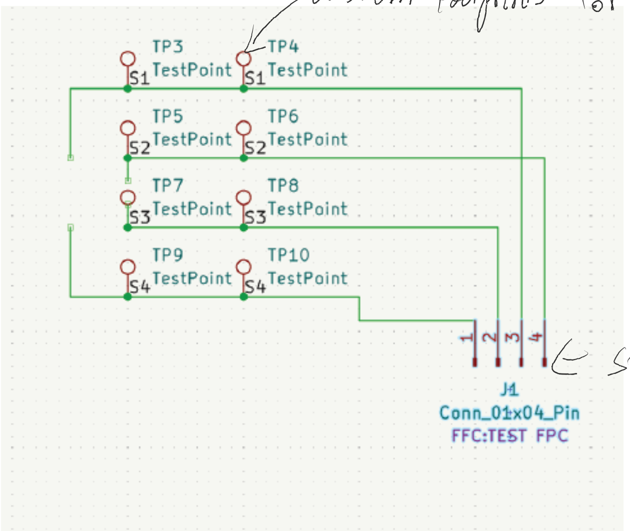
PCB design

custom footprints for the vias



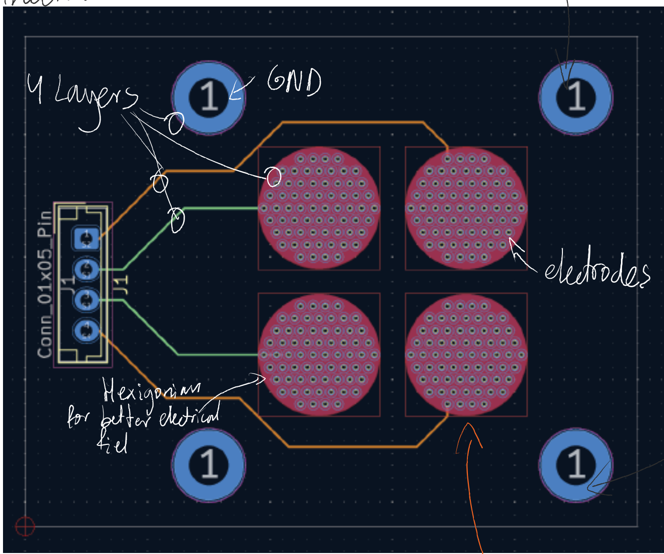
↳ pitch 1 mm ∅0.3 mm

standard connector

To be able to clamp the pump

4 Layers

GND



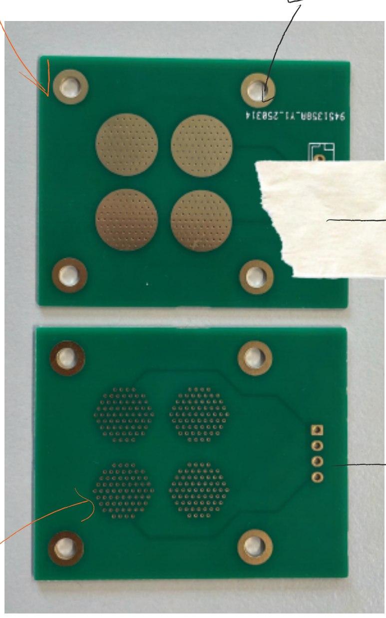
electrodes

Hexigonium for better electrical field

3M

surprised its very thin

ENIG finish



Can we automate? kiCAD API?

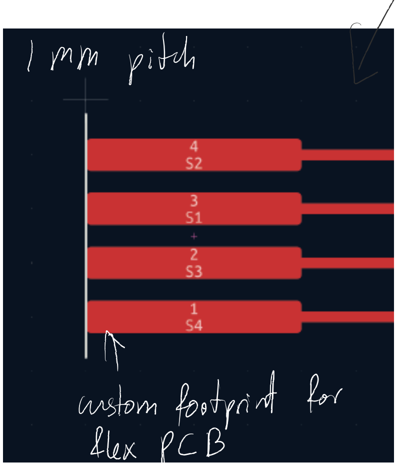
→ put to glass fiber filter

→ upper side

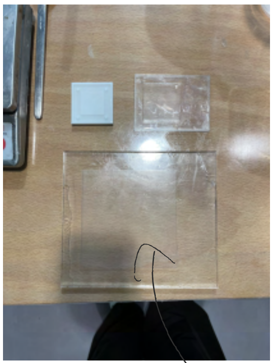
PCB shape is determining actuation for

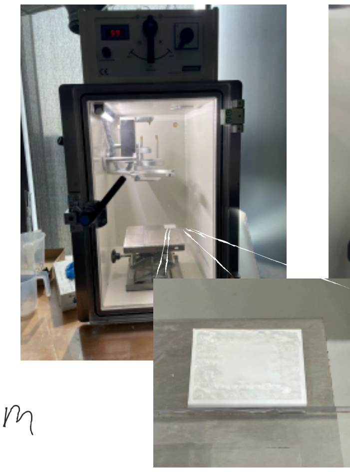
flex can only be 2 Layers

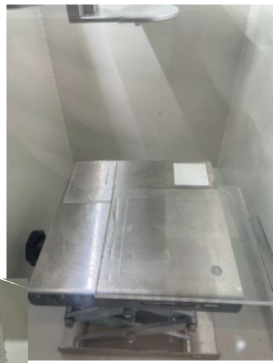
1 mm pitch



custom footprint for flex PCB





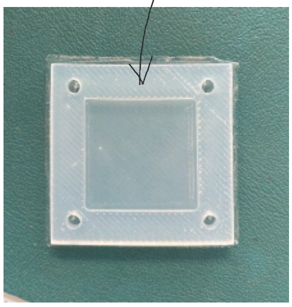


Thicker materials
more air bubbles

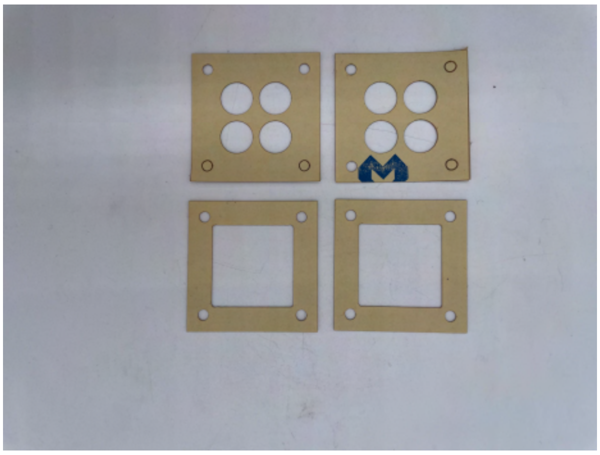
0,2 mm

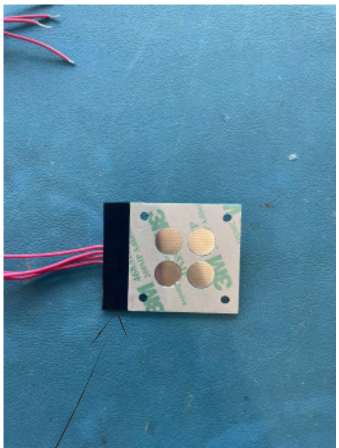
Too thin

Silicone
Reservoir



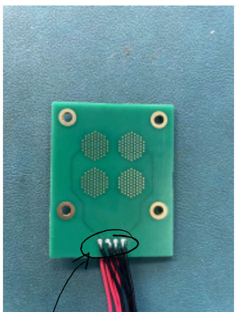
Lasercut

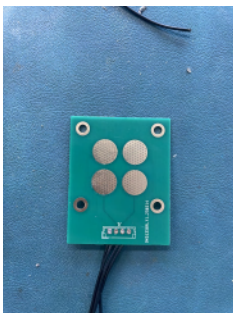


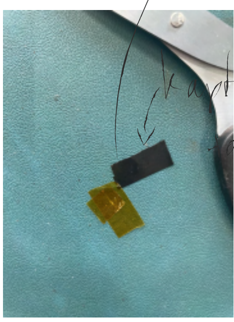


used
tweezers

Tape as
adhesive & spacer



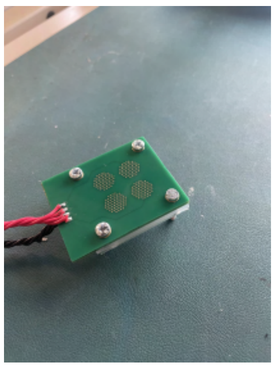


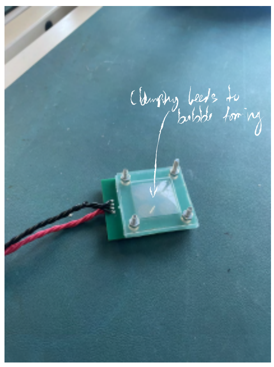


Kapton and insulating tape
put between the two PCBs

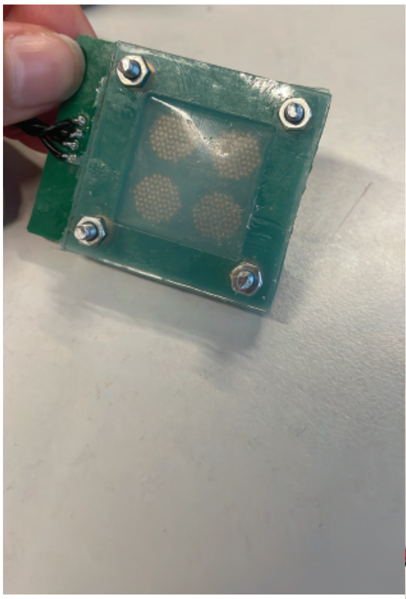
GND and V very
close to each other

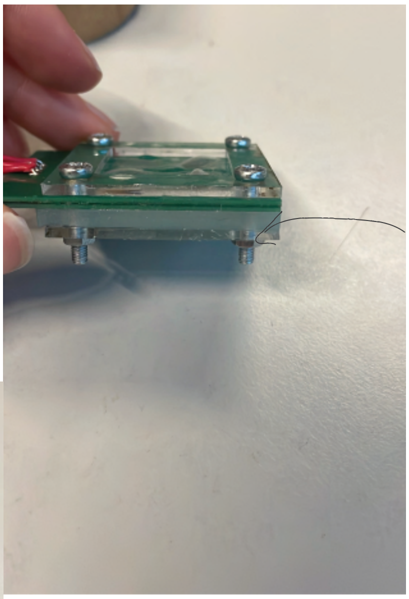
⇒ use pads
for PCB 2



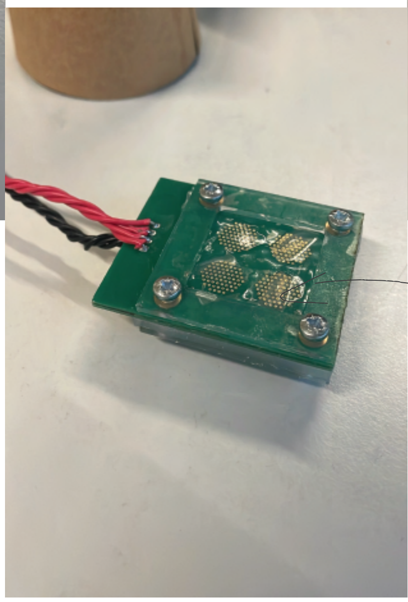


Version 1

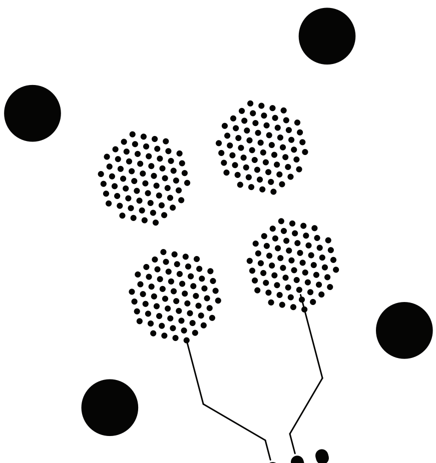


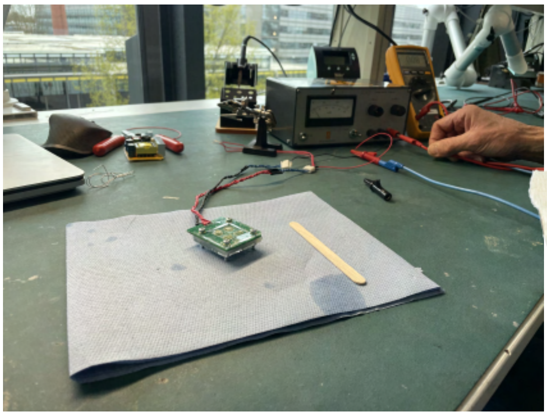


clamping adds height

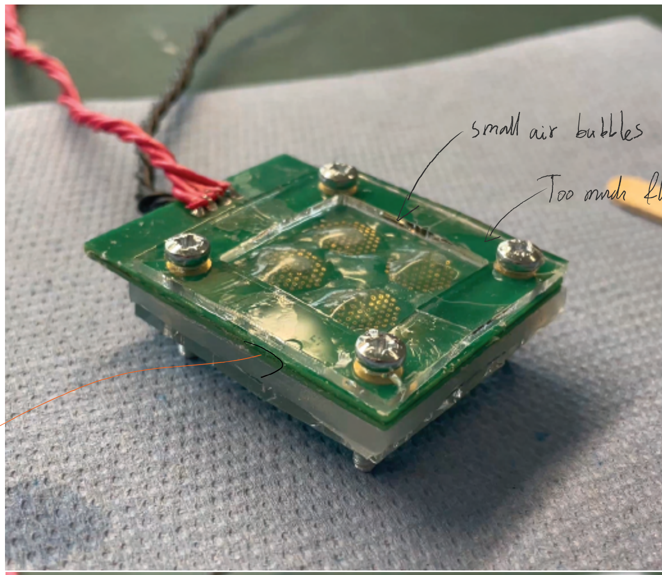


filled up Too much



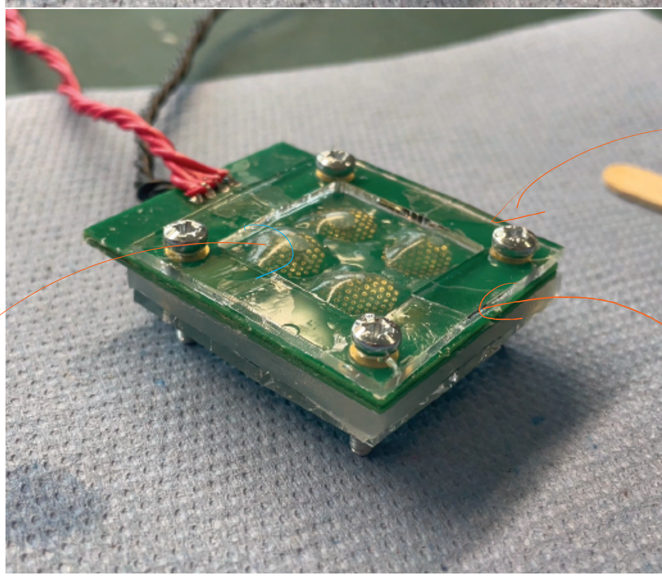


150 V
1.2 mA
↓
180 mW



small air bubbles
Too much fluid

Some Leaking



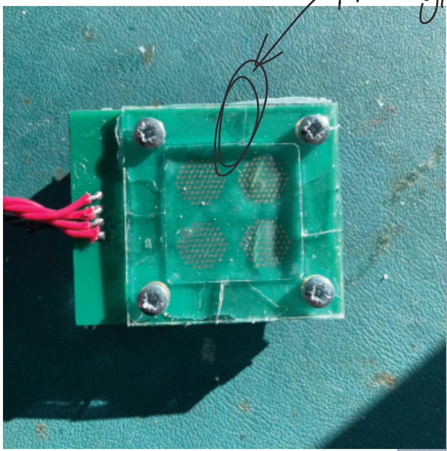
Relatively quick

Satisfaction

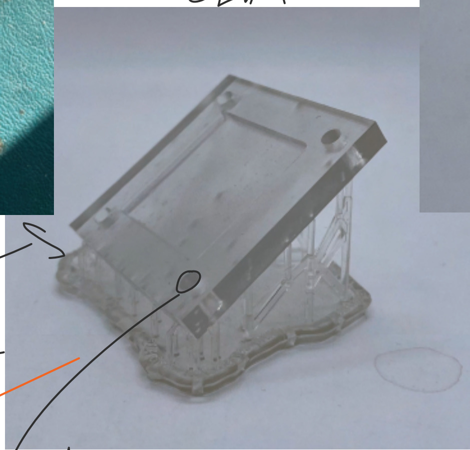
no noise or heat

Material properties

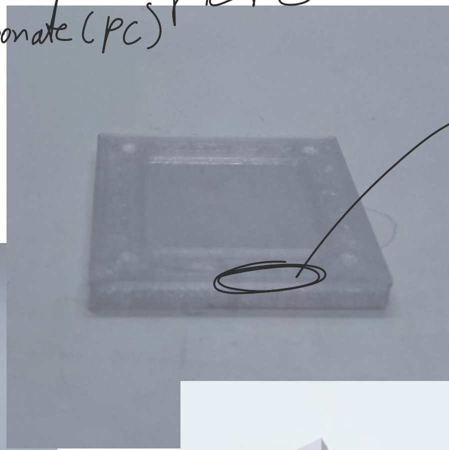
plexiglass internally cracks
due to the propylene carbonate (PC)



