Daniel Mendez: Board 3 Report

Project Overview and Plan of Record

This project involved the building, layout and testing of a golden version of an Arduino UNO using the atmega328P. This version would have conformed to a lot of the PCB best practices that have been learned thus far and should provide better figures of merit when compared to that of a commercial Arduino UNO. For my project, the commercial Arduino UNO being used is made by ELEGOO. The golden PCB which is being designed would have the basic features required to run code on the ATMega328P microcontroller via connection to the USB port. In addition to this, it would have the following features:

- Compatible shields for the Arduino UNO should be able to plug into it.
- There would be a 0.5-ohm series resistor on the power rail so that the supply current can be measured .
- Power can be supplied to the golden PCB using either a 5v AC to DC converter or a mini-usb port.
- Test points would be added to sniff the USB, UART,I2C and power rail lines.
- A ferrite bead filter to the AVCC pin of the 328 MCU would be added.

What it means to Work:

The following points define what it means to work for this golden PCB.

- A shield designed for the Arduino UNO should be able to fit into the board without issue.
- The EMI noise from the golden pcb should be less than that of the commercial Arduino
- There should be a minimal inrush current noise due to the Arduino.
- The board would be less than 3900 X 3900 milis in area.
- The board and any connected computer would be protected from ESD events by the TVS diode.
- There would be a push button that enables the ATmega328P to be held in reset.
- The MCU would be clocked by a 16 MHz crystal oscillator.
- A CH340 chip would be used to interface a usb mini port D+ and D- pins to the mcu RX and TX pins.
- There would be a 3.3v output on the golden PCB via the use of a 3.3V LDO.
- The MCU can be successfully programmed using a USB cable and the Arduino IDE.
- The MCU can be flashed using another Arduino and the Arduino IDE.

Risk Reduction

Risk reduction is key in the assembly and bring up of any electrical project. For this project, the following steps were followed to minimize risk during the bringing up and testing of the board.

- When soldering, goggles were worn to protect against any accidental splatters of solder.
- Isolation switches were placed by the VCC of the ATMega328P as well as the CH340. This is so that first the CH340 could be brought up and tested before connecting the MCU to the circuit.
- 20 mil traces are used for power lines to allow them to carry more current. Smaller width traces could lead to overheating and potential blowing of the trace.
- Decoupling capacitors are used by the VCC pin of several IC's as to prevent situations that can damage or affect the IC operation when there is a voltage droop due to noise.
- A TVS diode is placed by the mini usb outputs to prevent ESD events affecting the host computer.
- The circuit would be built using a solderless breadboard and tested for the same functionality(not performance) that is expected of the golden pcb circuit.
- Ensure that RX and TX lines are routed to be the same length.
- The number of cross-unders used as well as the length of them are limited as to leave the ground plane as continuous as possible.

Component Listing

The main components that make up this board include:

- 22uF and 1uF decoupling capacitors.
- An ATMega328P microcontroller
- 8 and 6 pin Female headers for connecting shields.
- 12 and 16 MHz crystals
- A CH340 chip
- A mini usb jack
- A ferrite bead filter rated at 10uH.
- Indicator LED
- Resistors

Schematic

The schematic for the circuit is shown below:

