21M.370 Digital Instrument Design

CAP GRID example instrument

This document is an overview of the creation of the CAP_GRID instrument using the m370 framework. Here I will describe the mechanical construction of the instruments, and I will also upload videos to the YouTube playlist going through the code and pure data patches.

Aesthetic Goals

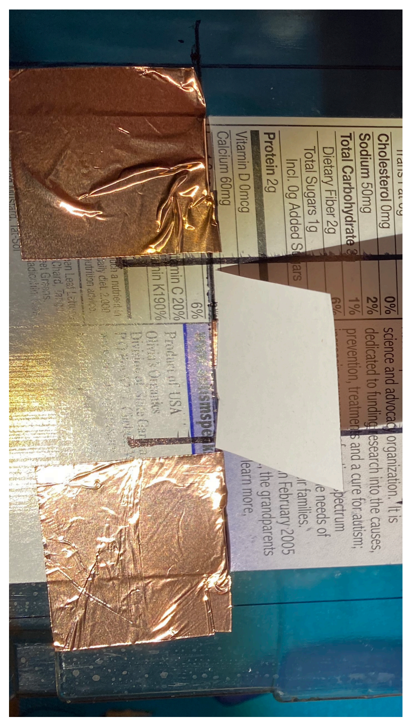
For this instrument, I knew I wanted to use the capacitive sensing that we have been working with, as well as some standard controls. I was inspired by a couple of things:

1. I really like the idea of working with rhythm in real-time, in a particular working with sequencers to create rhythms that are easy to repeat and easy to modify. This makes it easy to create complex patterns that are very tightly synchronized.
2. I also like the interface of the MakeNoise René:
   https://youtu.be/AqMCefZa1Gc
   https://youtu.be/moVbVBtYL5M
   https://youtu.be/Ab4rXKXGESs
3. So I wanted to create an instrument based on the René, but that also included more real-time control of the timbre.
4. I also wanted to explore getting different data streams out of the capacitive grid. In addition to just getting be touch values, I thought it would be fun to try looking at the overall capacitance of the grid as a control parameter as well as the average capacitance.

5. For this instrument, I wanted to the rhythm to be fairly consistent so that I could perform music that is in-time, but also wanted to include the ability to change the tempo by doubling or quadrupling it.
6. Sonically, I thought I would basically the instrument on a simple analog monosynth voice, and used an oscillator I hadn't used before (the wavetable oscillator) just to explore how it works.
7. Visually, it made sense to create this instrument in a ‘box’ of some sorts. To make it super easy I just reused a plastic container I had lying around .It isn't super robust but I knew that this version of the instrument wouldn't be the final one and a further prototype might be created using something that is more rigid.