SLA



PETG



+ chemically compatible
- Low resolution
Not air tight

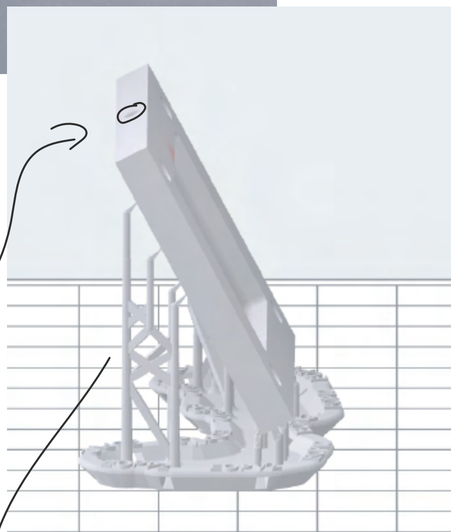
Transparent so
air bubbles are visible

Materials
reject each other

High resolution, chemically compatible
But hard to fill the EOPs

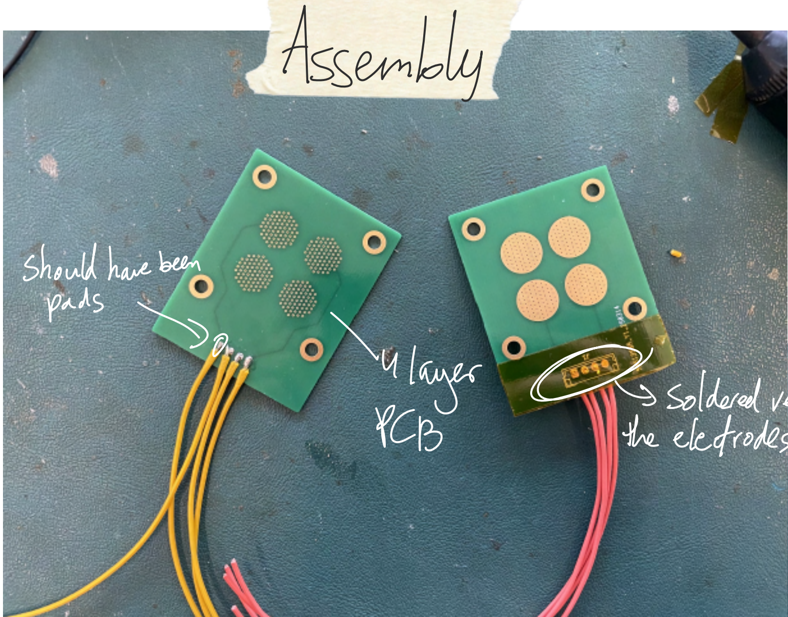
I went back to
ecoflex silicone as
its flexibility allows me to
inject easily and let air out

added an
injection hole



Formlabs slicer automatically
generates supports. Took an
afternoon to clean the formlabs
in the lab and put the print on

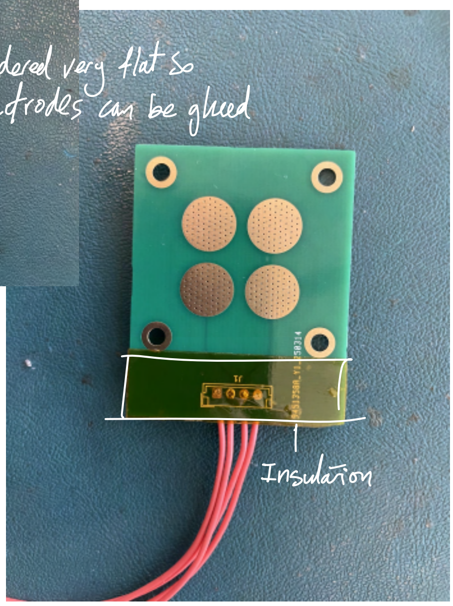
Assembly



Should have been
pads

4 layer
PCB

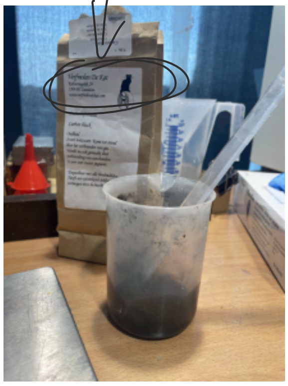
Soldered very flat so
the electrodes can be glued

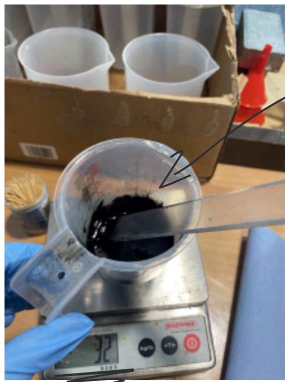


Insulation

Glueing 0,5mm perspex on plexiglass to make thin membranes
↳ 0,4–0,5mm thick

Adding carbon black powder to the Usilicone.

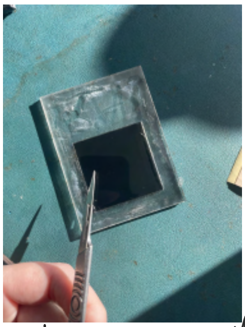




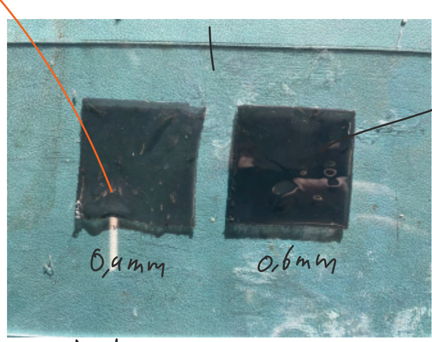
Gets pitch black

about 1gram added black powder

Feel very flimsy and sticky



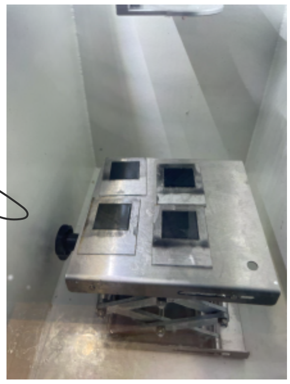
Get out of the mold with a scalpel for clean edges

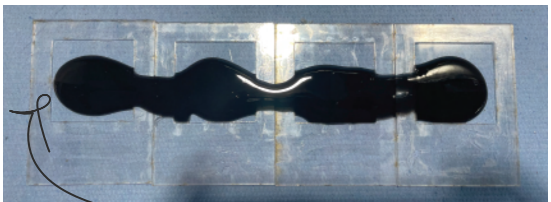


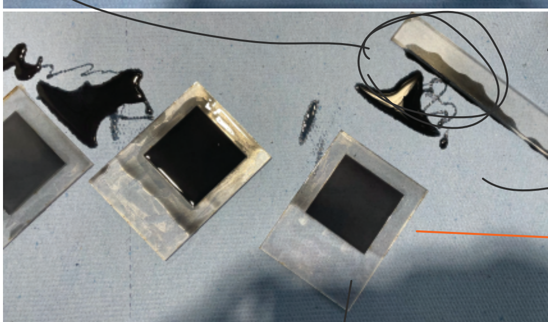
* thickness changes to "blackness"
↳ can never achieve pitch black as its so thin

Ecoflex 30 for very soft and flexible top layers

one air bubble is fatal so a vacuum-chamber is used







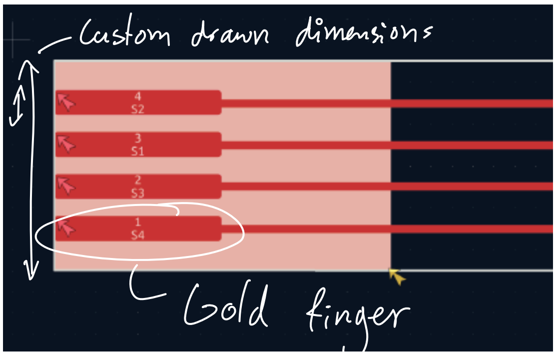
swipe excess off

very hard to get full control over material thickness

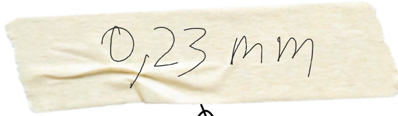
↳ Cures in about 2 hours

Flex PCB design

1mm pitch



← edge cuts
← copper traces

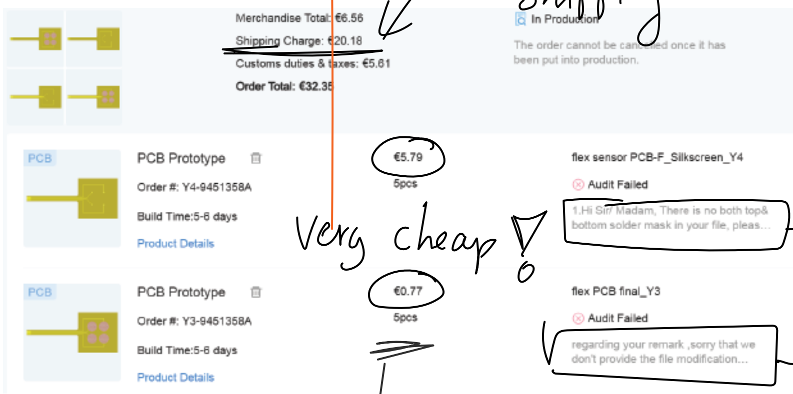


very thin

Dimensions based on datasheet
* must be very accurate as you want a secure connection with the connectors
* No wiggle room
  ↳ even more important with High voltages

PCBs have become more accessible
→ Can now be part of an iterative design process

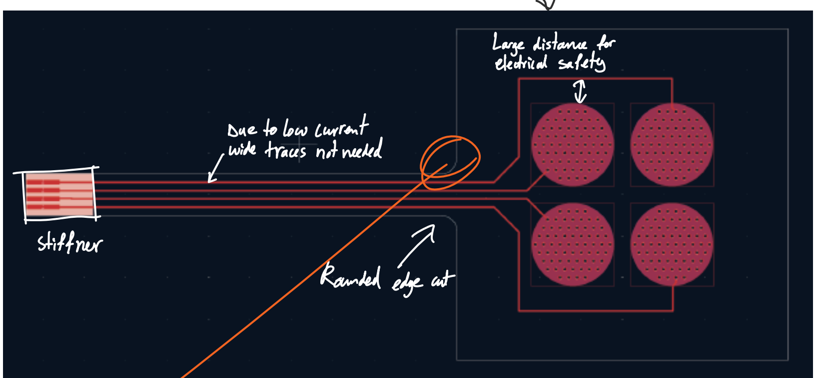
You mainly pay for shipping



very cheap !

Took two days of emailing the manufacturer on how to set the masking layers and exposed copper zones

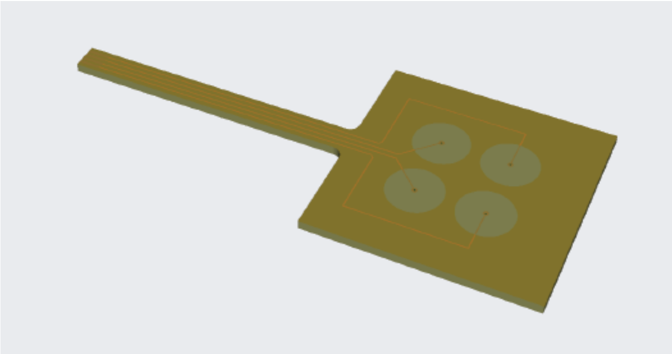
when working with high voltages the distance is the most important !



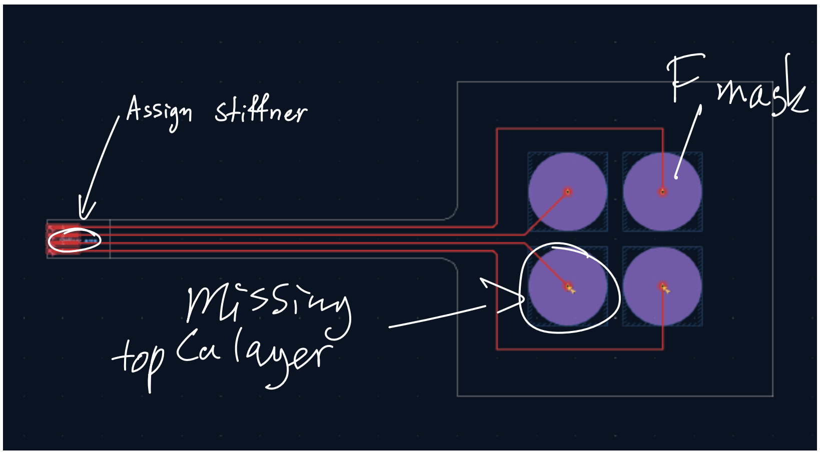
As KiCAD is not a drawing software it was hard to create rounded corners

you always order 5 PCBs as minimum

Sensing



3D gerber viewer



Wrong masking layers

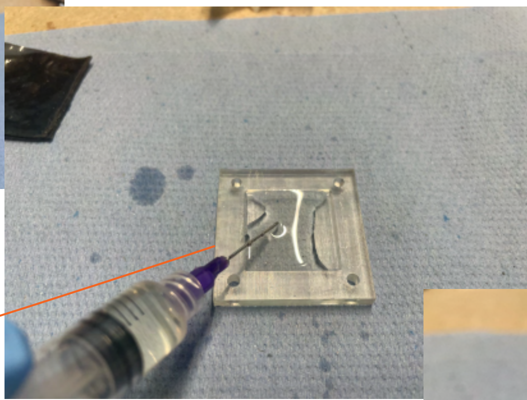
Mistakes happen

led to failed sensing PCB

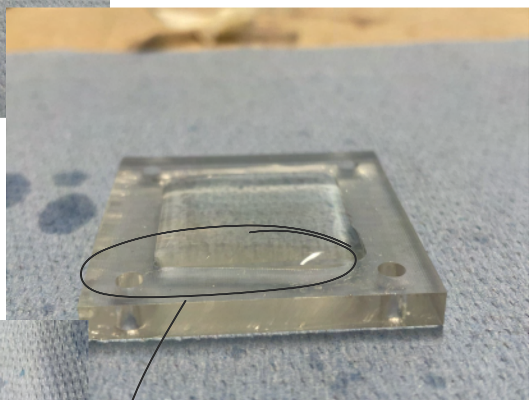
Enough exposed copper to be used as a sensor -> you get 5 PCBs so you can use 2 for the pump and 2 for sensing

Took 4 weeks to obtain propylene carbonate
↳ May only be ordered via a lab or company
↳ inaccessable for individual makers

