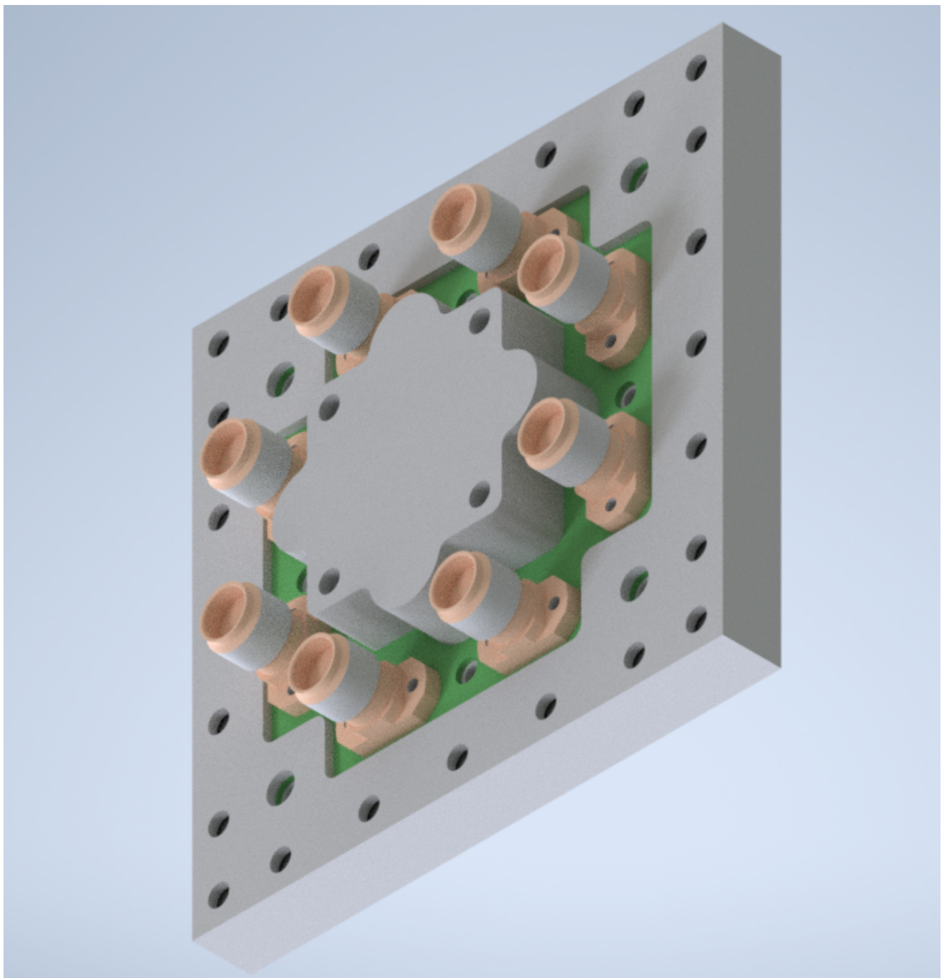
OQTO User Manual

Linus Andersson, anlinus@chalmers.se
Date: 2022-12-16



1 Overview of OQTO

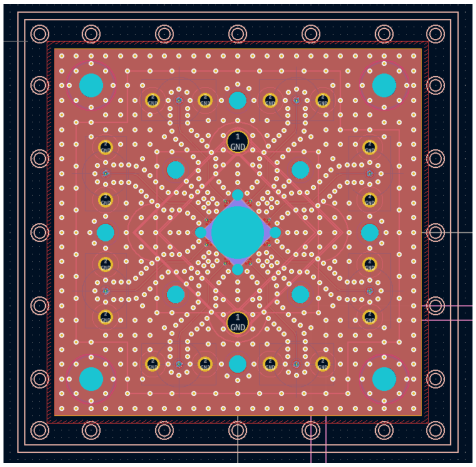
The OQTO sample holder is an 8-port sample holder developed in the 202Q-lab for cryogenic measurements supporting 7x7mm and 10x10mm chip sizes. It is a PCB-based design with additional copper parts, occupying a total area of 6x6cm, regardless of chip size. On the PCB, the signals are carried by stripline waveguides for reduced cross-talk between the different ports, with additional via-shielding to improve the leakage between lines. In addition, the bondwire pads are designed to be slightly capacitive to compensate for bondwire inductance. The striplines are excited with a solderless, vertical spring-loaded SMA connector (2921-40049-1S) from SV Microwave, which is non-magnetic. The PCB is coated with nickel-less gold plating, making the entire sample holder, including the copper parts, non-magnetic. The copper pieces are four in total, consisting of a back-plate, a lid covering the chip, a ridge surrounding the chip, and a final plate on the top to clamp the PCB to the bottom copper piece. The purpose of releasing this sample holder in an open-source format is to enable other research groups to manufacture and use this sample holder to be used in their research. Please read Section 5 for terms of usage.

2 PCB design

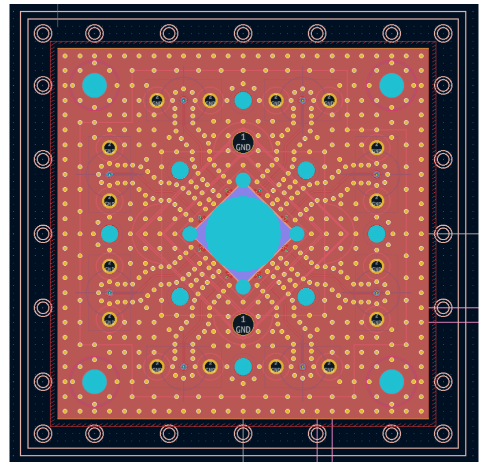
PCB Layer #	Material	Thickness [μm]	ϵ_r	tan(δ)
Metal 1	Cu (BF-TZA)	35	n/a	n/a
Substrate 1	Astra MT77 (Prepreg 1067)	78.75	2.91	0.0018
Metal 2	Cu (Core)	15	n/a	n/a
Substrate 2	Astra MT77 (Core)	381	3.00	0.0017
Metal 3	Cu (Core)	15	n/a	n/a