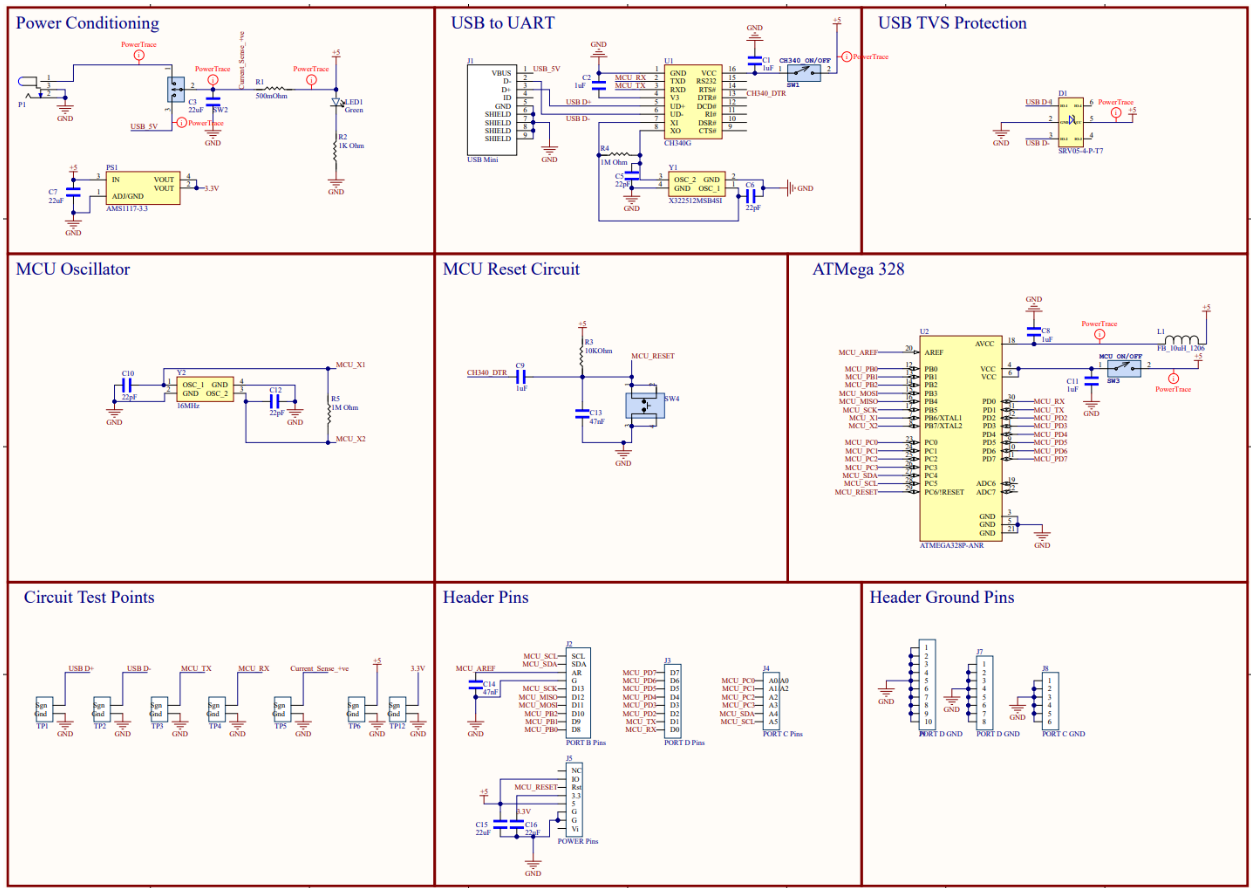

Figure 1 Schematic of Golden Arduino PCB

PCB Layout

The finalized PCB layout can be seen below in both 2-D and 3-D

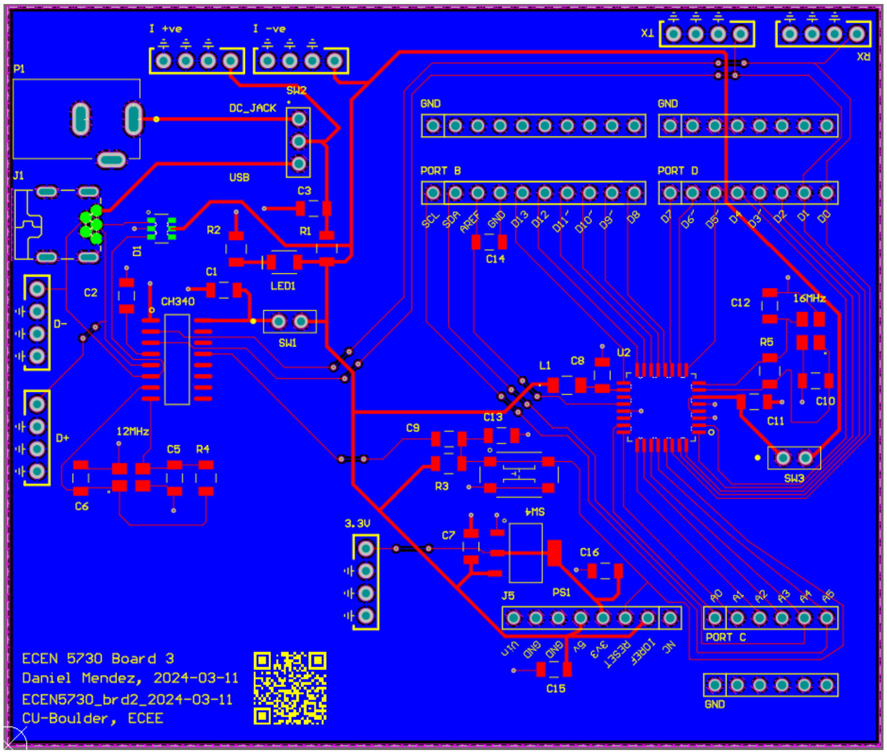

Figure 2 2-D PCB Layout

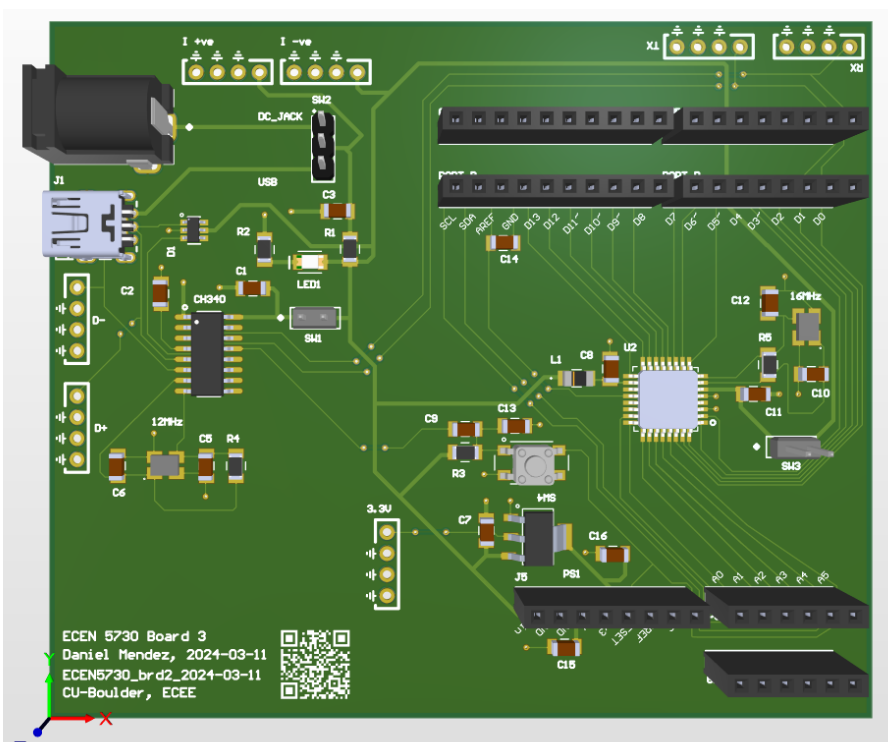

Figure 3 3-D- PCB Layout

Pictures of the board

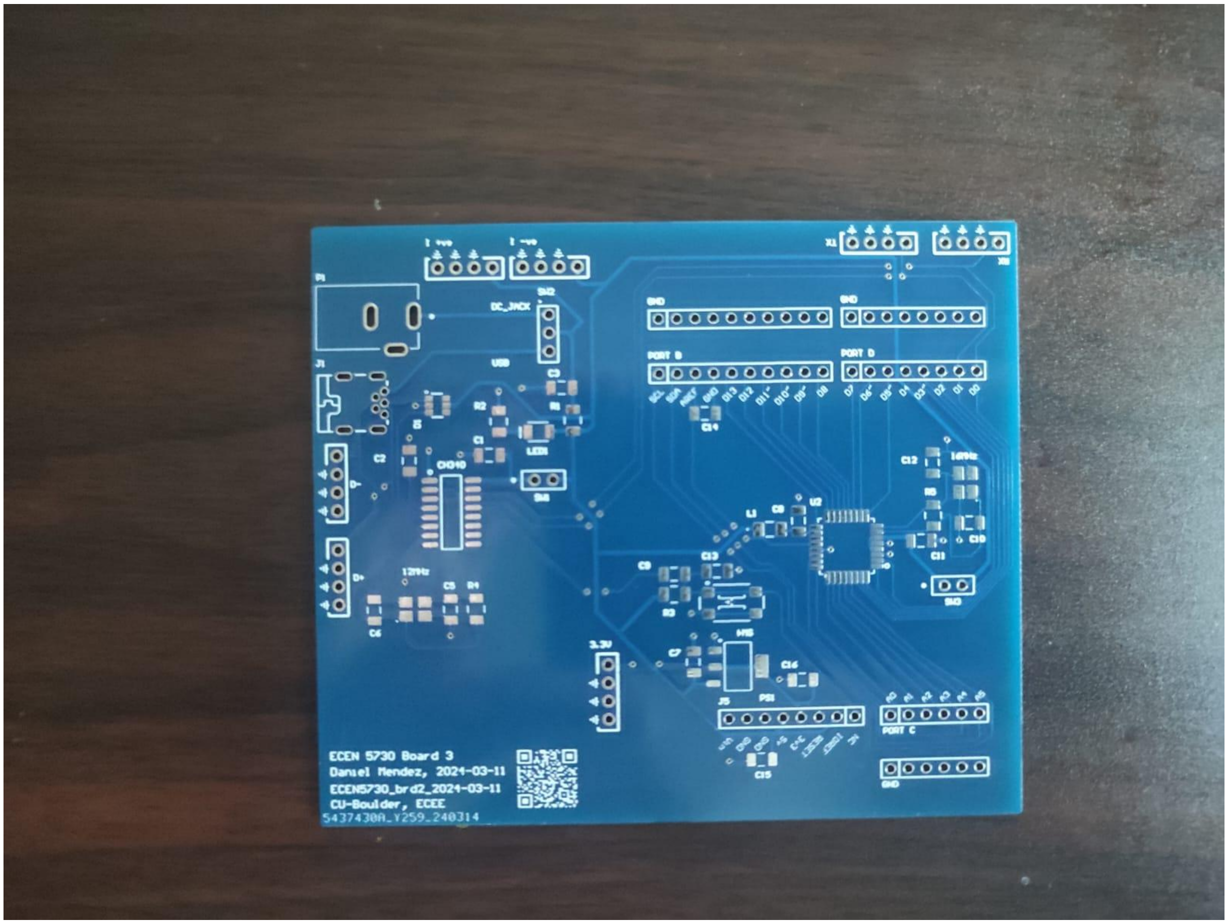

Figure 4 Bare PCB as printed

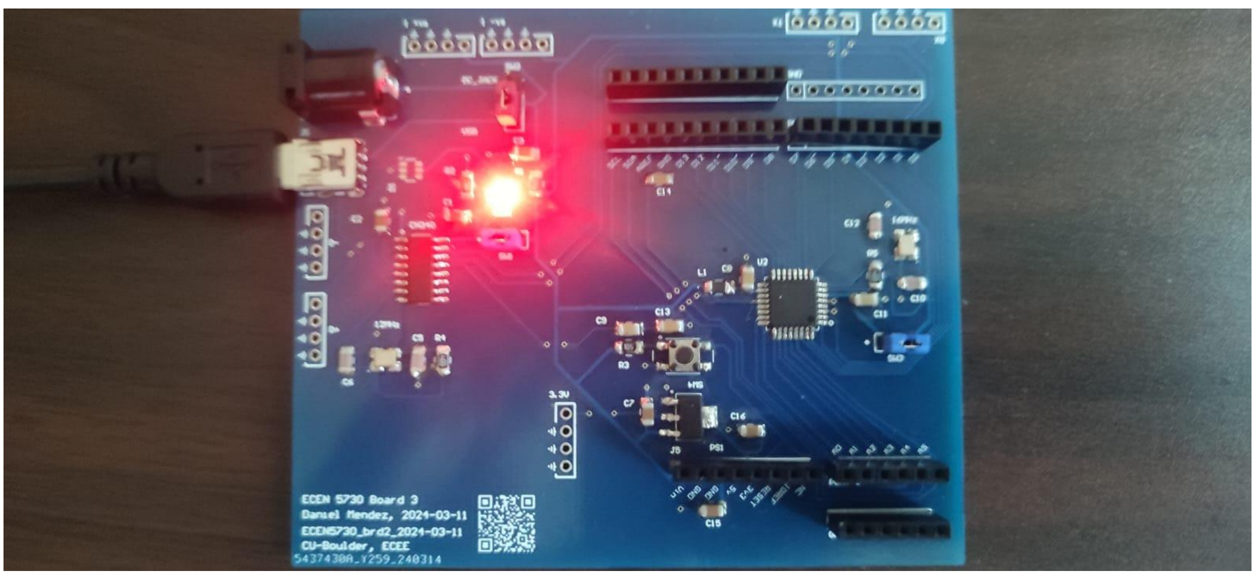

Figure 5 The Board with components soldered and powered on

One thing to note here is that the bottom left header pin of the golden PCB board was unfortunately routed to be upside down. This was truly unfortunate as I saw this only when it was printed and was bringing up the board for testing. For all my comparisons, I would therefore have to use jumper wires which would introduce some artifacts to the measurements being taken.