Filling EOP

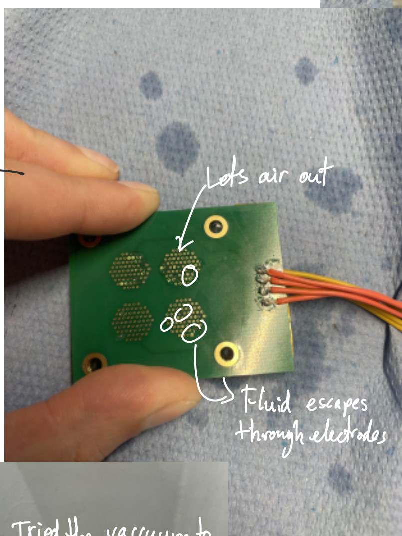


Trasporacy serves a functional purpose

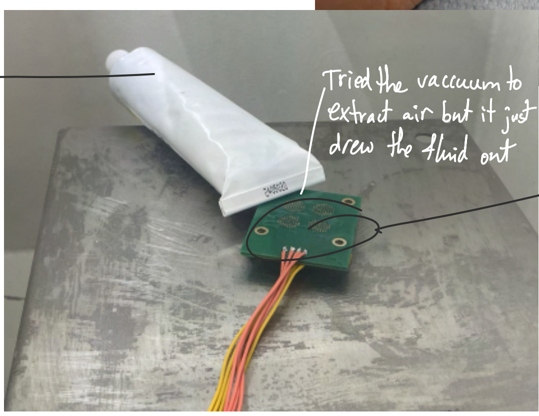


Fill until just over the edge

Press PCB firmly on

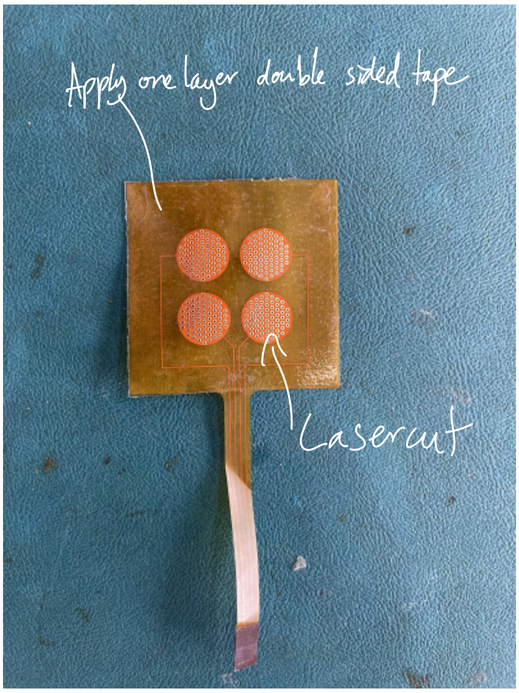


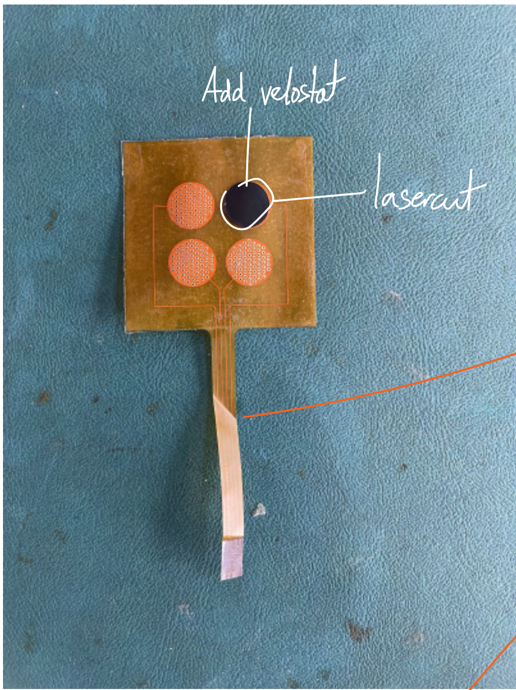
Silpoxy



Very messy assembly method
Top part gets wet → hard to glue
lots of air gets in while glueing
↳ now use silicone reservoirs
- easy to inject
- keep air out

Sensing





Irreversible denting from manufacturer

Enabling embodied reciprocal interaction

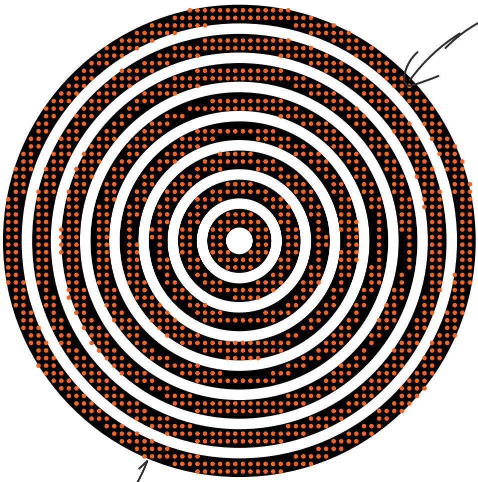
kapton layers give orange effect

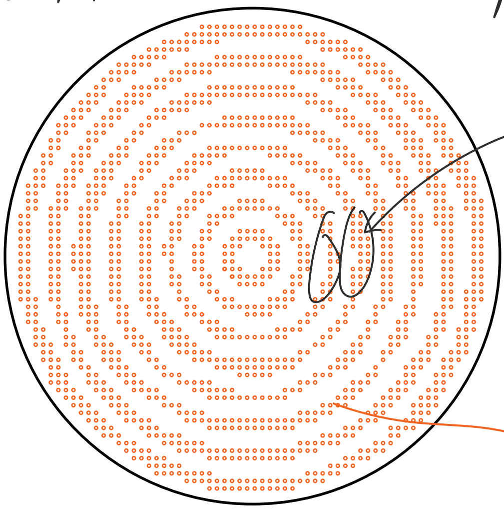
larger velostat would have been better

↳ lasercut tape was the issue
2nd try used Z-axis conductive
Connecting tape
↳ no 4 holes just one sheet

Shape exploration electrodes

Mainly drew flowy and round forms → complex shap



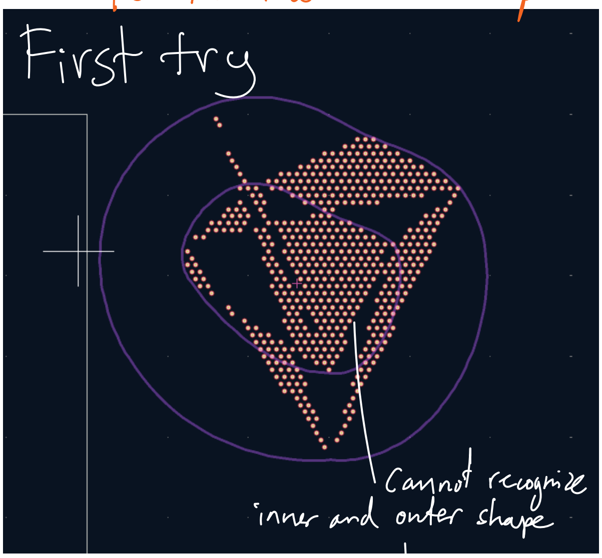


irregular
↓
Bad for electric field

Shape explorations in illustrator before moving to KiCAD

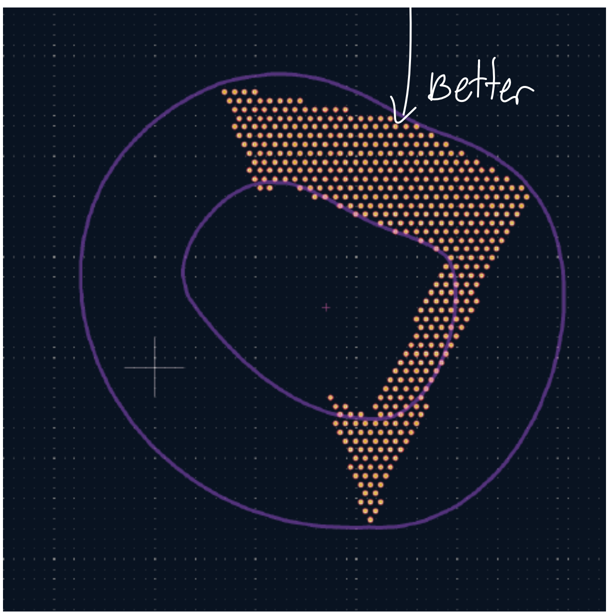
Drawing electrode designs in Illustrator but manual placement took way too long

— To be able to make expressive novel shapes




```
def pip(x, y, pts):
    inside = False
    for i in range(len(pts) - 1):
        x1, y1 = pts[i].x, pts[i].y
        x2, y2 = pts[i+1].x, pts[i+1].y
        if (y1 > y) != (y2 > y):
            xin = (x2 - x1) * (y - y1) / (
(y2 - y1) + 1e-9 ) + x1
            if x < xin:
                inside = not inside
    return inside
```

check if a point is inside a polygon




```
for gi in shape_fp.GraphicalItems():
    if gi.GetLayer() == pcbnew.F_Cu:
        try:
            ps = gi.GetPolyShape()
            for i in range(ps.Outline-
Count()):
                ...
                loops.append(pts)
```

Collect all polygone outlines
↳ if not closed → forced shut

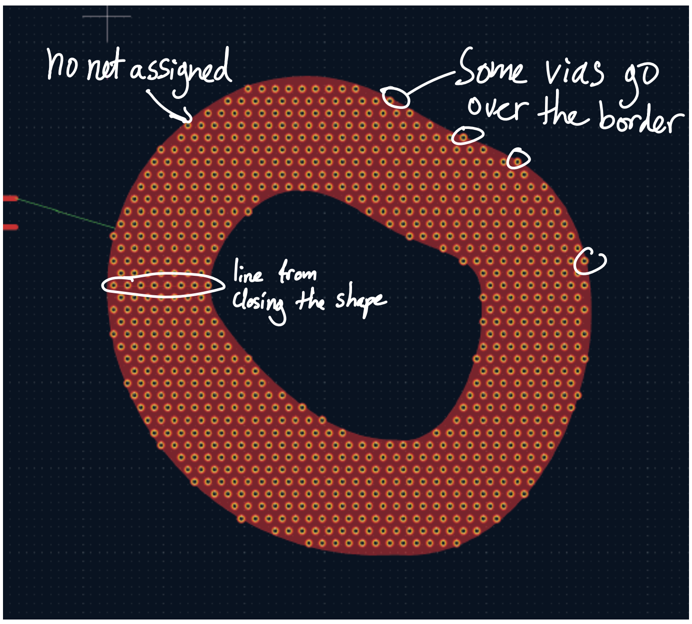
```
def seg_dist(x, y, x1, y1, x2, y2):
    dx, dy = x2 - x1, y2 - y1
    if dx == 0 and dy == 0:
        return math.hypot(x - x1, y - y1)
    t = ((x - x1)*dx + (y - y1)*dy) /
(dx*dx + dy*dy)
    t = max(0, min(1, t))
    px, py = x1 + t*dx, y1 + t*dy
    return math.hypot(x - px, y - py)
```

Calculate shortest distance from a point to a polygon edge
↳ to create a copper border

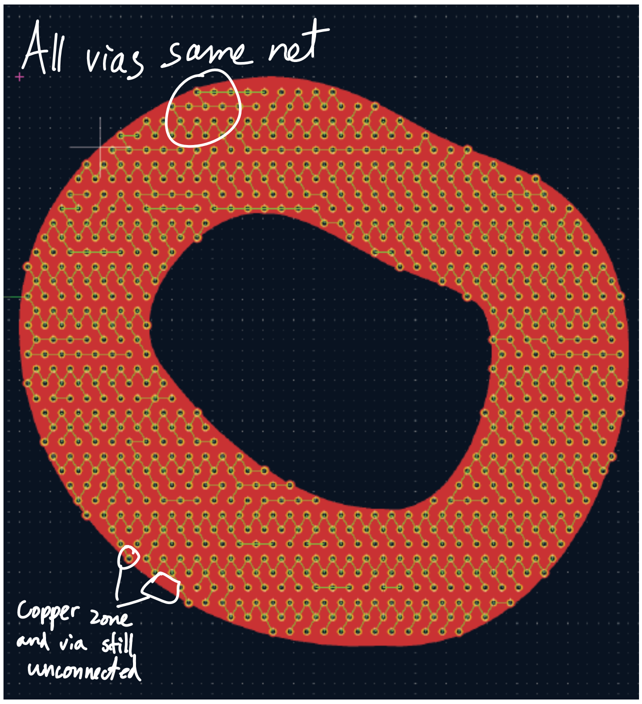
```
# 2a) Place vias (must be ≥border from any edge)
        placed = 0
        y = ymin; row = 0
        while y <= ymax:
            xoff = 0 if (row % 2) == 0 else pitch/2
            x = xmin + xoff
            while x <= xmax:
                if pip(x, y, pts):
                    dmin = min(
                        seg_dist(x, y, pts[i].x, pts[i].y, pts[i+1].x, pts[i+1].y)
                        for i in range(len(pts)-1)
                    )
                    if dmin >= border:
                        via = tp_tpl.Duplicate()
                        via.SetPosition(pcbnew.VECTOR2I(int(x), int(y)))
                        # **Only** override net; leave pad layer set alone!
                        for pad in via.Pads():
                            pad.SetNetCode(netcode)
                        board.Add(via)
                        placed += 1
                        total_vias += 1
                x += pitch
            y += dy; row += 1

        print(f"• Loop {idx+1}→{netname}: placed {placed} vias")
```

makes hexigonial grid

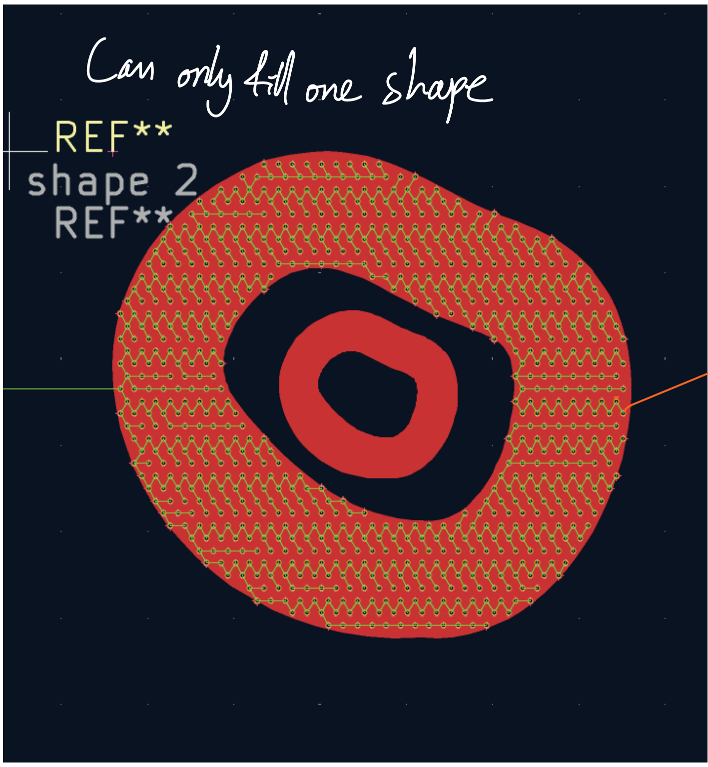
only place via if not too close to the edge