Substrate 3	Astra MT77 (Prepreg 1067)	78.75	2.91	0.0018
Metal 4	Cu (BF-TZA)	35	n/a	n/a

Table 1: Material stack-up from top (Metal 1) to bottom (Metal 4) of PCB. When we talk about different layers of the PCB, we refer to the metal layers.

As mentioned above, the PCB carries eight signal ports to the chip. It comes in two versions depending on if you use a 7x7mm or 10x10mm chip size, seen in Figure 1a and 1b respectively. The transmission lines are excited from the top layer of the PCB with vertical SMA connectors from SV Microwave by a transition consisting of a micro-via connecting the top layer to the second layer. This transition is EM simulated to provide as close to 50 Ω as possible. This was done in the simulation software Comsol Multiphysics [2], and you can find the simulation files included among the shared files. In Table 1, you find the PCB build-up. The PCB build-up that is used in the design consists of 4 layers of copper using the dielectric Astra MT77 from Isola Group. When it comes to the bondwire pads, there is an additional via-transition to the top copper layer where you will bond to your chip. These pads are made slightly capacitive, to compensate for the inductance added by the bondwires. All the vias on this PCB are filled, plated vias. We use filled vias to avoid solder from leaking through to the other side of the PCB in the soldering process, described in Section 4. The chip-to-PCB clearance is 100 um, so you have a 100 um margin on each side of the chip (7x7mm or 10x10mm). The PCBs are designed in KiCad [3], which is an open-source EDA software. All the KiCad design files are provided in the shared folder.



(a) 7x7mm chip PCB-layout in KiCad.



(b) 10x10mm chip PCB-layout in KiCad.