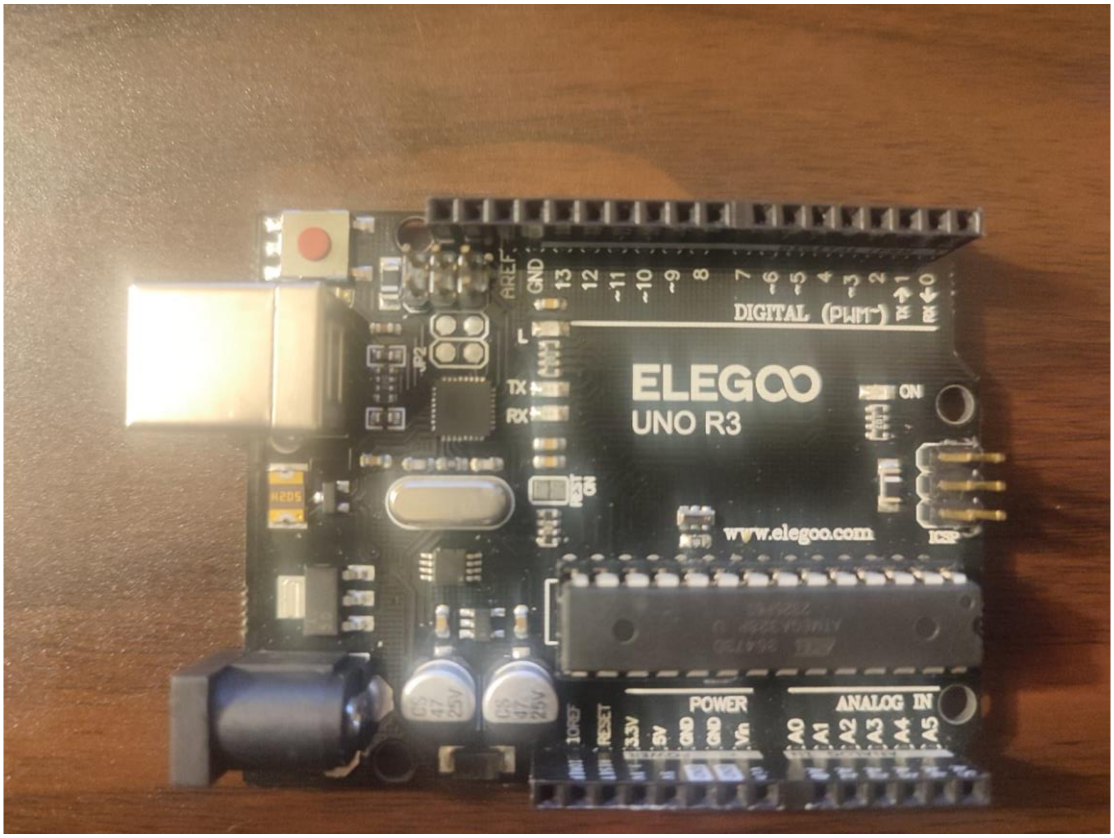

Figure 6 The commercial Arduino used for comparison. Cheaper than the official arduino

Bring up Plan.

1. Firstly, the PCB would be inspected so that the layout is as expected and free of error.
2. The components would be soldered making sure to take the regular safety measures and best practices.
3. The soldered board is visually inspected for any bridging of pins especially by the MCU and the CH340 IC's.
4. All the switches are left open and then firstly the dc power is enabled by placing the switch in that position, followed by the USB power. The LED is checked to see if it turns on in both cases.
5. The 3.3V test point is then checked to ensure the LDO is operating correctly.
6. The ch340 circuit is powered on while the circuit is connected to a PC using a USB cable. Then a serial terminal is opened, and a character is held. The D+/D- and RX test points are checked.
7. The power switch to the MCU is then connected and then flashed using another Arduino.
8. Example blink code from the Arduino ide is then programmed to the Golden PCB board ensuring that it operates as should.

Measurement Setup

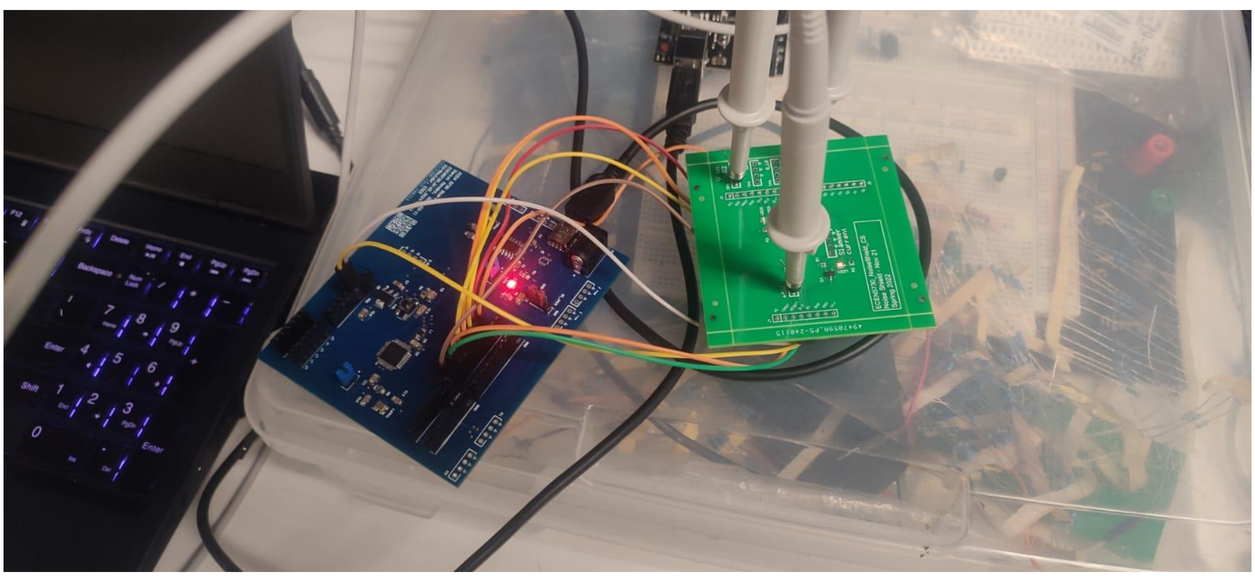

Figure 7 Setup of the shield with the board.