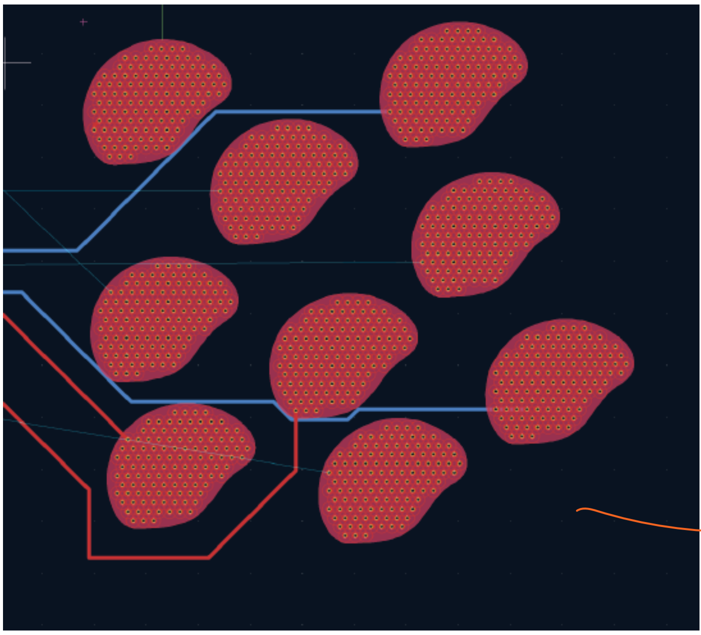
```
  # 2b) Create the F.Cu zone (outline
only; you'll press B to fill)
        zone = pcbnew.ZONE(board)
        zone.SetLayer(pcbnew.F_Cu)
        zone.SetNetCode(netcode)
        zone.SetIsFilled(True)
        zone.SetPadConnection(pcbnew.-
ZONE_CONNECTION_FULL)
        zone.SetLocalClearance(0)

        chain = pcb-
new.SHAPE_LINE_CHAIN()
        for p in pts:
            chain.Append(p)
        zone.Outline().AddOut-
line(chain)
        board.Add(zone)
```



Coding improves reproducability of the work



AI tools have democratized coding → allowing also designers to code more advanced



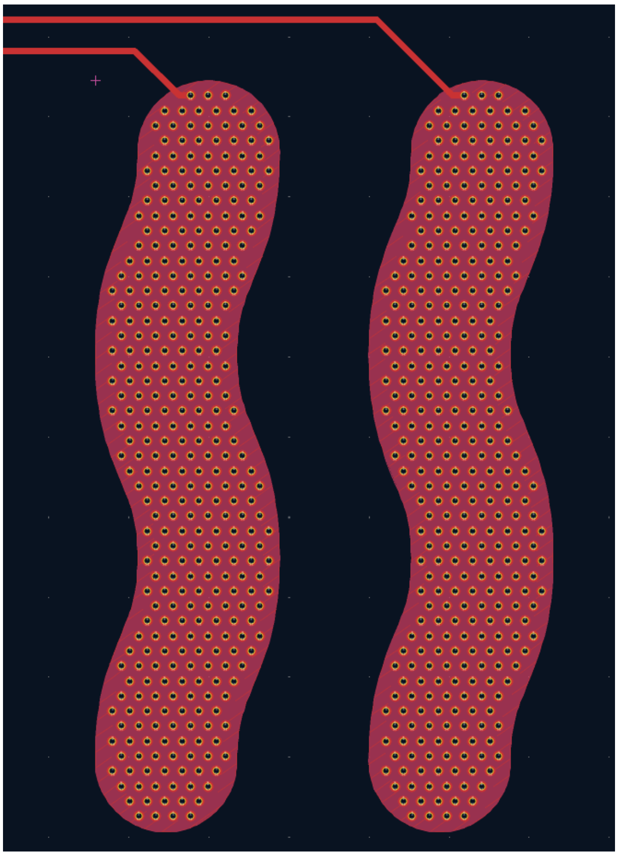
Give each shape an automatic different net

```
# Build list of net names: ["/S1","/S2",…]
NET_NAMES = [f"/S{i+1}" for i in range(MAX_NETS)]
```

Automation -> improves manufacturability

```
exec(open(r'C:\Users\tessa\OneDrive\Documents\ID thesis\first PCB\Scripts\script.py').read())
• Loop 1→/S1: placed 1015 vias
• Loop 2→/S2: placed 792 vias
• Loop 3→/S3: placed 548 vias
• Loop 4→/S4: placed 329 vias
• Loop 5→/S5: placed 130 vias

□ Done. 2814 vias placed; zones created—press B to fill.
```

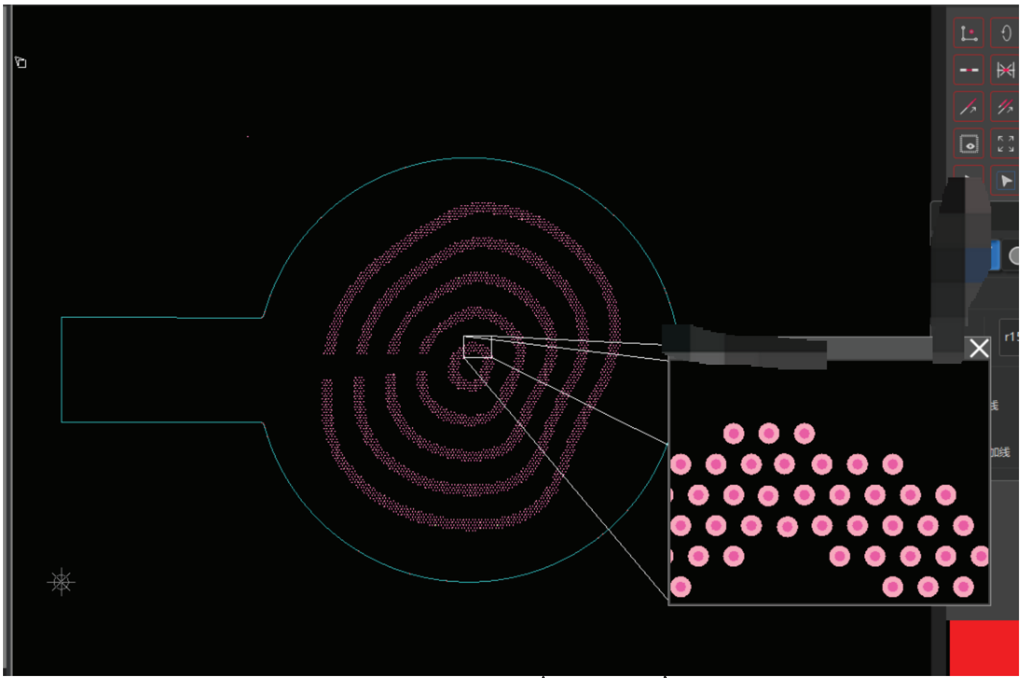
output gives how many vias placed



Defining a new copper zone based on the input SVG finally allowed me to connect the vias to the zone with correct masking for exposed electrode designs.

```
# === USER CONFIGURATION ===
SHAPE_FP_NAME = "W3"      # footprint containing multiple copper loops
TP_FP_NAME    = "TP 0.15mm"    # your through-hole testpoint footprint
MAX_NETS      = 16             # supports nets /S1 … /S12
BORDER_MM     = 0.3            # required copper annulus width
PITCH_MM      = 0.55           # hex-grid pitch for vias
```

Put SVG shape, via footprint, amount of order and pitch



Going back and forth with the manufacturer on what is physically possible with their machines → the scripted helped in iterating fast → Final specs

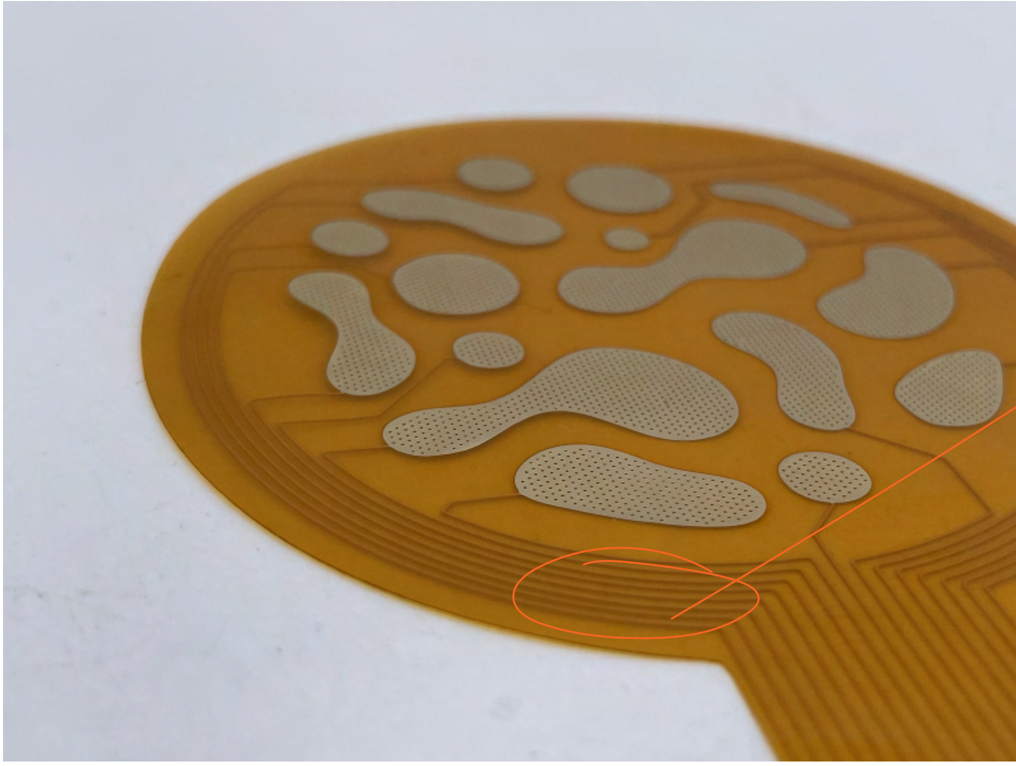
Minimum via hole size (flex PCB)	0.15 mm
Minimum via pad size (flex PCB)	0.35 mm
Minimum annular ring	0.10 mm
Minimum hole-to-hole clearance (same net)	0.13 mm
Required copper clearance between testpoints	≥ 0.20 mm

minimum pitch = 0.35 mm (pad diameter) + 0.2 mm (required clearance)
= 0.55 mm

Manufacturer constrains + costs limit the possibilities

had to keep it within 10x10 cm

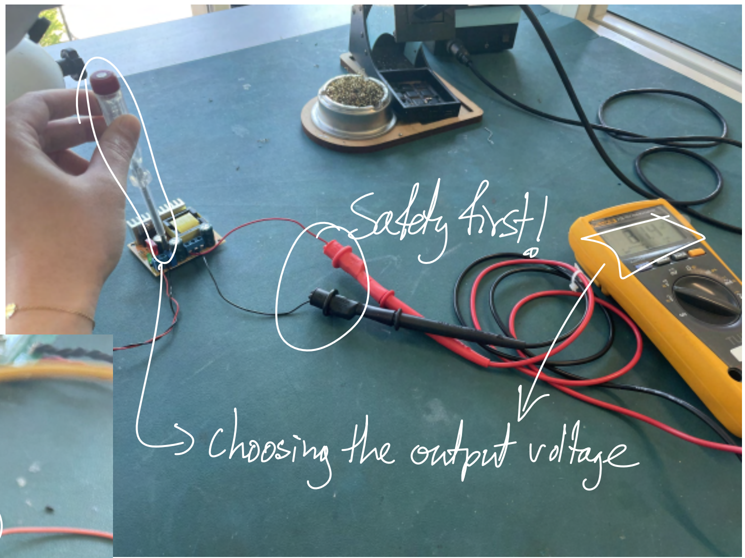
The final electrodes

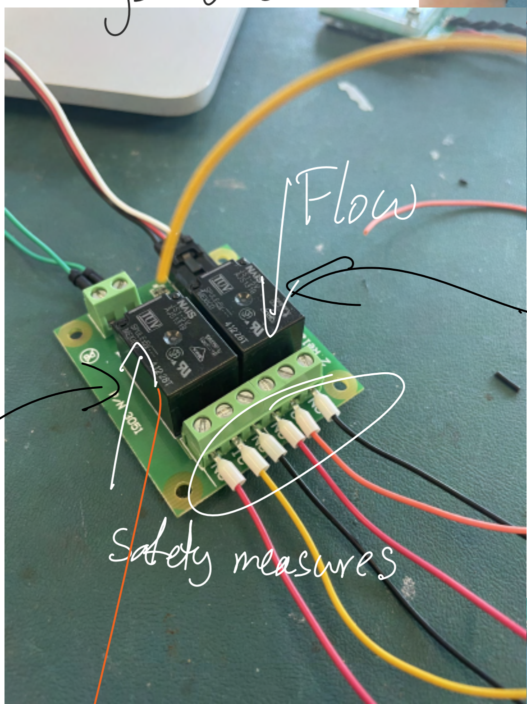


As kicad is not drawing software these traces took 2 days to make → was needed for aesthetic and safety purposes

Control

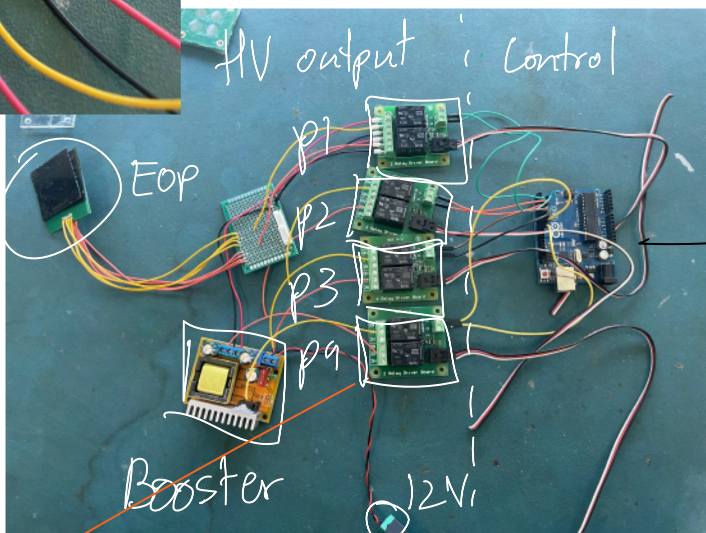
Regular electronic components cannot handle 200V so mechanical relays were used