Figure 1: PCB layouts in KiCad.

3 Mechanical designs

The metallic parts of the sample holder are designed in the CAD software Inventor from Autodesk. Firstly, the parts consist of a bottom plate where the PCB and chip is to be placed. To suppress resonant modes in the chip substrate, we have placed a cavity underneath the chip, which reduces the effective dielectric constant. This gives a resonant frequency of 22.3 GHz for the 10x10mm chip and 30 GHz for the 7x7mm chip, which was simulated in Comsol Multiphysics for a 280 um thick silicon chip (simulation files provided in shared folder). The cavities above the chips are modeled as a rectangular cavity resonators, with a dominant resonance frequency of 17.6 GHz (12x12x6mm) and 24.9 GHz (8.5x8.5x6mm) for 10x10mm and 7x7mm chip sizes respectively. The shelf where the chip is resting was made slightly asymmetric so that when the PCB is placed on the bottom copper piece, you can push the PCB to one of the corners of the shelf, see Figure 2. This was done in order to have some tolerance when placing the chip. By aligning the chip in the center of this space, you will have a 100 um gap to each side of the chip.

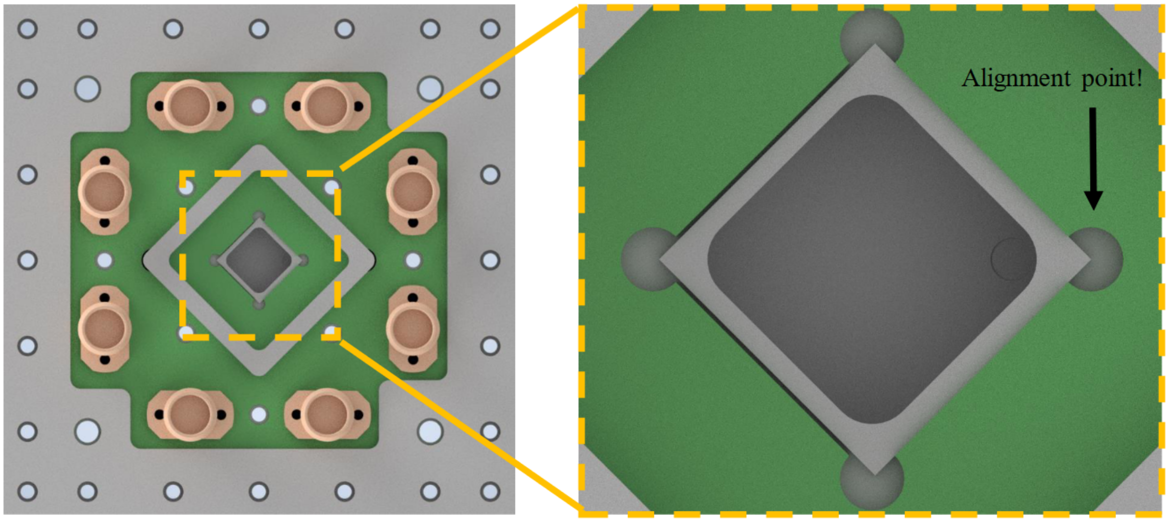
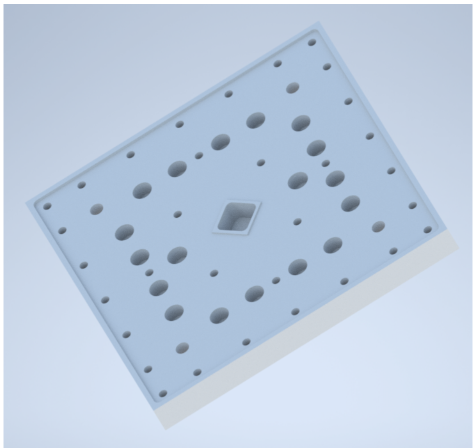
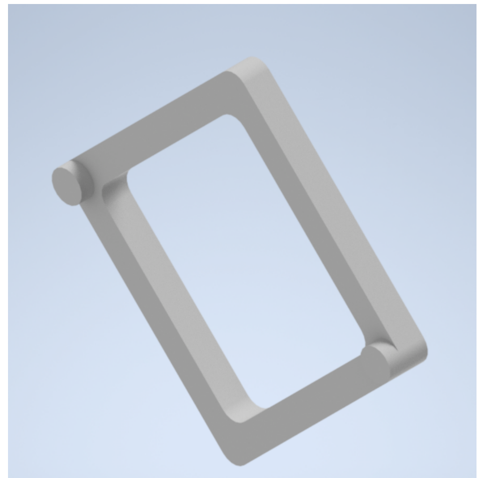