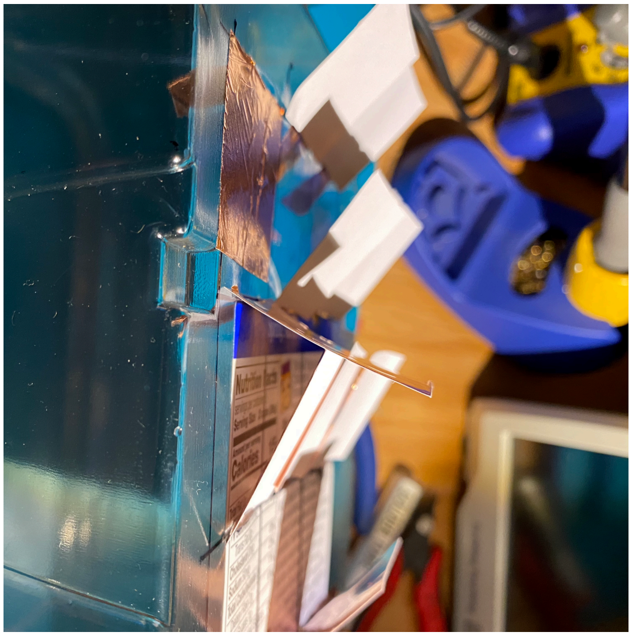
Mechanical design

Capacitive pads

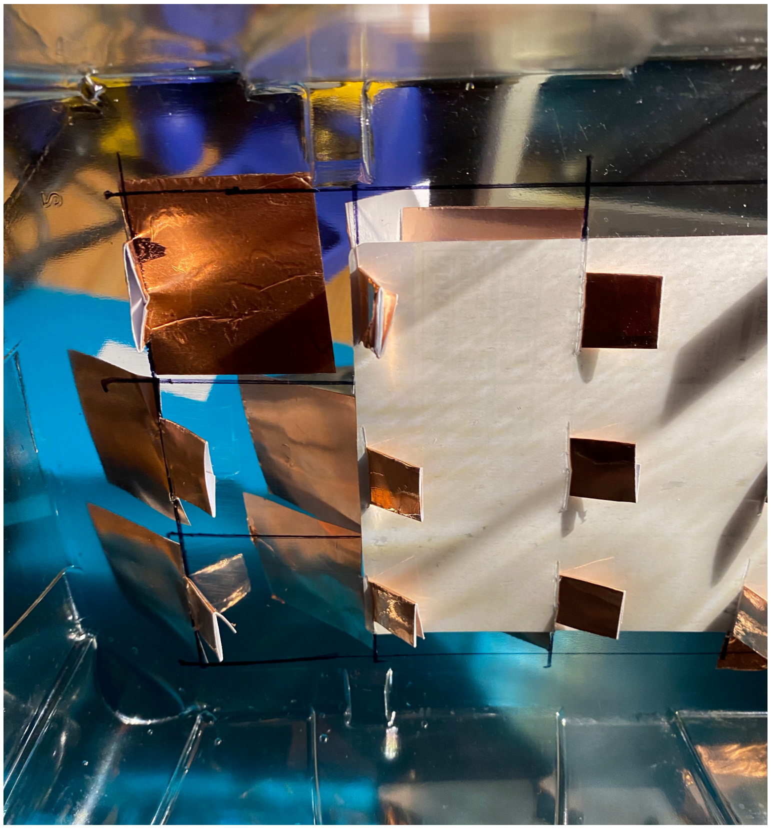


The touchpads are made out of the copper tape, in order to make a more robust connection I slaughtered wires to the pads and then also soldered them to headers which plug directly into the jumper wires we have been using.

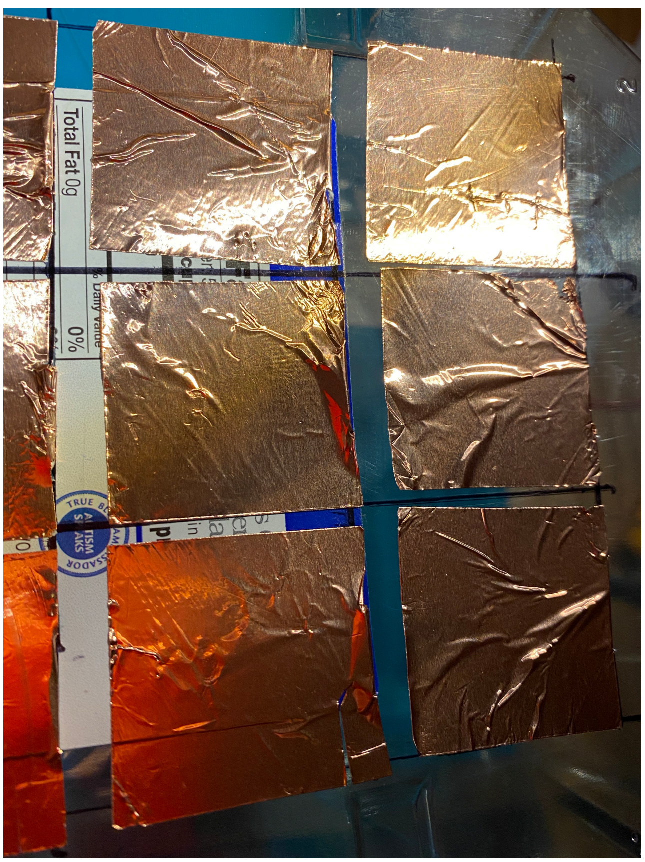
I wanted to avoid having the solder joints be touchable, to create a smooth playing surface, so I cut notches in the top of my enclosure which the tabs would slide through. First, I cut out all of the touchpads which are about 3 cm squares and with an additional 2 cm to create the tab. I cut slits to partially separate the tab from the touch pad, folded the tape over, and then slid the folded tab through the notch into the interior of the enclosure. I did all of that without removing the paper backing of the copper tape.