suck fluid down

pump fluid up



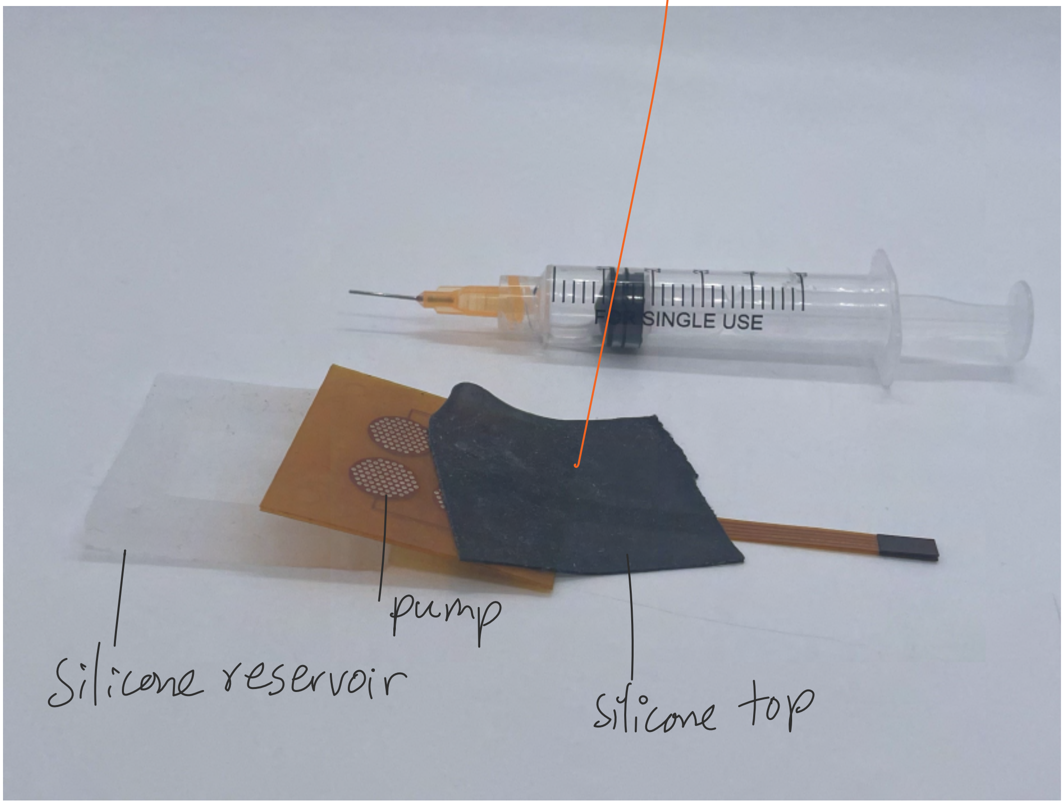
5V switching Signals

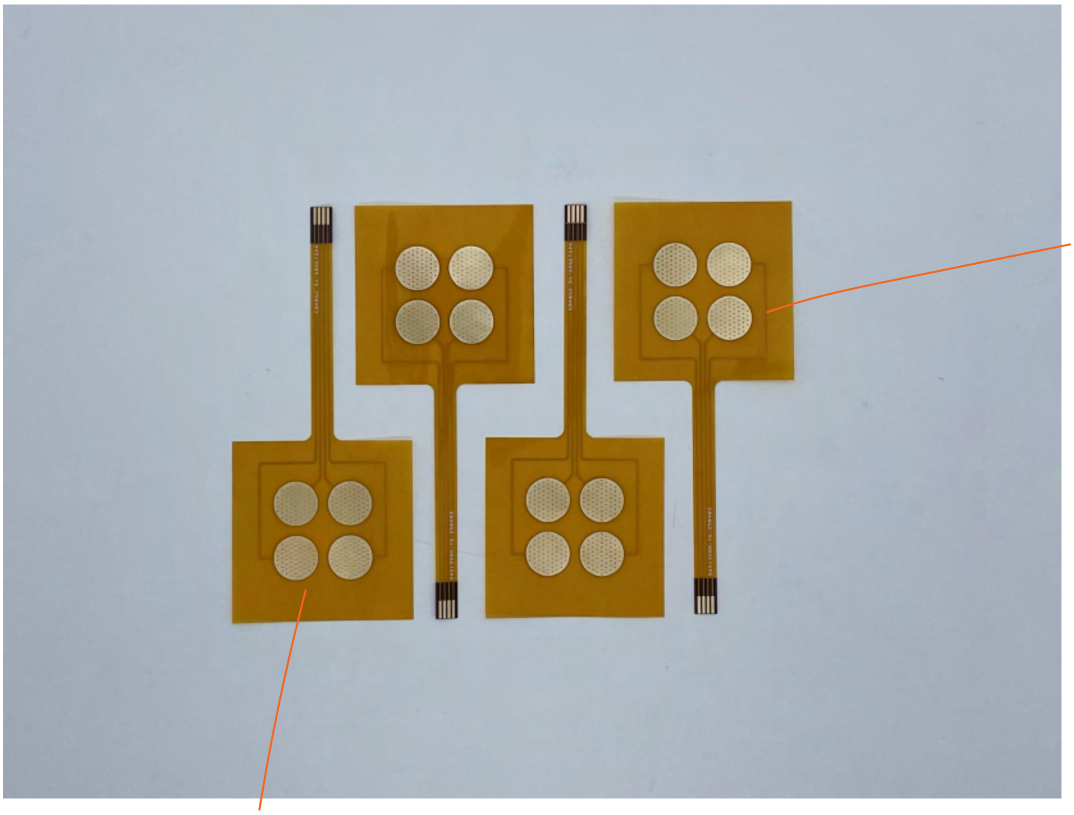
using these based on availability for quick results

