be near a corner



The space between two holes are 2 cm, when designing parts, its good to have the space from the hole to the edge be 1 cm to allow for other fasteners beside it.

How-To Design Sensor Mounts

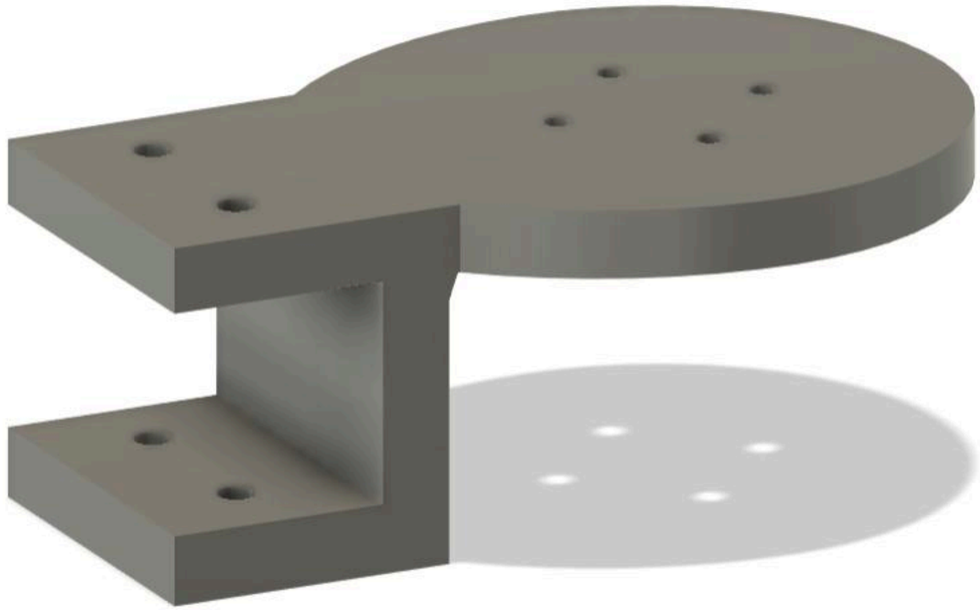
Fillet



The connecting pieces have a fillet of 2.5 mm, when designing parts around it, use a fillet of the same size on inner corners.

How-To Design Sensor Mounts

Example fastener

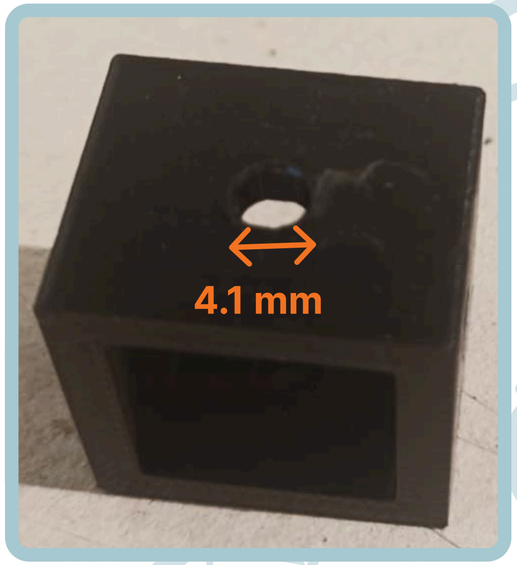


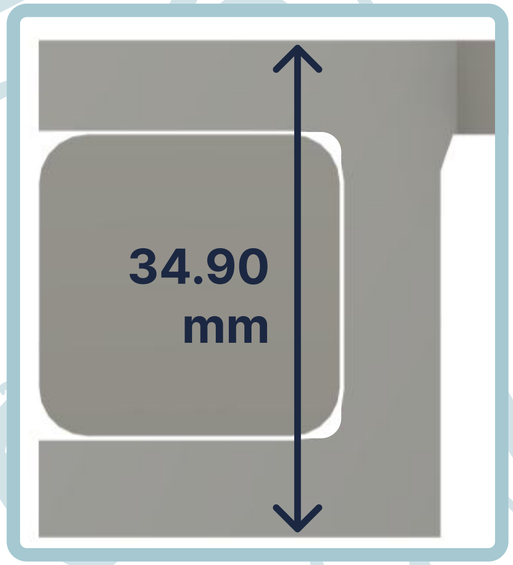
This is a simple and good attachment method for lighter sensors that does not require much material and offers many placement options.