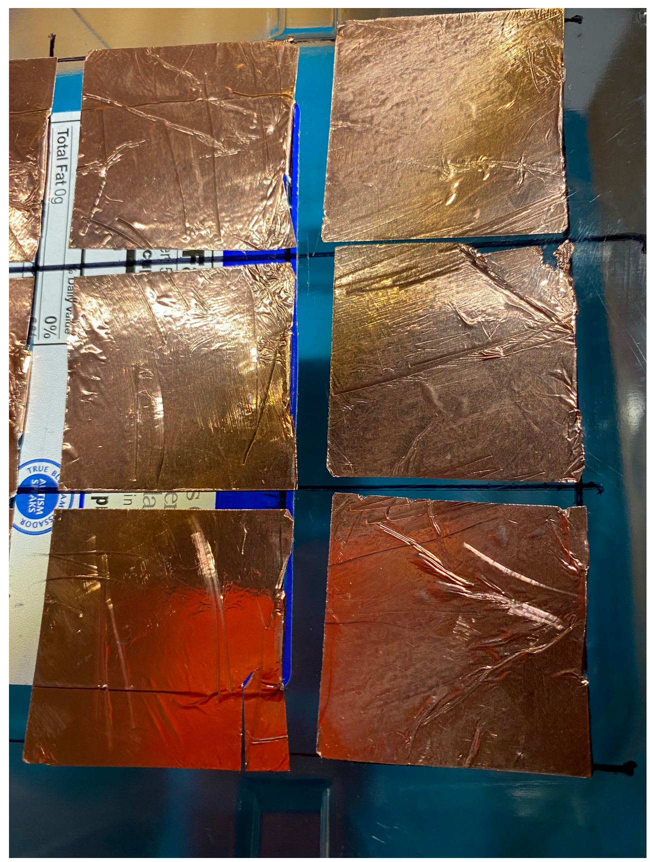


Once all of the copper tabs were inserted into the slots, then I took the paper off the touchpads, leaving the paper on the tabs, and stuck the touchpads onto the surface. I made sure to leave space between each touchpad to make sure they weren't touching each other, but also wanted them to be close enough that my finger could bridge the gaps.



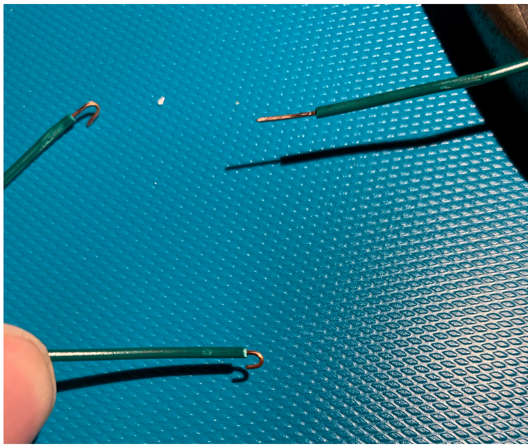
One additional step I highly recommend is to use a rigid object to squeeze out as many of the wrinkles in the tape as you can, and to firmly stick the edges of the tape to the enclosure. The tape edges can be sharp, and you don't want to have sharp services on your playing surface, and the smoother the tape is the more easily you can slide your fingers over the surface. In my case, the biggest problem is the fold where the tab is located, and little bits of paper backing still preventing that side of the touchpads from lying flat against the top of the enclosure. In future iterations of this instrument I might remove the paper from the tabs as well, and stick the tabs to the underside of the enclosure to make them more