Board Functioning and Analysis

Measuring D+,D- and the RX and TX signals

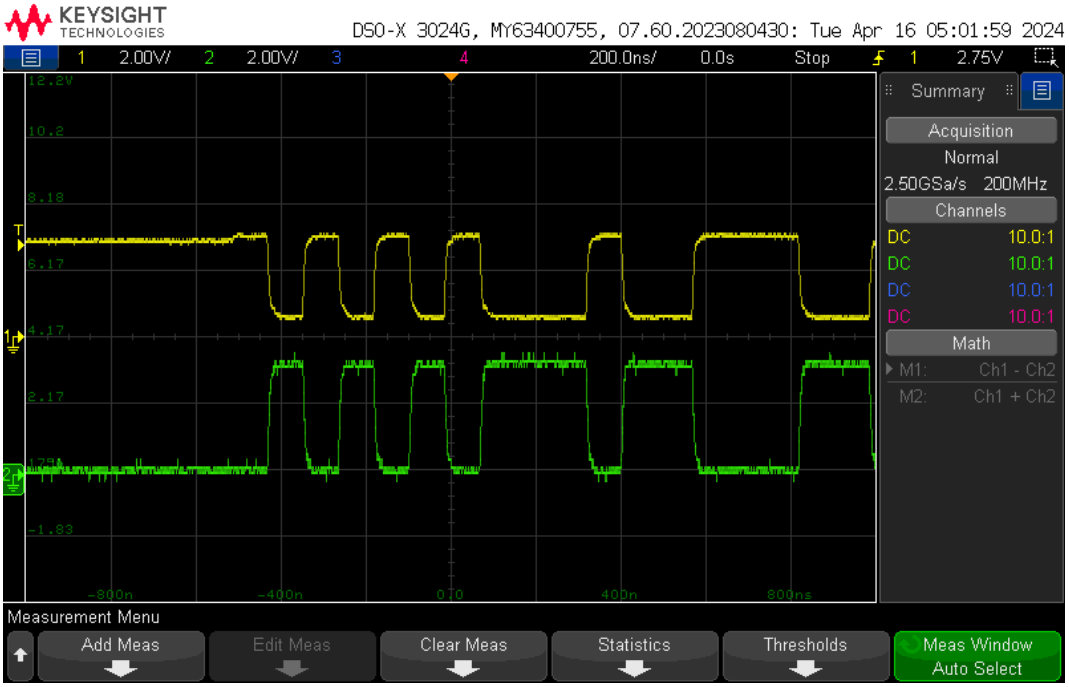

Figure 8 D+ (yellow) and D- (green) signals.

The screenshot above was obtained using the D+ and D- test points. This simply confirms that the usb port was wired properly and the computer can send signals via the USB cable to the CH340 chip.

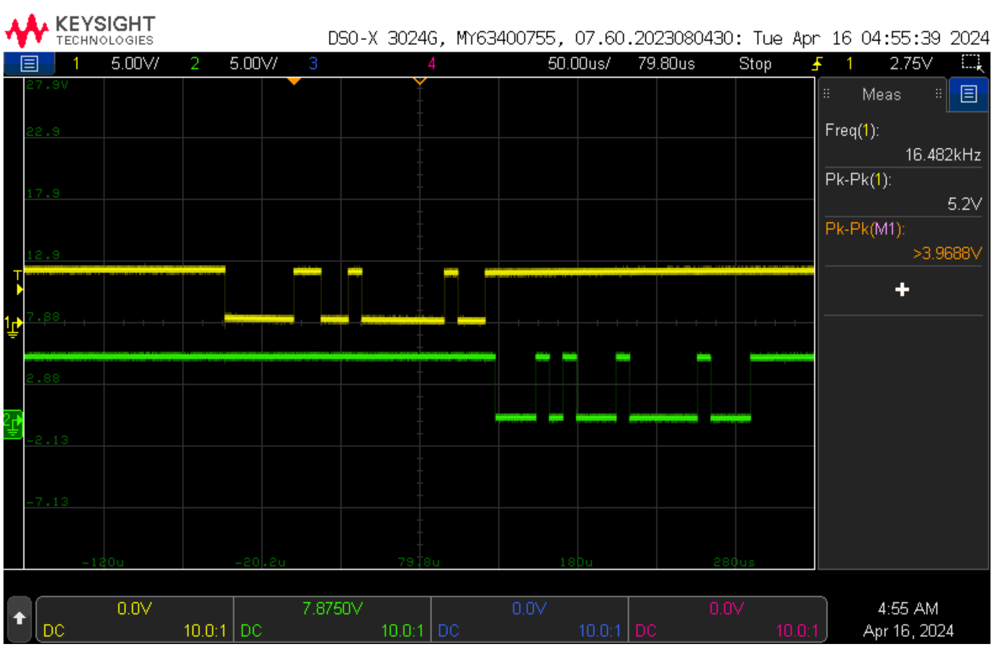

Figure 9 RX(yellow) and TX(green) signals.

The screenshot above in Figure 9 is what is seen when the board is being programmed. These were obtained from the RX and TX testpoints

Here we can see the RX and TX toggling between 0 and roughly 5V as required. This means that the CH340 is operating as it should and is converting the signals from the D+ and D-.