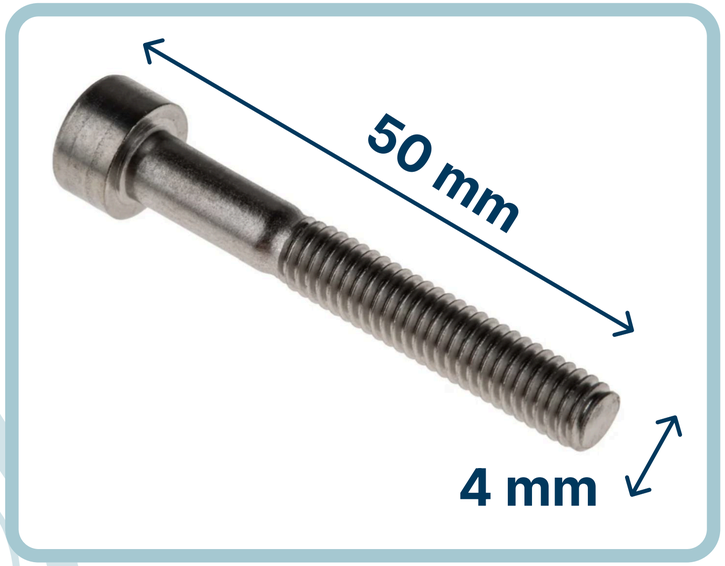
How-To Design Sensor Mounts

Screws and nuts

M4 screws and nuts are good for fastening the sensor mounts
screws: https://no.rs-online.com/web/p/socket-screws/1247349
nuts: https://no.rs-online.com/web/p/hex-nuts/0189579?gb=s







How-To Print Sensor Mounts

Printer settings to be mindful of

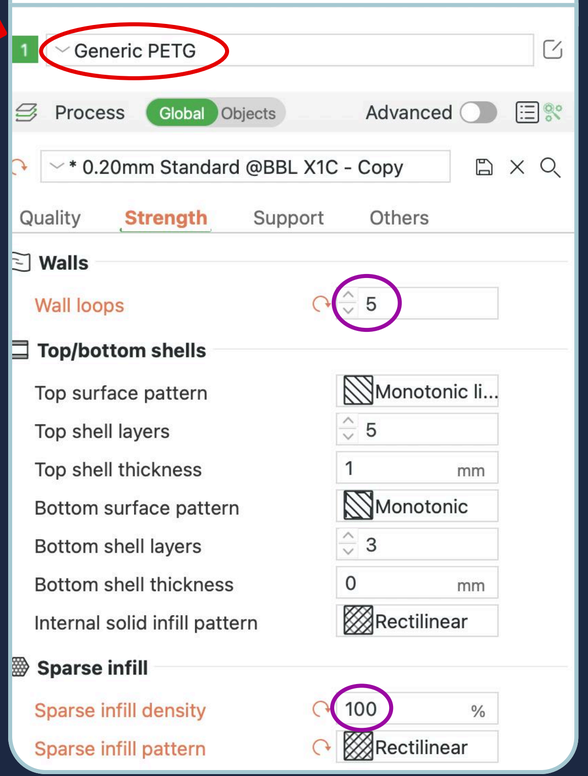
1. Import file into your preferred slicer program.

2. Settings to be mindful of:
   a. **Filament**. Recommended to use PETG or PETG-CF. These work well in saltwater.
   b. **100% infill vs. open model and gyroid infill**
   To prevent implosion in deeper waters, the parts should either be 100% infill **or** open to let water run through. Which setting you should use when depends on the desired durability, and which one we recommend for which part is specified in the guides on the next slides.

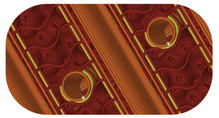
Note:
The examples are from the bamboo slicer but equivalent settings will exist on every slicer program. You only need to change the specified settings, the rest is fine as-is.

Airtight and 100% infill

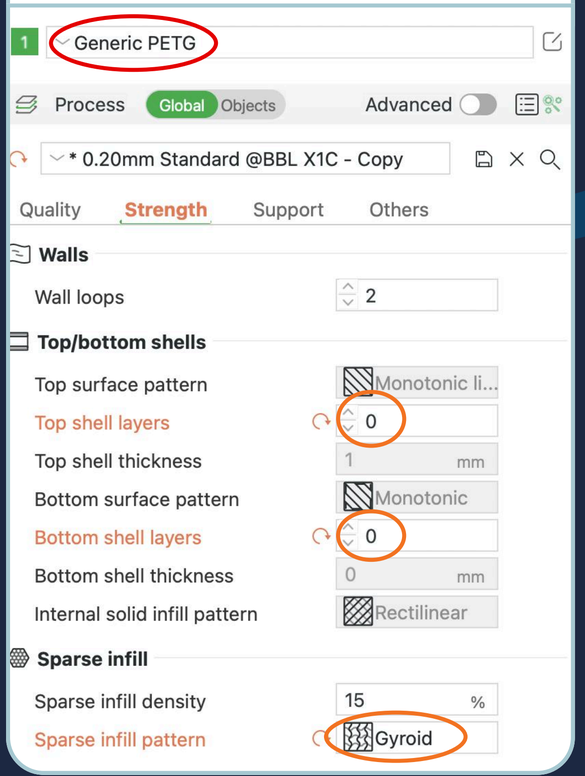
To make the parts airtight increase infill density to 100%. It is also recommended to set wall loops/thickness to 5 for increased durability.