Figure 2: Alignment point of the PCB on the bottom copper part.

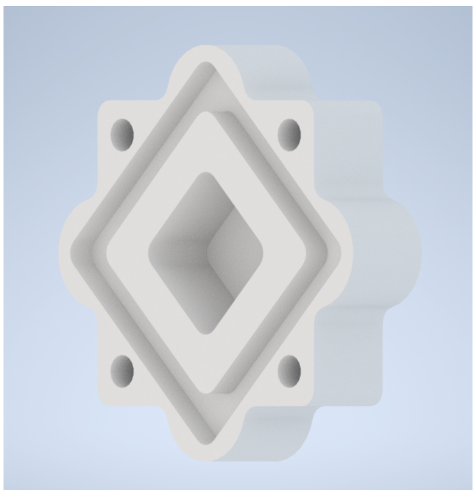
Around the area where the chip is placed, there is a copper ridge soldered onto the PCB. It is a feature that was added in the belief that it improves the shielding from high-frequency line-of-sight radiation, which might leak through a metal-to-metal contact. All the copper pieces used in the design can be seen in Figure 3. All the CAD files are provided in .STEP format.



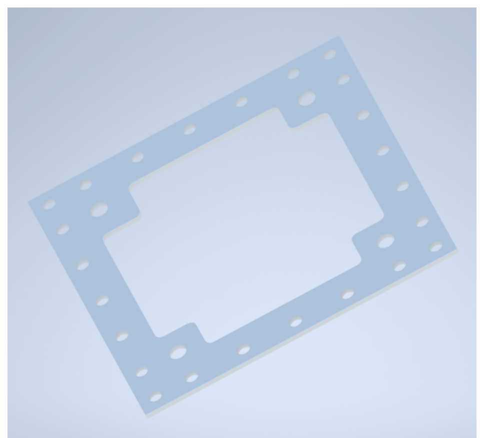
(a) The bottom copper piece containing screw-holes and drill-outs for fitting the screw-heads of the SMA connectors.



(b) The ridge which is soldered on to the PCB, having one guide-pin in two of the corners.



(c) The lid seen from the bottom, where a pocket for the ridge and chip can be seen.



(d) The top plate with holes for clamping down the PCB to the bottom copper piece.

Figure 3: All copper pieces in the sample holder design.

4 Assembly

In the following section, the steps needed in order to assemble the sample holder are described. The trickiest part is to solder the ridge on top of the PCB. This section describes a method which produced a successful result for us.

4.1 Soldering the copper ridge to the PCB

For this step you will need the following:

- A hot-plate to heat up the PCB and solder the ridge.

- Aluminum foil to cover the hot-plate if you want to avoid any potential stains from the soldering process.
- Two copper brackets to support the PCB while soldering. This creates a gap between the PCB and the hot-plate which makes it easy to remove when the soldering is done.
- A pair of tweezers.
- A PCB.
- A copper ridge.
- Solder paste with low melting point. We used the solder paste TS391LT from Chip Quik. The datasheet for this solder paste can be found in the shared files folder.