Switching noise on Quiet low comparison

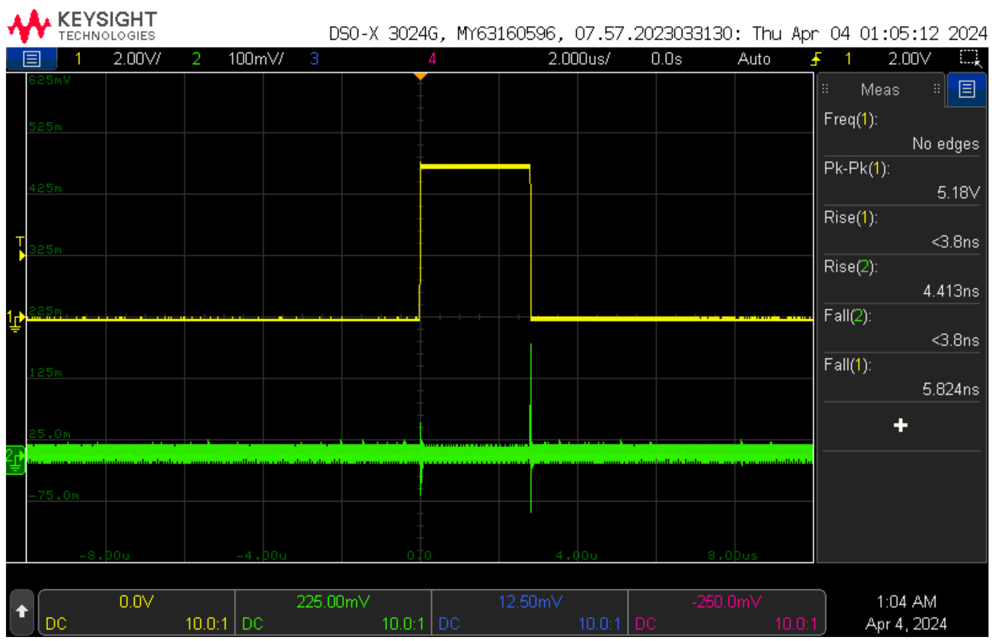

Figure 10 Golden PCB Quiet low switching noise

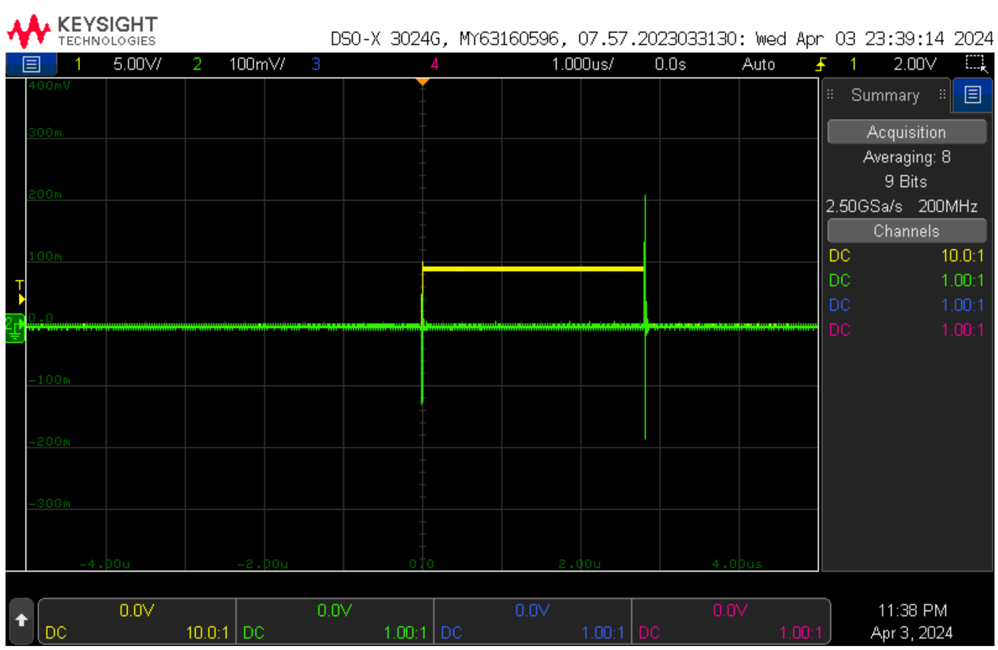

Figure 11 Commercial PCB quiet low switching noise.

From both diagrams, the switching noise seen on the golden PCB is at maximum at the falling as compared to the rising edge of the signal simply due to fall times typically being shorter than rise times which leads to a larger dI/dt. In comparing the both however, the peak-to-peak noise of the commercial Arduino is around 400mV at the falling edge whereas the noise seen at the falling edge of the Golden PCB is around 205mV peak to peak. This could be possibly due to a more continuous ground plane in the golden arduino leading to less ground bounce than in the commercial. Also another contributing factor would be the rise and fall times of the pins which would be analyzed later on.