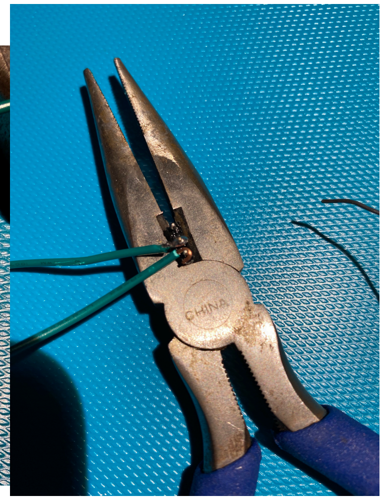


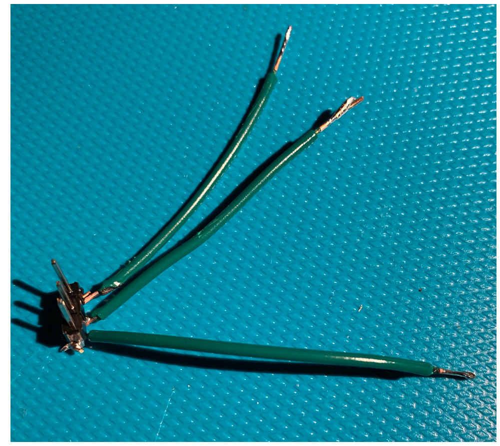


mechanically rigid.

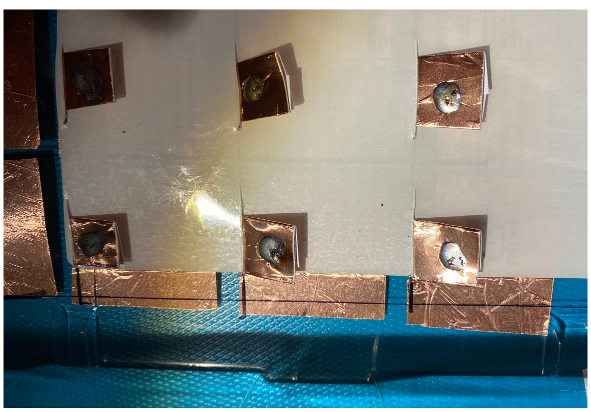
Once all of the copper touchpads had been stuck down to the surface of the enclosure I soldered wire to four sets of headers.

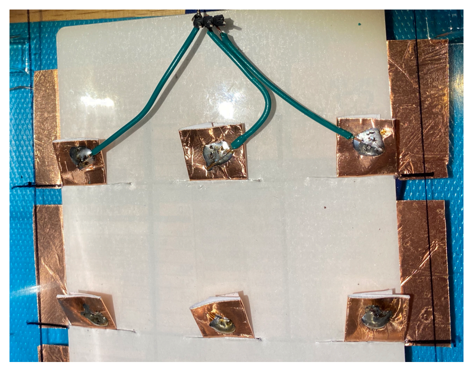






I soldered the wires to the headers first as that is a little trickier, and then once all four sets of headers have been prepared I soldered the wires to the tabs.





Then the last thing to do was just to connect the headers to the MPR121 breakout using the jumper cables.

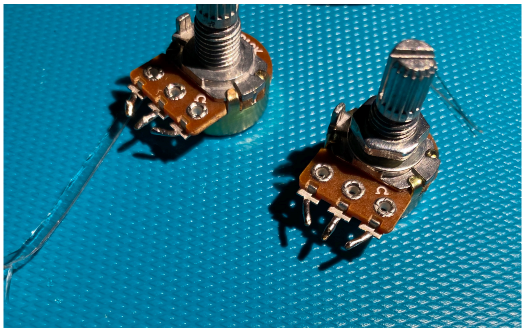
One quick note about removing the insulation from the wires. Your soldering kits include a cheap little wire stripper, but if you don't have access to a better quality wire stripper I find that using the pliers and cutters you can actually do a good job of removing the insulation fairly easily.