4.1.1 Step 1 - Apply solder to the ridge piece

In this step you should apply solder paste on the backside of the ridge. Here it is important to not put too much. A line approximately 1 mm wide should be enough, see Figure 4.

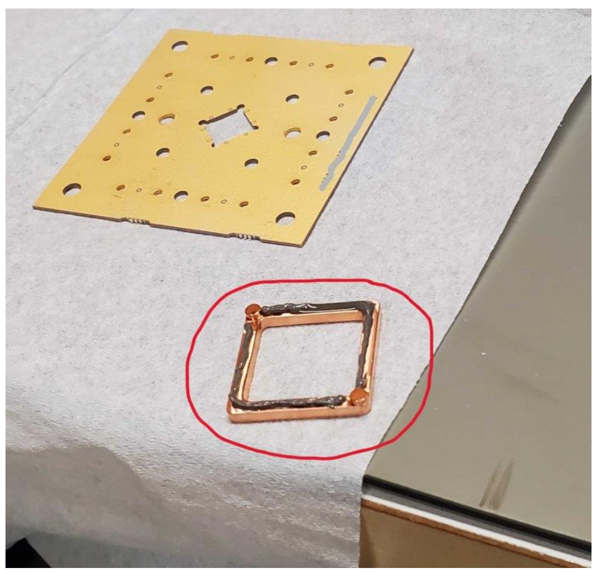


Figure 4: Less solder paste than this is might be better. But this amount also worked fine.

After this, place the ridge with the applied solder on top of the PCB (tweezers is recommended to use for this). Try to be as gentle as possible to prevent solder paste from ending up somewhere else on the PCB. If you are not happy with this step you can use IPA to clean off the solder paste and redo the above step.

4.1.2 Step 2 - Solder the ridge to the PCB with the hot-plate

- Set the hotplate temperature to 235°C.
- Place the PCB onto the copper brackets on the hot-plate with a tweezer, see Figure 5.
- Make sure you have some air vent to save yourself from soldering fumes.

- Now it's time to wait. It should not take long before the solder melts if the oven is hot from the beginning. Keep an eye on the copper ridge and you will see when the solder melts. Wait until you clearly see that the solder paste has melted all around the ridge. If you see smoke at this point, it's just the solder paste flux that burns and you don't have to worry.

- When the solder paste has melted, take the tweezers and place the PCB on some metallic piece where it can cool down. For the assembly in the picture, we used pieces of folded aluminum foil for the PCB to hover over the workbench, see Figure 6.

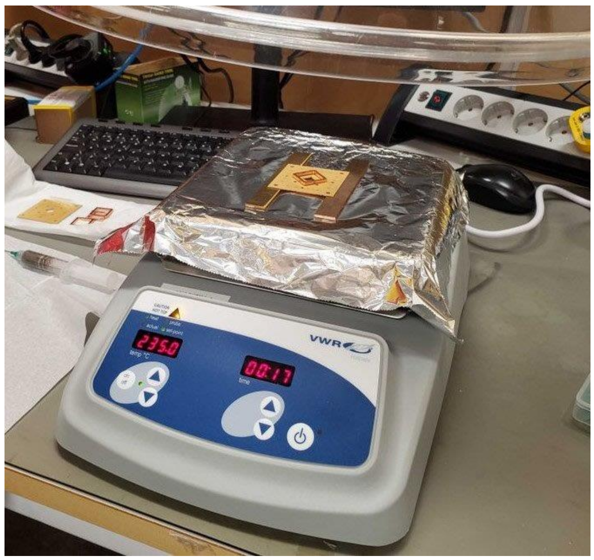


Figure 5: The PCB with ridge lying on top of two copper brackets placed on the hot-plate.