Can control 4 pumps up and down

4 channels to show expressivity and movement

baby powder to resolve the sticky feel

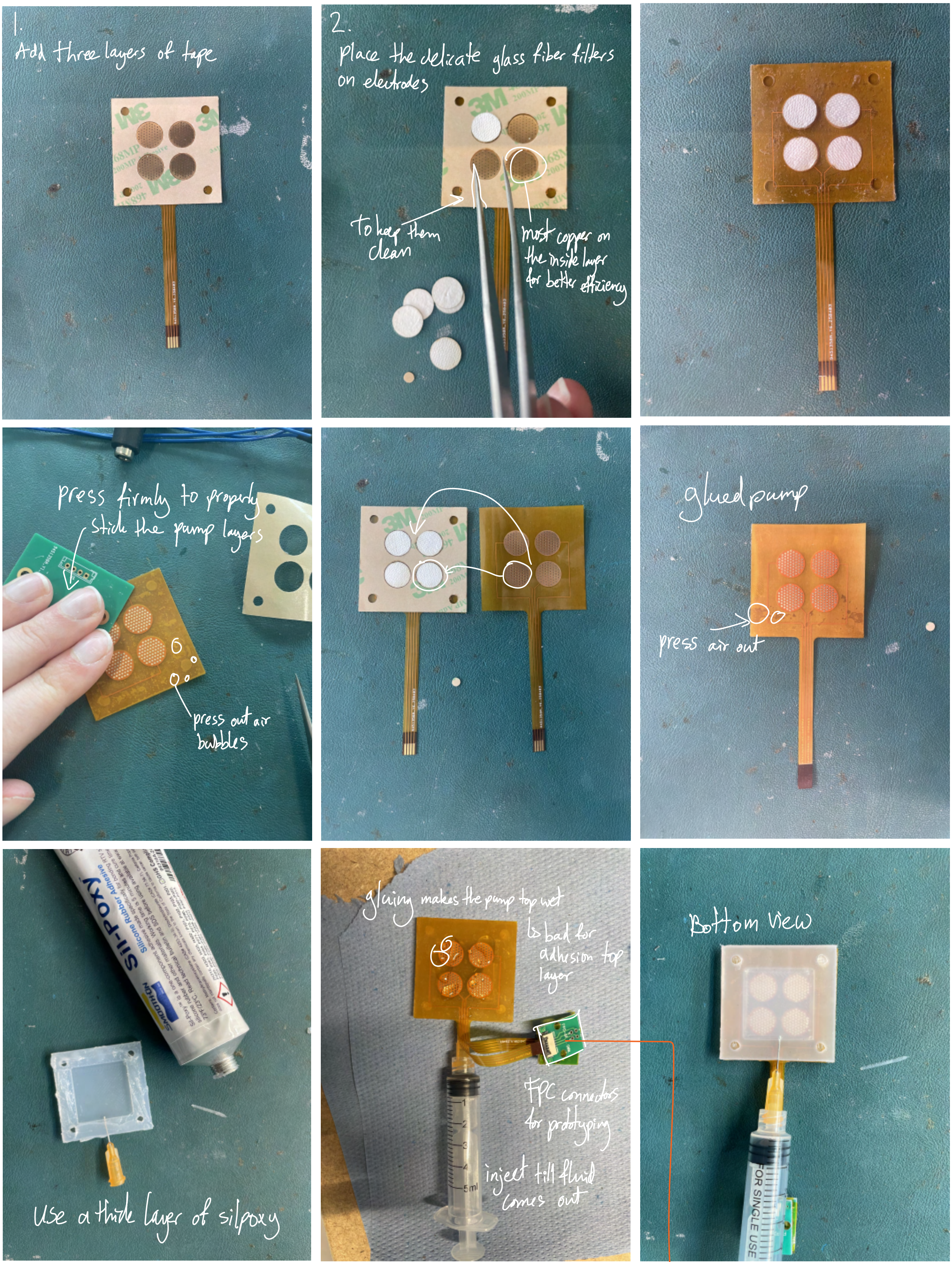




Transparacy in design

The kapton and gold give aesthetic qualities

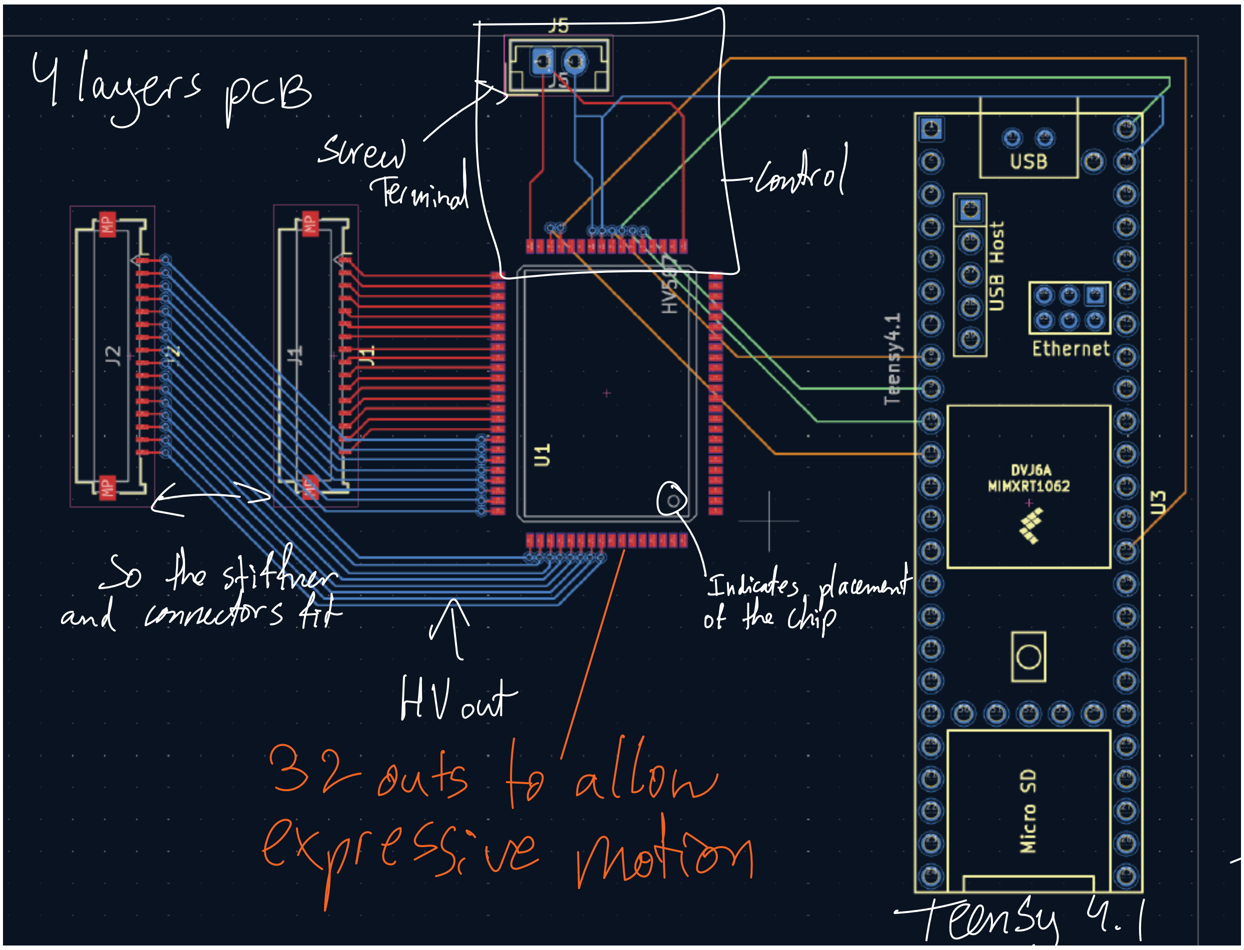
Flex EOP assembly

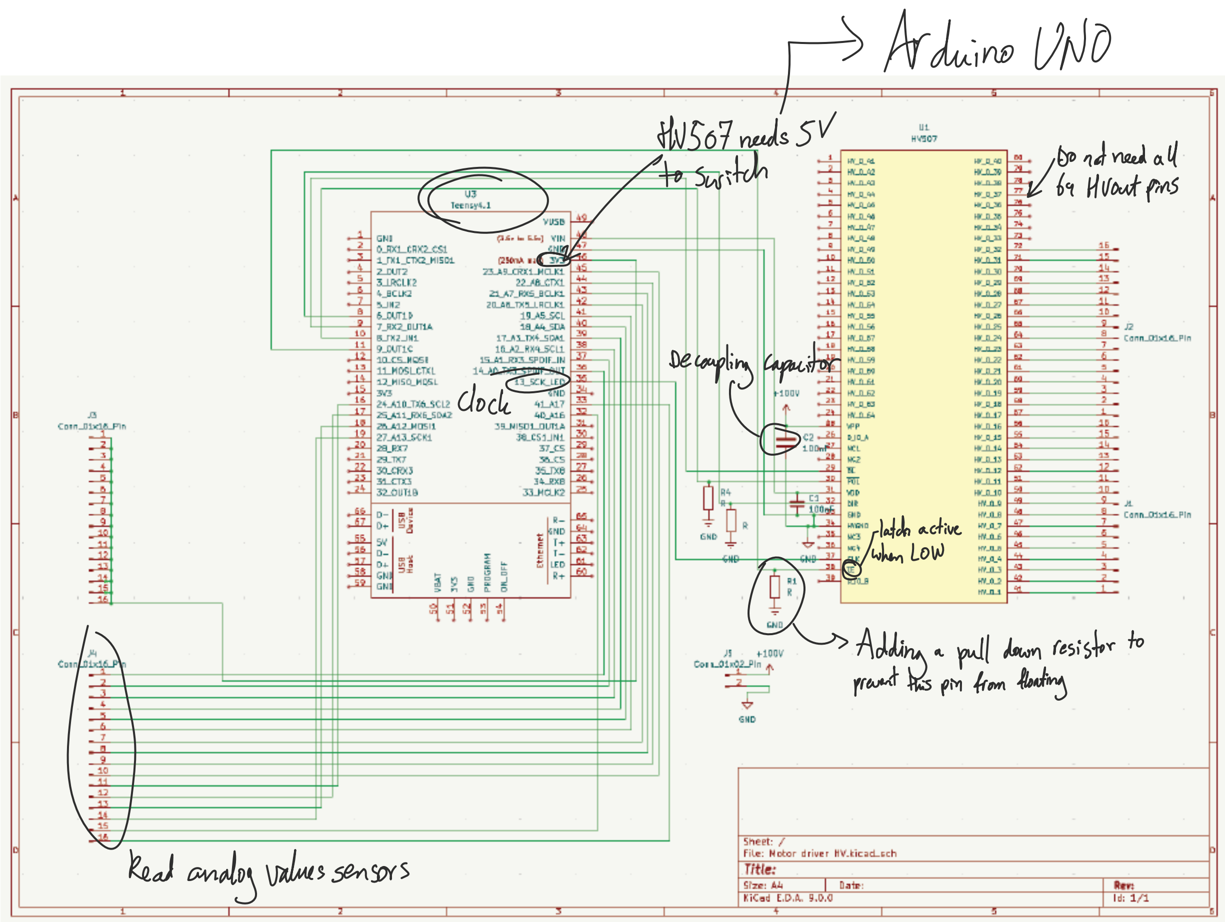


Full assembly takes about an hour

low vs high fidelity electronics are needed per design phase

PCB design

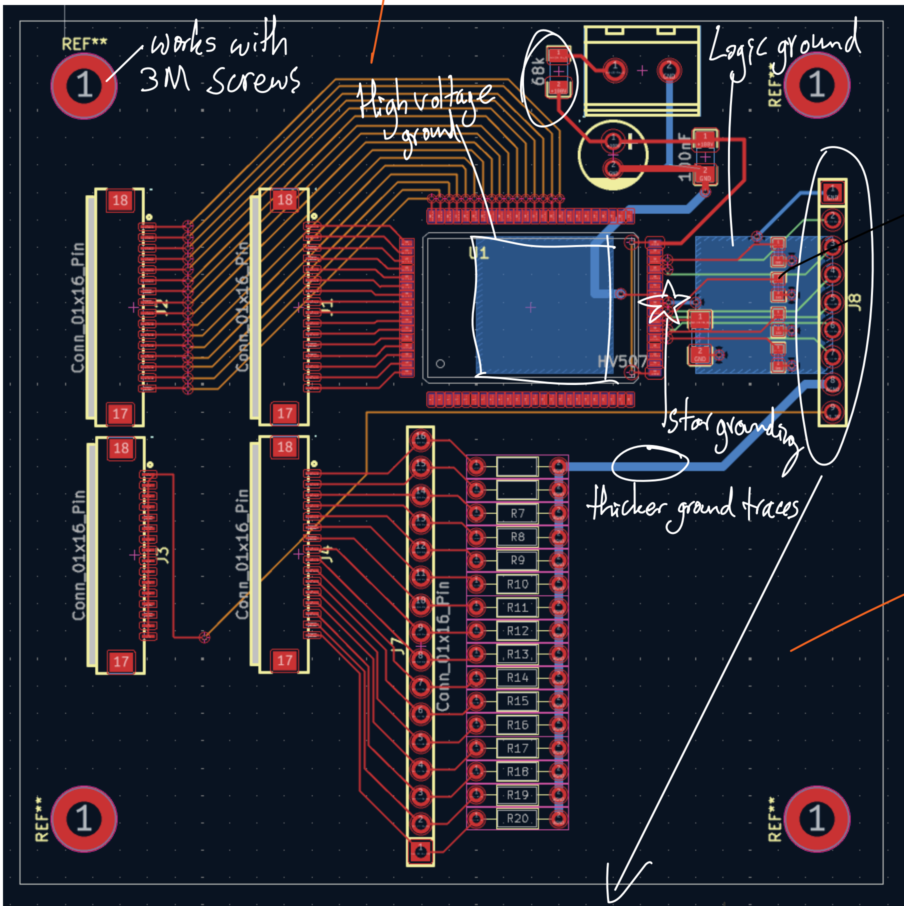




Familiarizing with the HV chip

put functionality above aesthetics for routing

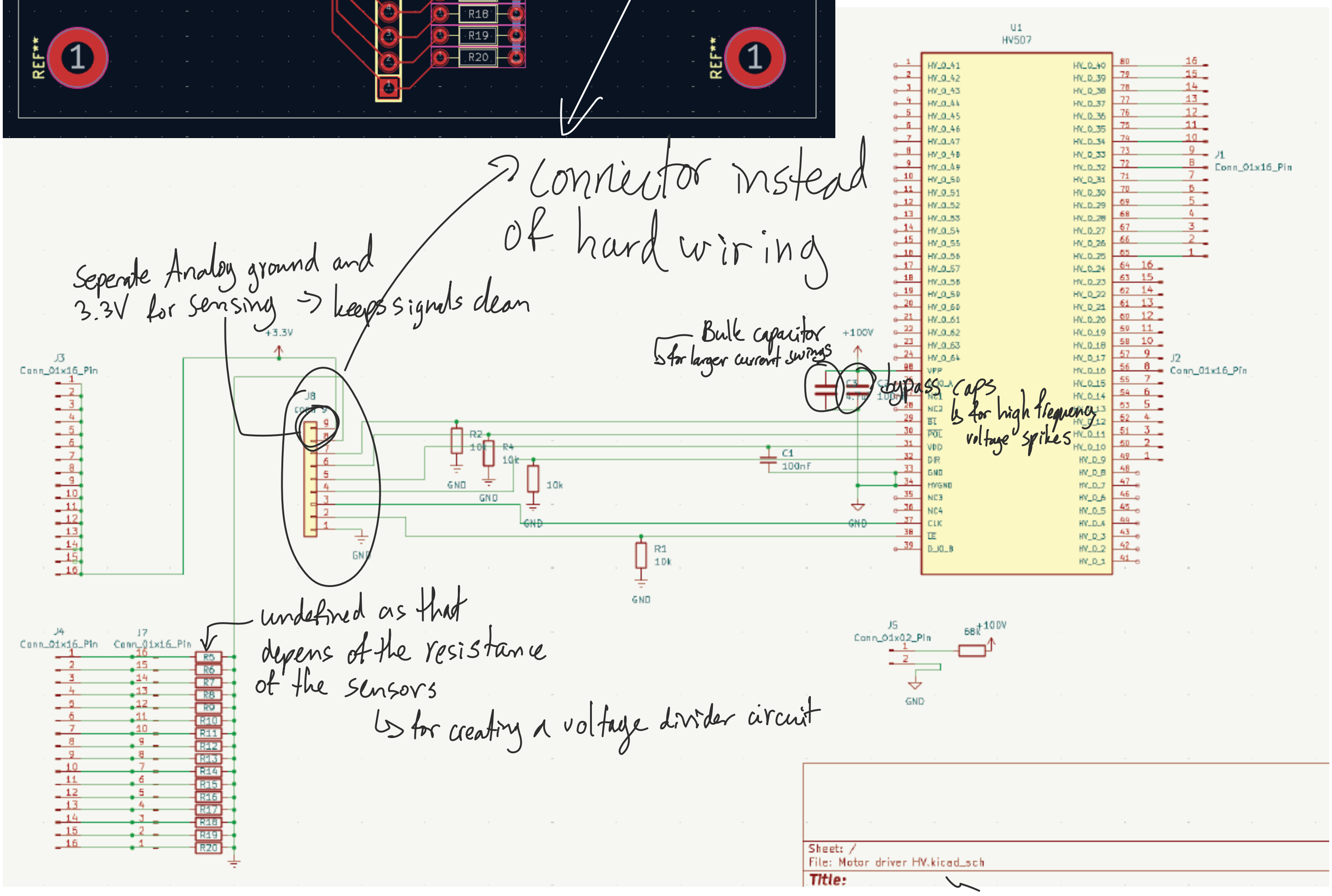
Current limiter



Star point grounding

Same size for sym and better performance

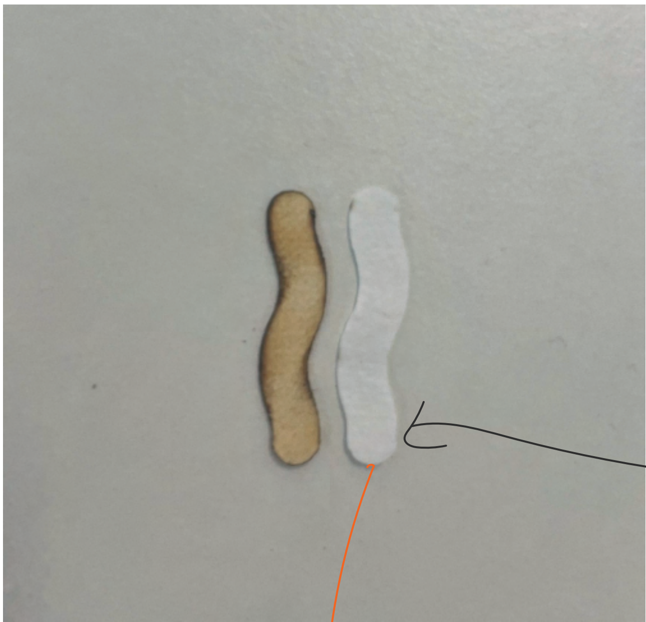
Did not minitorize



Learning a lot

Fabrication

Ventilation causes some to fly away



Tweaking lasercutter settings so the delicate glass fiber filter does not burn

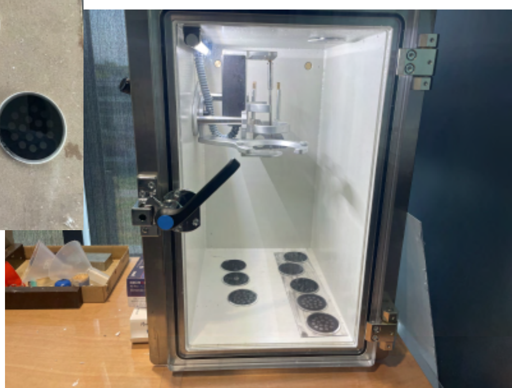
picked up with tweazers as its so delicate

Very fragile and delicate

Dragonskin 20
→ less flexible than ecoflex

extruded to get cleaner glueing results

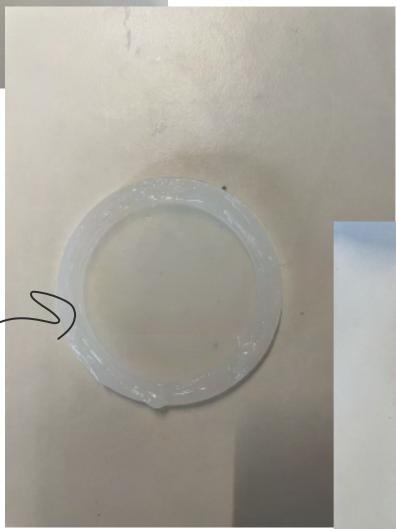
As it's so thin hard to get properly black



↳ Tried old silicone dye to get the top layer more black → DID not cure anymore

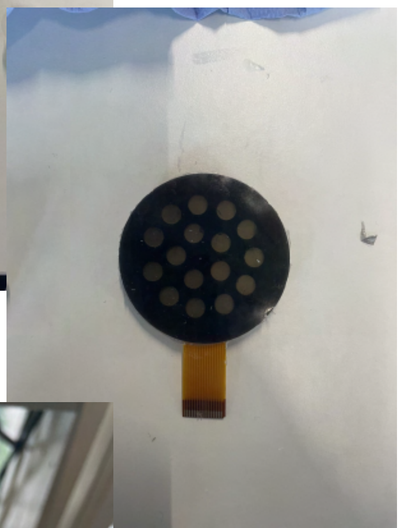
↳ switching to silc pig black dye worked well

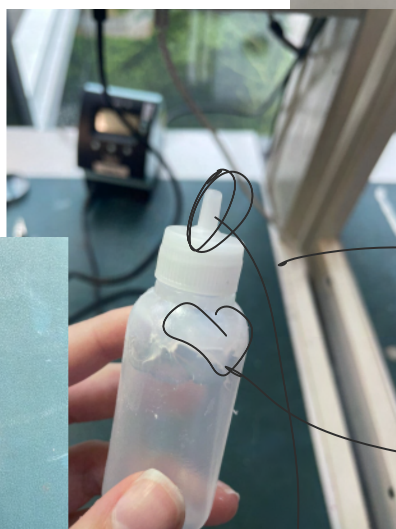
Learned by doing



Apply thick layer of silpoxy

Used a toothpick to neatly glue around the pumps





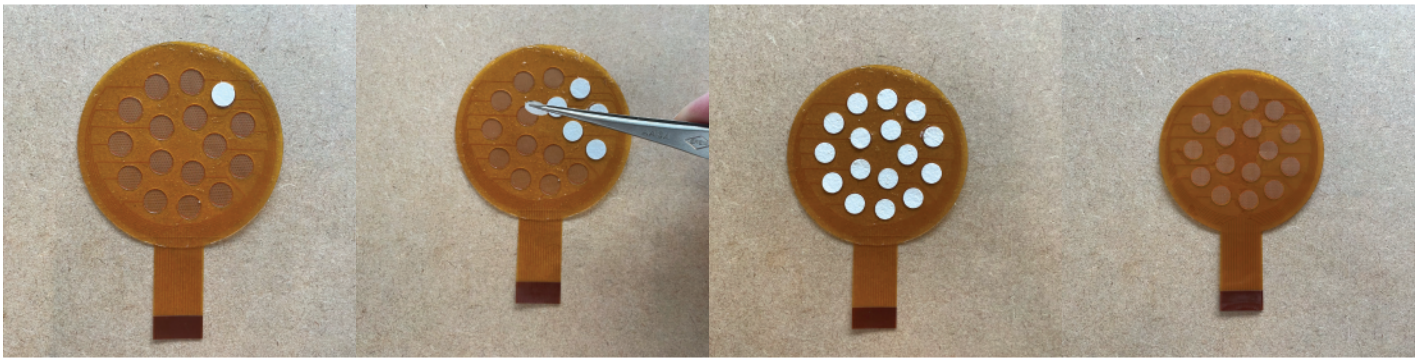
Tried this to apply glue precisely

dried too soon

Did not want to come out

Scalpel to clean up the edges

Assembly



Making of the pump layer

Testing without top layer

First assembly failed so I pulled it apart.

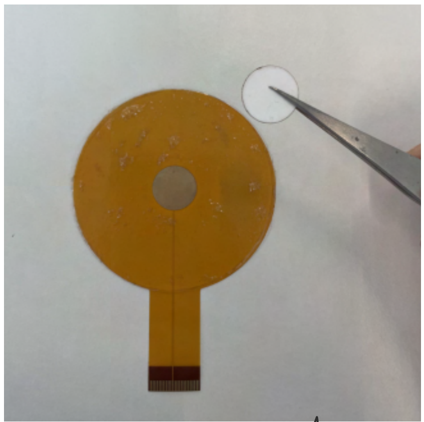
Too thick to actuate, cannot measure the thickness

liquid coming up at 50V already so it is working!