Switching noise on Quiet high comparison

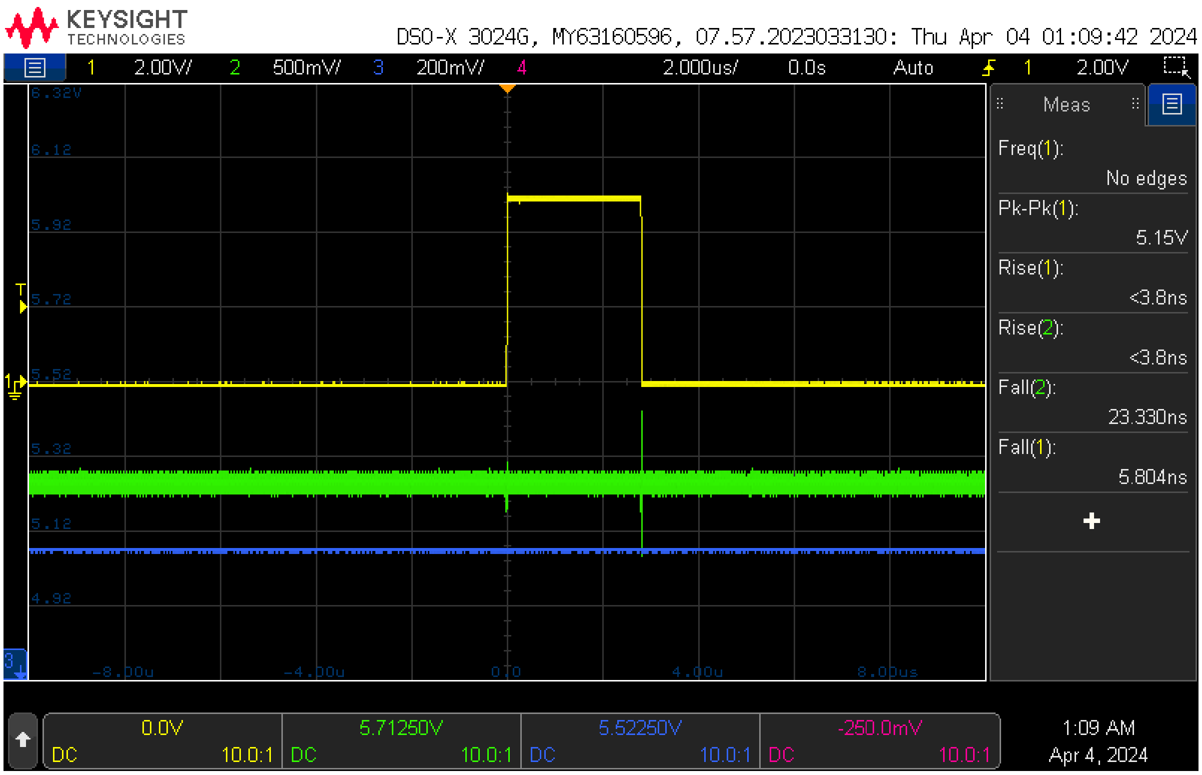

Figure 12 Golden PCB d13(yellow) Quiet High(green) 5V(blue)

Here, we can see that in the golden PCB (Figure 12 above), there is around 125 mv Peak to peak noise at steady state for the quiet high in the golden pcb whereas in the commercial Arduino (Figure 13 below) it is somewhat above that at around 240mV. Noticeably though, at the rising edge, there is an around a 1V drop on the Quiet High line at the rising edge. Also there is a droop on the 5V power line when the signal is high when looking at the commercial Arduino plot. This may be since in our golden pcb, we may have had decoupling capacitors closer to the VCC pin of the MCU than the commercial board.

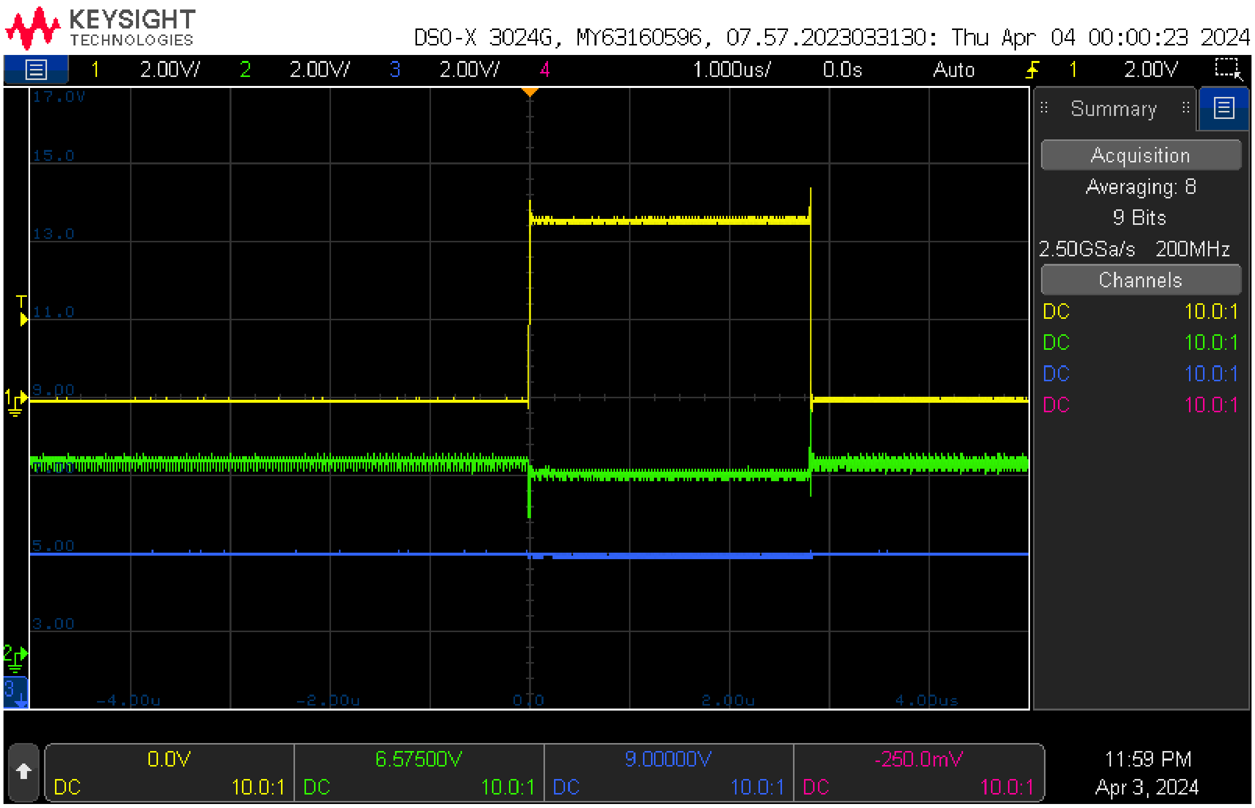

Figure 13 Commercial Arduino d13(yellow) Quiet High(green) 5V(blue)