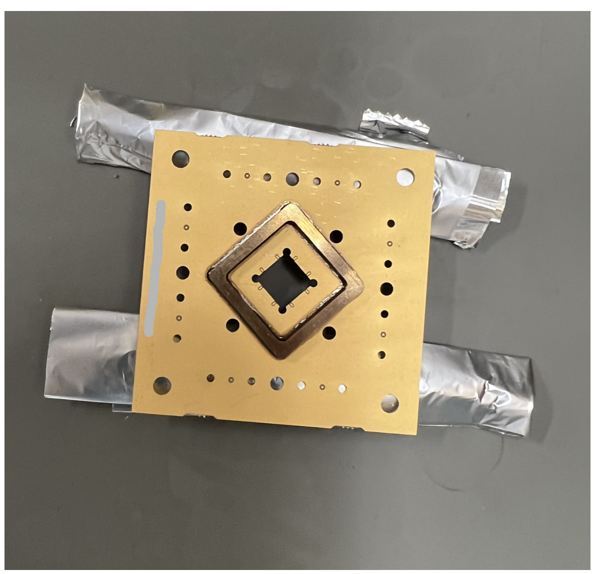


Figure 6: After the soldering was done, the PCB was placed on two pieces of folded aluminum foil.

4.2 Cleaning

Before assembling the rest of the sample holder, the solder flux residue needs to be removed. This can be done (and was done) with an IPA bath with ultrasound. A before and after picture of the cleaning can be seen in Figure 7.

(a) Before cleaning. (b) After cleaning.

Figure 7: Cleaning of the PCB post soldering.

4.3 Screw all the piecies together

4.3.1 Step 1 - Fix the SMA connectors to the PCB.

In this step you will attach the SMA connectors to the PCB. The connectors are mounted with screws that come in from the backside of the PCB, see Figure 8.

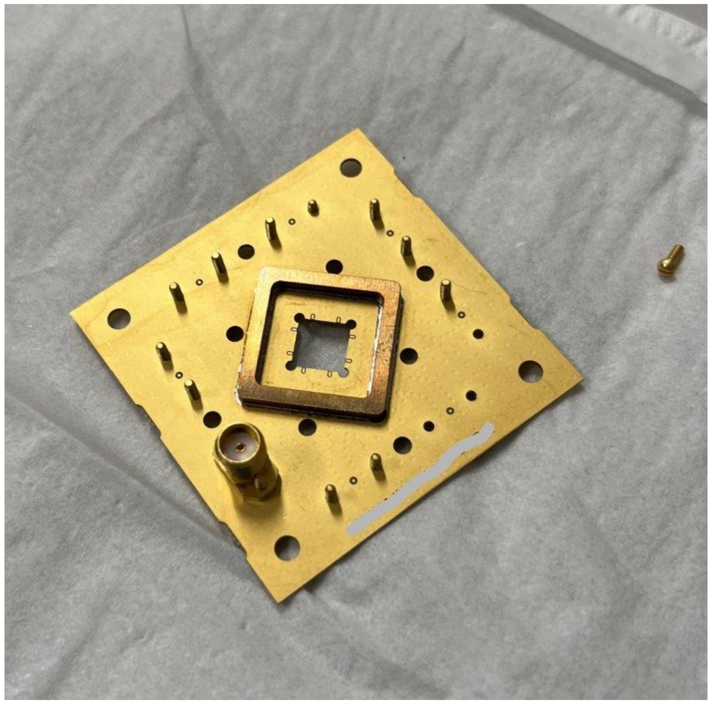


Figure 8: Attach the connectors to the PCB.

NOTE: The holes for the screws on the PCB is a bit to small for the screws due to the added thickness of the gold coating. To make them a bit larger, a tool similar to the one in Figure 9 can be used (which fixes the problem). Just a slight adjustment is needed.