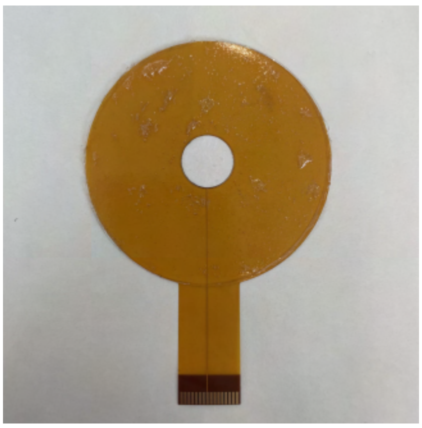
Testing, Trying, Testing → Learning

— logging so others can replicate my work

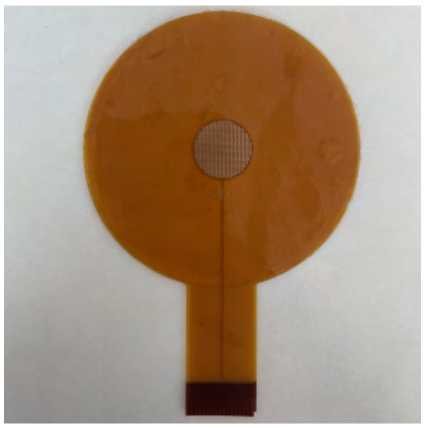
3layer double sided tape

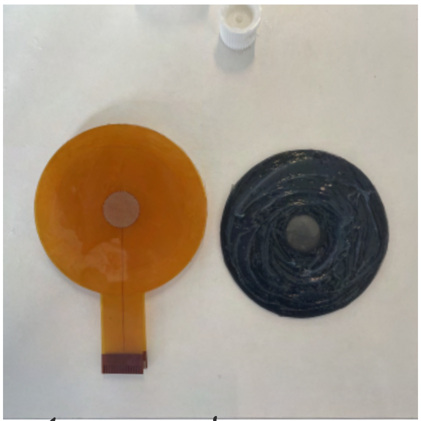


Add glass fiber filter

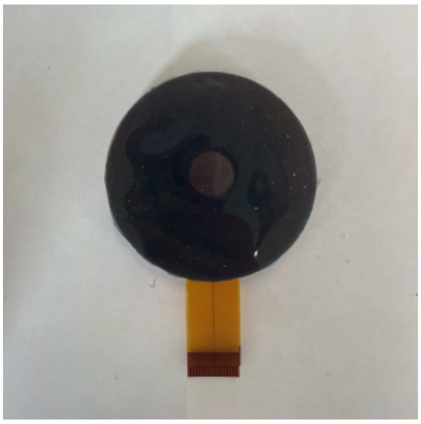


Add top electrode





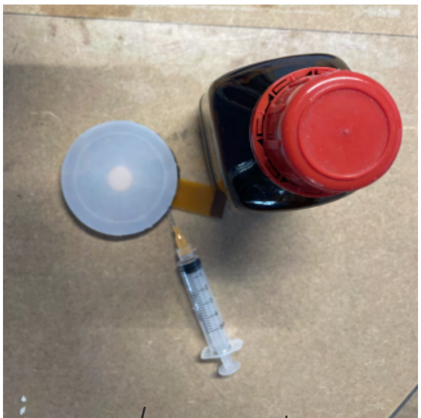
glue top layer



Press

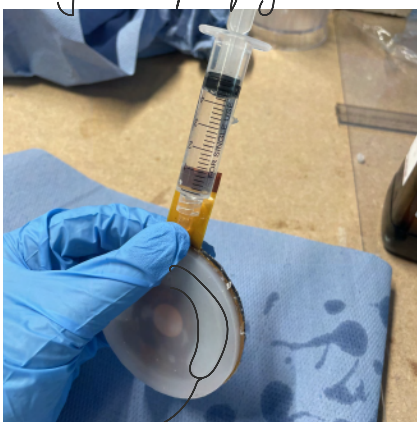
check edges

Test if air tight



Inject propylene carbonate

Vacuum and the fluid fills the reservoir



See air

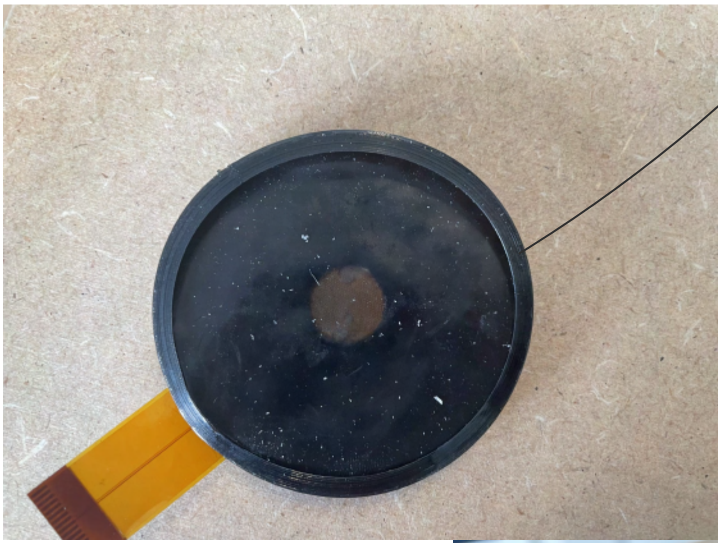
extract air

Seal hole with silpoxy glue

this was very hard to glue cleanly

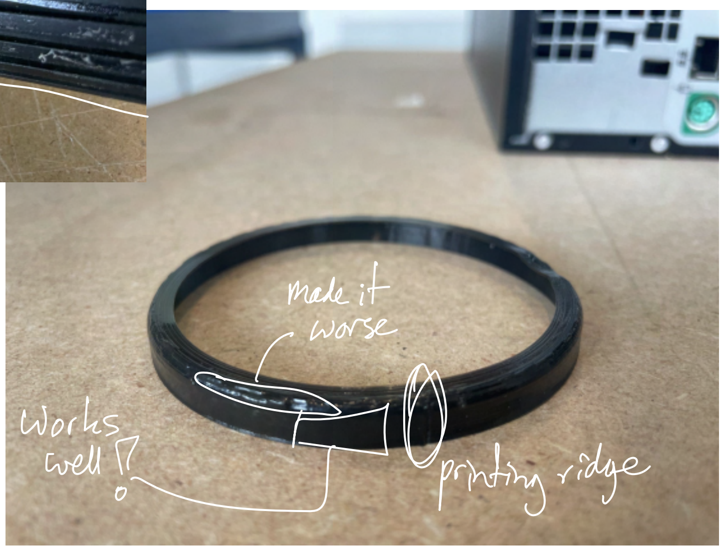
Did work out nicely

Making mistakes is costly with EOPs

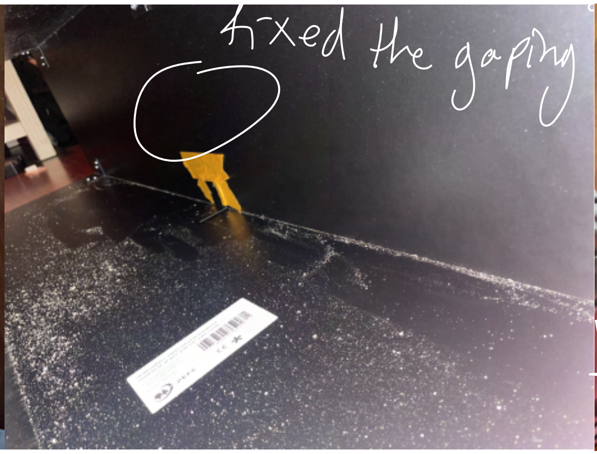


TPU cover to keep flexibility but covers up the edges

For a cleaner Look



Demoday

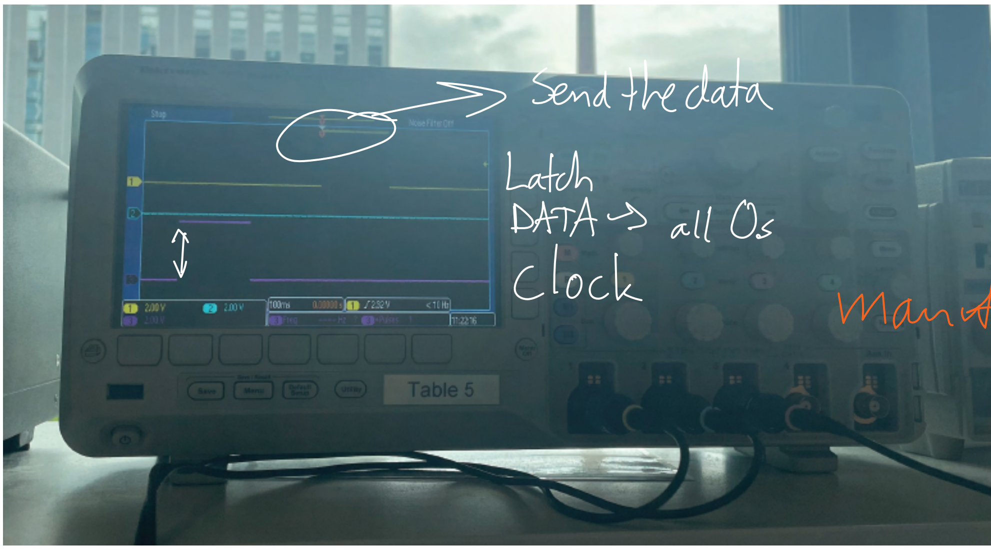



Using "keukenpanelen" to make a black pillar of 30 x 30 cm

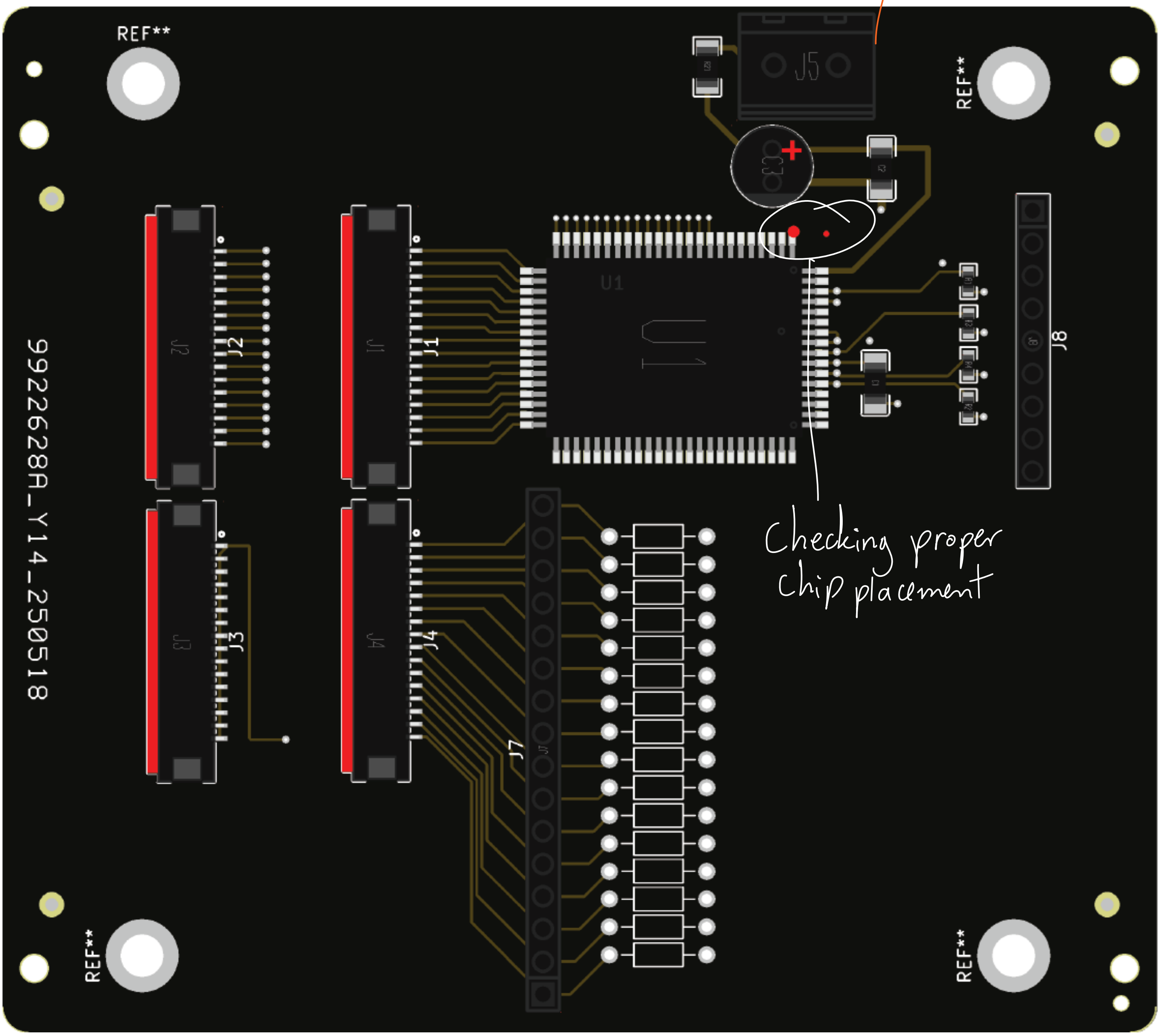
EOPs put into the display box

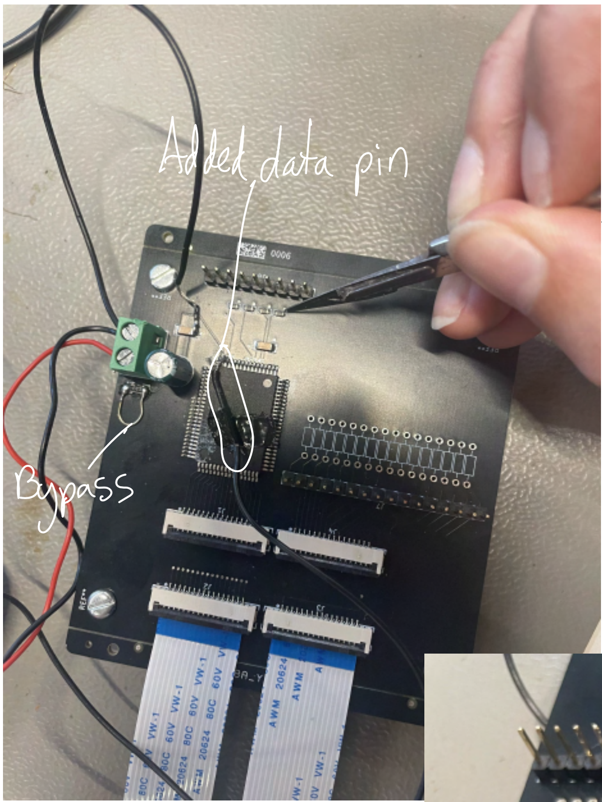
Simple explanation of electro osmotic flow was easily understood

Showed its potential

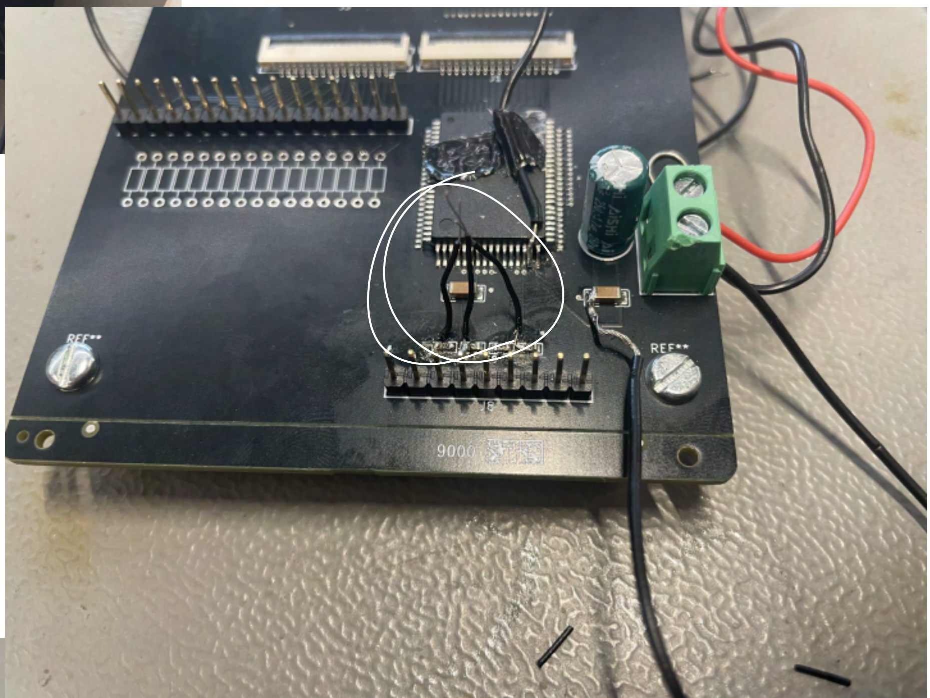


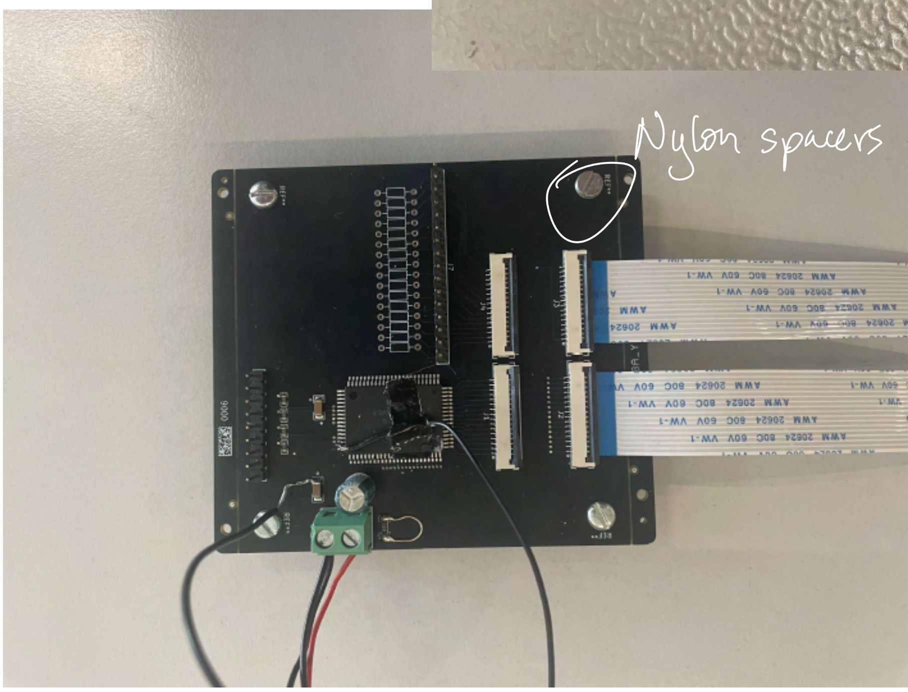
Visuals for manufacturer communication





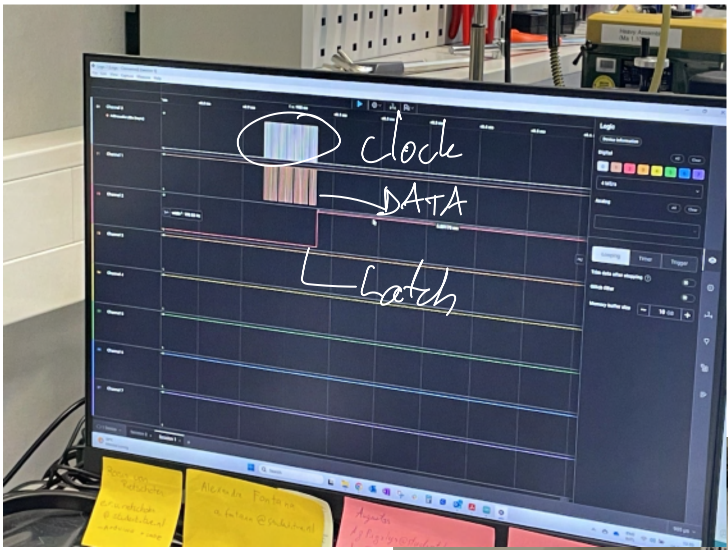
scratch out pull down resistors and make them pull up
↳ did not solve our issues