Open model and gyroid infill

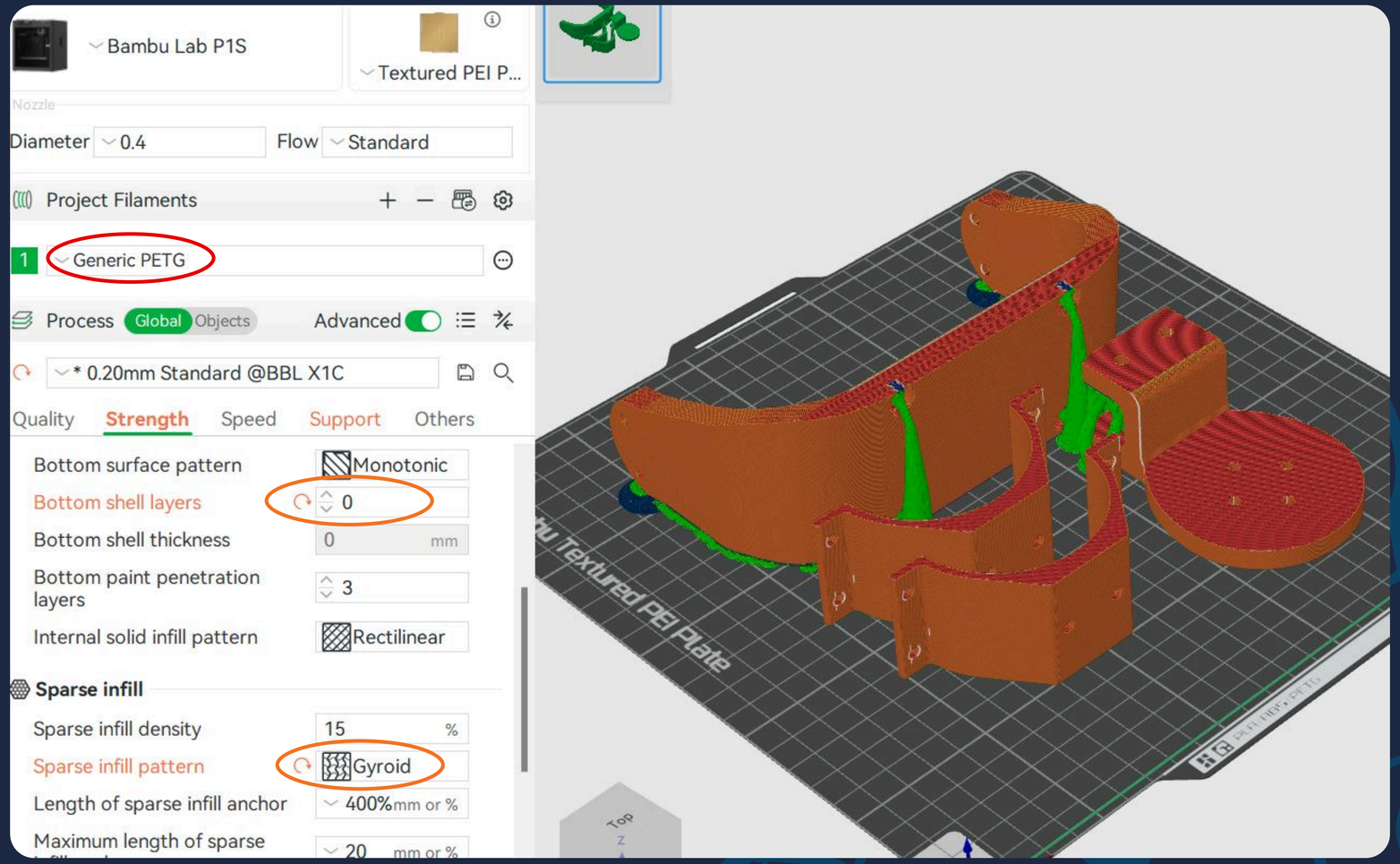


To make an open solution, set top and/or bottom shell layer to 0 and change sparse infill pattern to "Gyroid".



How-To Print Sensor Mounts

Several in one bed



Time: ca. 4h and 7 minutes

Edited printer settings

Recommended to use the **Open model** settings

Bottom shell layers: 0
Infill pattern: gyroid
support enabled
support type: tree/organic

(the rest is fine as is)