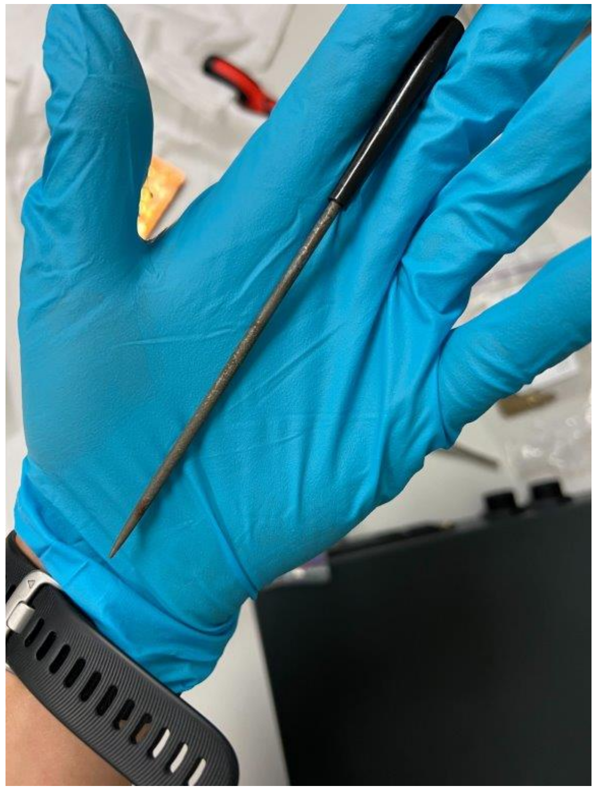
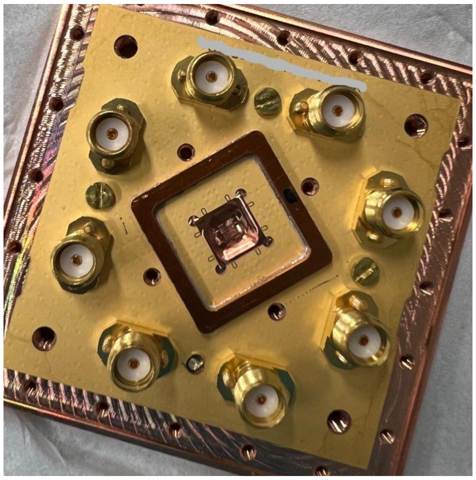


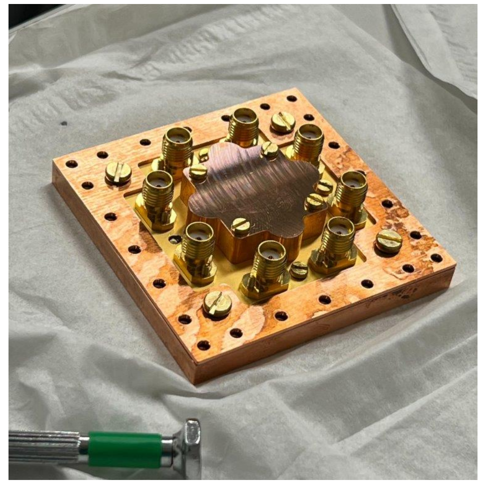
Figure 9: Tool for widening the holes for the SMA screws to fit in the PCB.

4.3.2 Step 2 - Attach the rest of the parts

The last step of the assembly is to mount the PCB to all the copper parts. This is done by simply screwing everything together. Start to mount the PCB on the bottom copper part and then attach the top plate, see Figure 10.



(a) PCB with ridge and connectors attached to the bottom copper piece.



(b) The fully assembled sample holder.

Figure 10: Final assembly step of the sample holder.

The sample holder is now assembled and ready to be used.

5 Terms of usage

This design is shared under the CERN-OHL-S v2 license [1]. Please refer to and read the license text before using or further working on the design.

References

[1] *CERN-OHL-S v. 2.* [Online]. Available: `https://ohwr.org/cern_ohl_s_v2.txt`.

[2] *COMSOL Multiphysics ® v. 6.1* COMSOL AB, Stockholm, Sweden. [Online]. Available: `www.comsol.com`.

[3] *KiCad EDA v. 6.0*, KiCad. [Online]. Available: `www.kicad.org`.