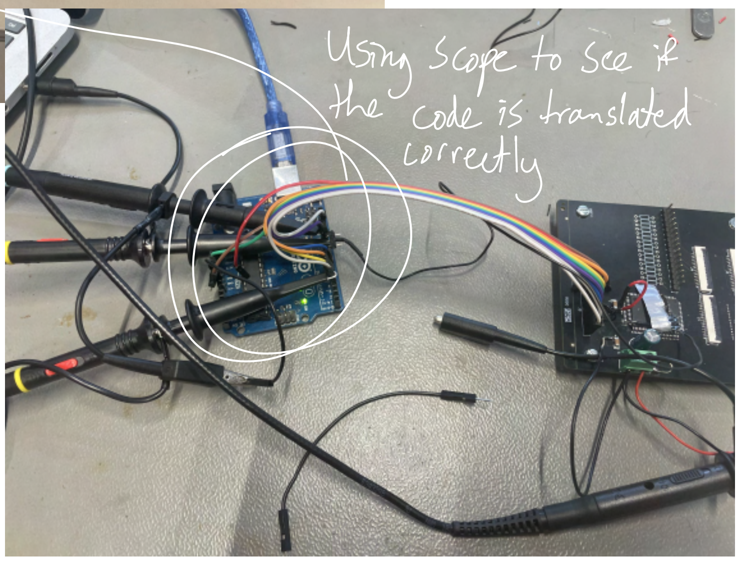




Issue must be in the software then!

Documentation on High Voltage chips is very minimal as they are not often used
↳ caused a lot of struggle
↳ there were no example codes or schematics

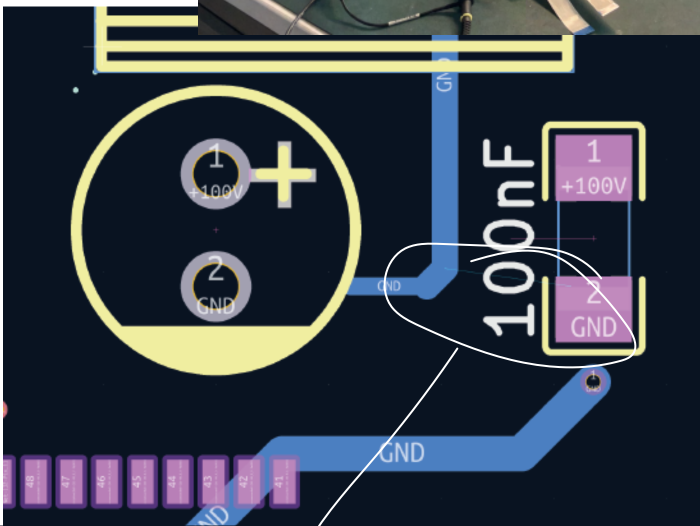


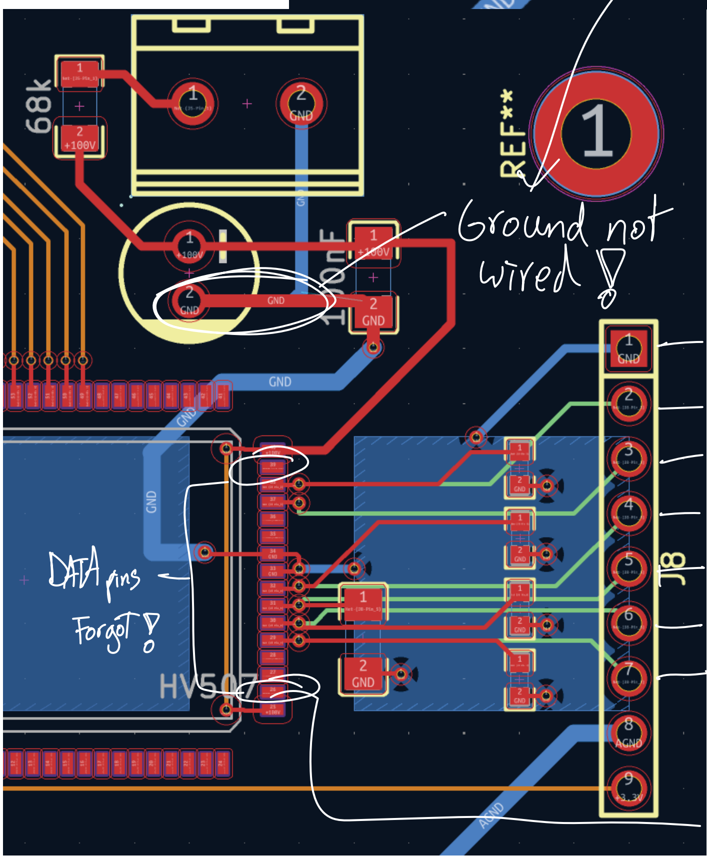


Very frustating process
↳ I just want to design motion

Control

First test measuring signal gave very weird values this was due to the high voltage ground not being connected





UNO

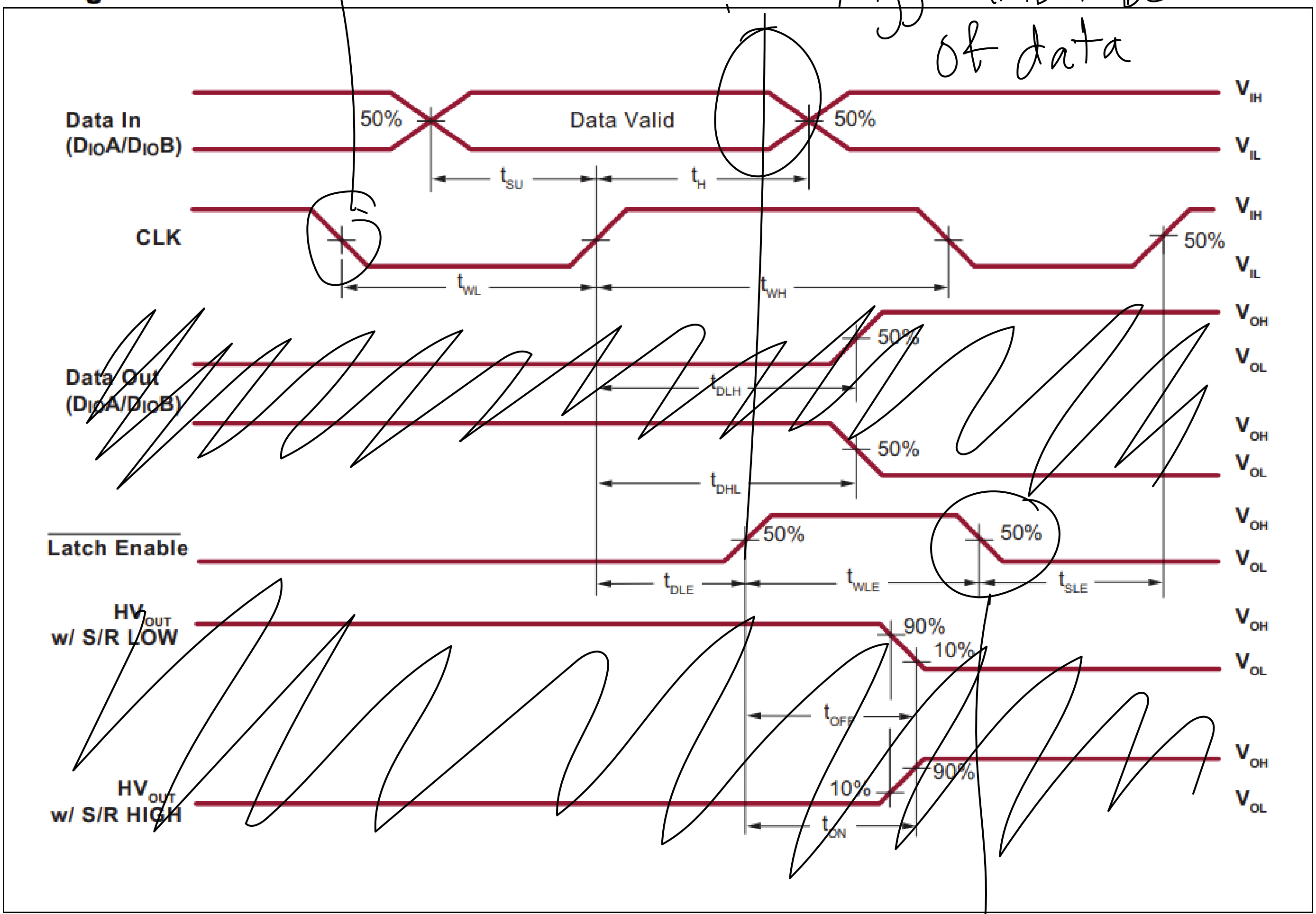
GND – GND
Latch – 4 digital
clock – 13 sck
Direction – 5 digita
VDD – 5V
Polarity – 6 digital
Blanking – 7 digital

DATA – 11 MOSI

First start clock

Timing Waveforms

Trigger latch before end of data



Data is stored when latch goes from high to low

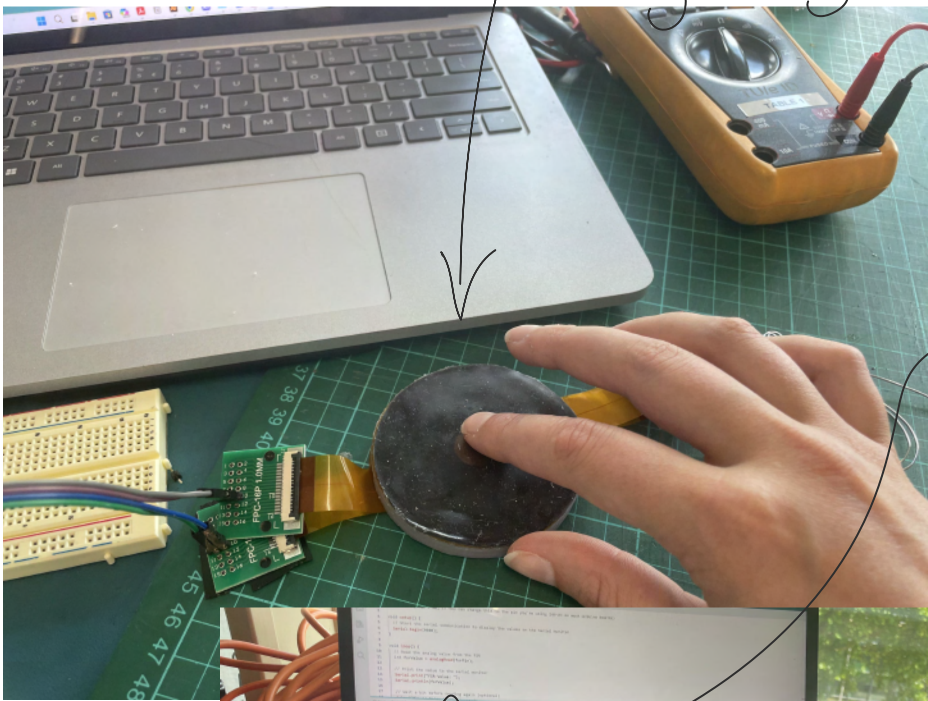
Felt more engineer than a designer

Spend two days exactly matching these timings → still did not work!

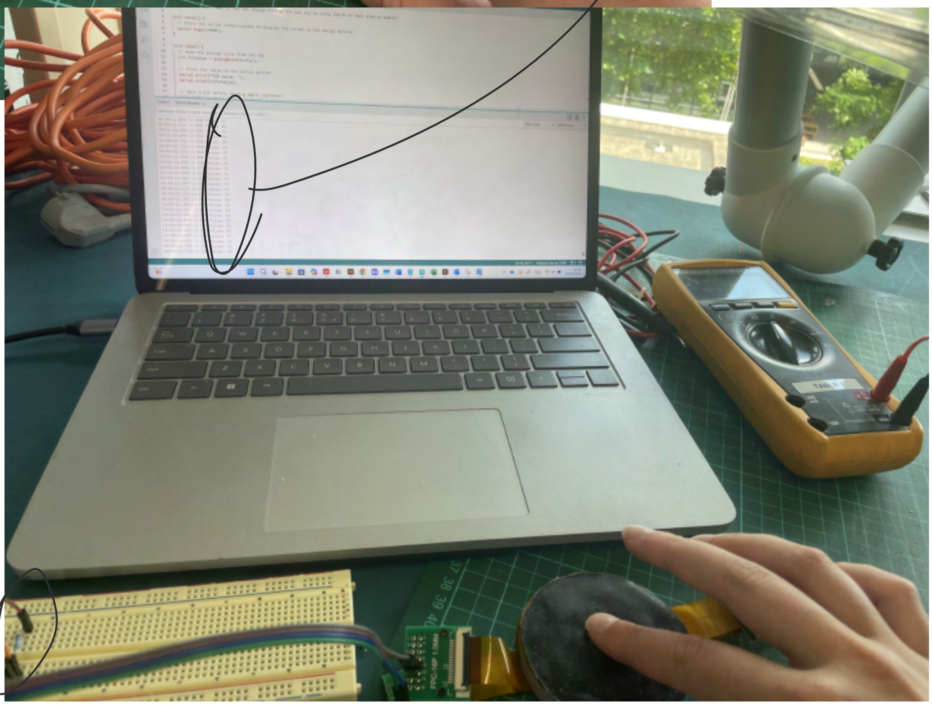
↳ found a github that set blanking to High → now it worked

Sensing

Testing sensing through the pump layer

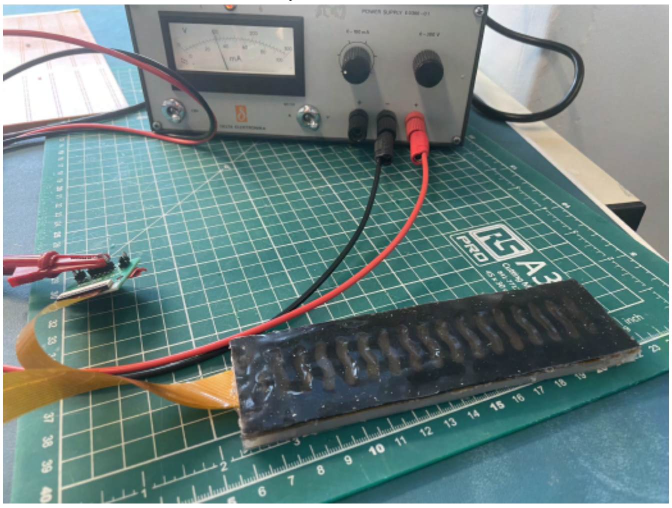


Works well



Simple voltage divider, the sensor has a resistance of ≈ 250Ω
so a 250Ω resistor was used

High voltage power supply



Actuation started to decrease over time

Stopped working

↳ drawing higher currents also

↳ 10–20mA → overheating

Refilling with new propelene carbonate fixed it

So the dragonskin is contaminating or degrading the fluid Temporarily

↳ switched to ecoflex reservoirs

EOPs are delicate and require precision