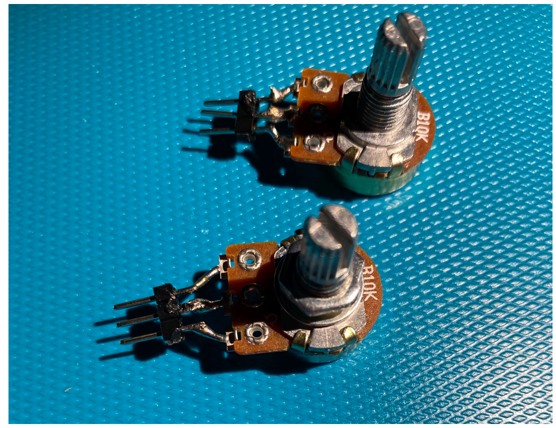
The key to this is that you can stretch the insulation to break it rather than cutting it. Cutting the insulation often leads to nicks in the wire itself, while stretching the insulation to break it is less likely to do so. To do this, I hold the wire really firmly in a pair of pliers, and then take my cutter and squeeze gently on the insulation where I want to strip it. Then squeezing firmly on the pliers and gently on the cutters (so as not to cut through the wire), I pull the cutters away from the pliers and off the edge of the wire. This stretches the insulation and it will break where the cutters are located. It takes a while to get the hang of this, but I recommend trying it out if you don't have access to a good set of wire strippers.

Standard controls

For the standard controls, I used a box cutter to cut a slot in the top of the enclosure for the fader, and then also cut out two holes for mounting screws. You have to make sure that the slot is wide enough that the fader doesn't hit the top of the enclosure at all in order to prevent friction.

For the potentiometers, since they don't already have headers I bent the pins together and soldered the wires directly to the pots.

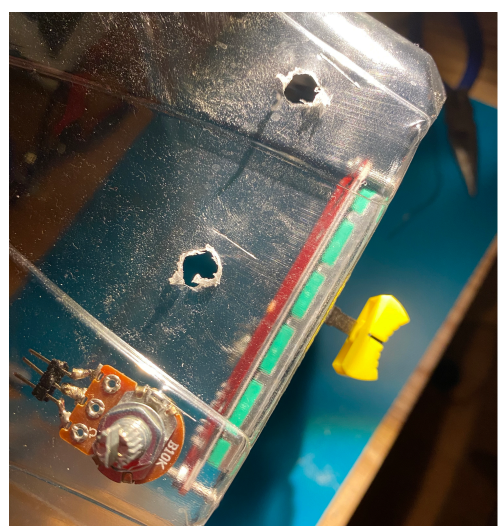




Note that the signal layout is different than our PCBS:

- for the PCB the standard layout is signal/3V/GND.
- for the pot it is 3v/signal/GND (with the signal on the middle pin). Which of the outer pins 3v and GND are connected to will determine the direction of rotation, i.e. whether turning the pot clockwise increases or decreases the signal. This is easy to change in software so I wouldn't worry about it at this point.

I cut out holes for the potentiometers and also cut out a little slot that the potentiometer's tab would fit through. Most potentiometers have that little tab, and when you are mounting it to a rigid surface like wood or metal it is normal to break little tab off with a pair of pliers. In this case, since I am using a really flimsy enclosure I had a sense that the potentiometers would not grip the plastic firmly enough and so I used the tabs to keep them from rotating when I turn the knobs.



Mounting the PCB

I cut a couple of holes in the enclosure for screws to mount the main PCB to the enclosure.

I also cut a slot for the USB cable to slide through so that I could seal the enclosure up. For this prototype, I left the MPR121 breakout bored just hanging loose inside the enclosure. If I were to make a more permanent model I would find some way of tying the jumper cables to the enclosure or otherwise fixing the MPR121 board so that it doesn't rattle around.