Comparing the rise times of the switching signal

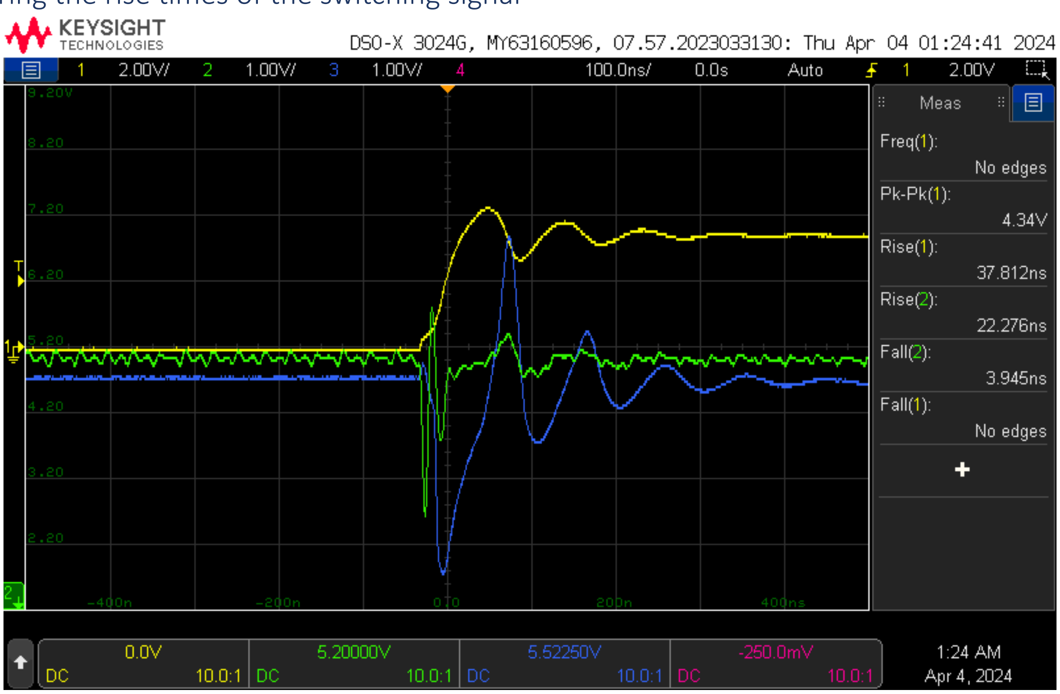

Figure 14 Golden Arduino d13(yellow) qhi(green) 5v(blue)

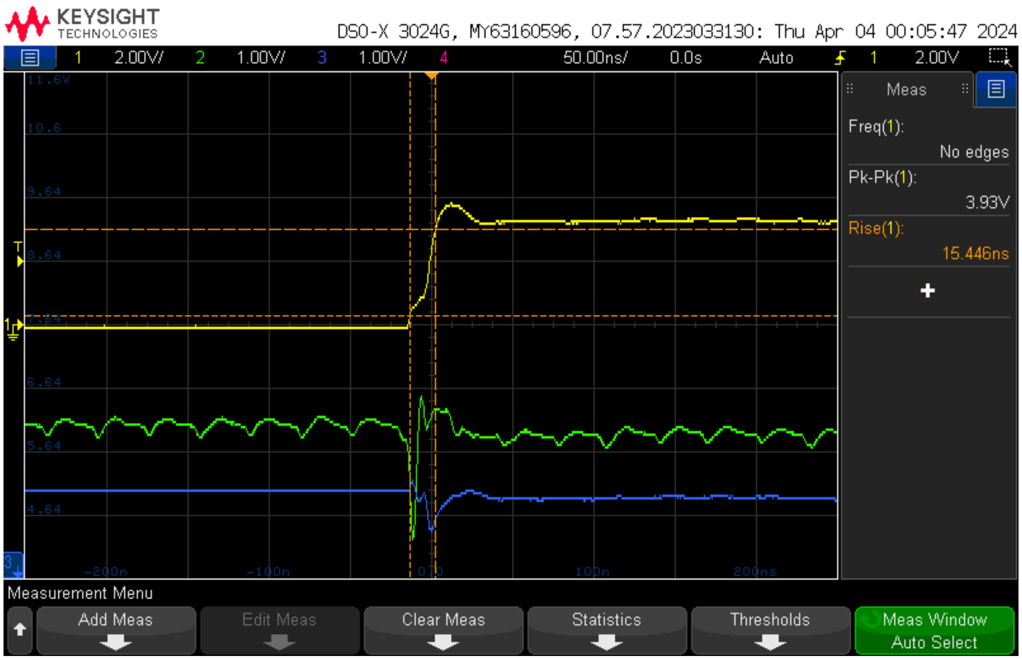

Figure 15 Commercial PCB d13(yellow) qhi(green) 5v(blue)

The two images above compare the rise times of the switching signal in the commercial and golden PCB's. Note that in the golden Arduino, the rise time is 37.8 ns as opposed to the commercial PCB having a rise time of around 15.5 ns. This would be since a larger capacitor could have been used on the golden PCB which was around 22uF.=

Voltage lines

The below is a screenshot of the 5V and the 3.3V line

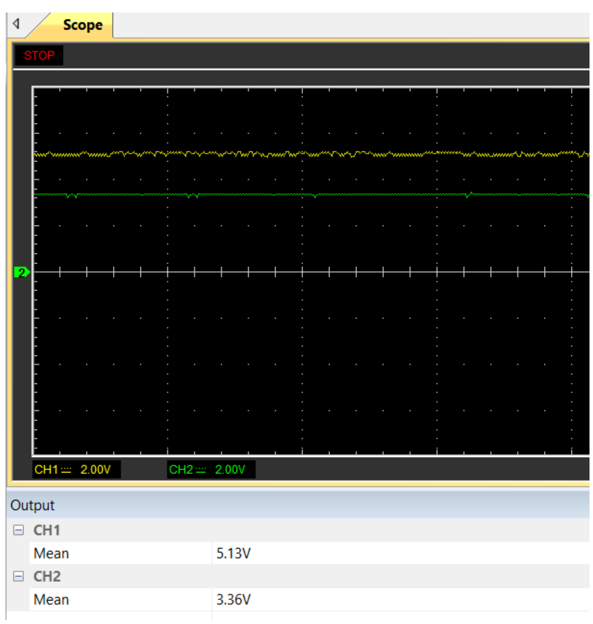

Figure 16 5v(yellow) and 3.3v (green)

As we can see, the line voltage is roughly 5 and the 3.3V regulator is operating as it should.

Noise comparison on the 5v line due to a slammer circuit

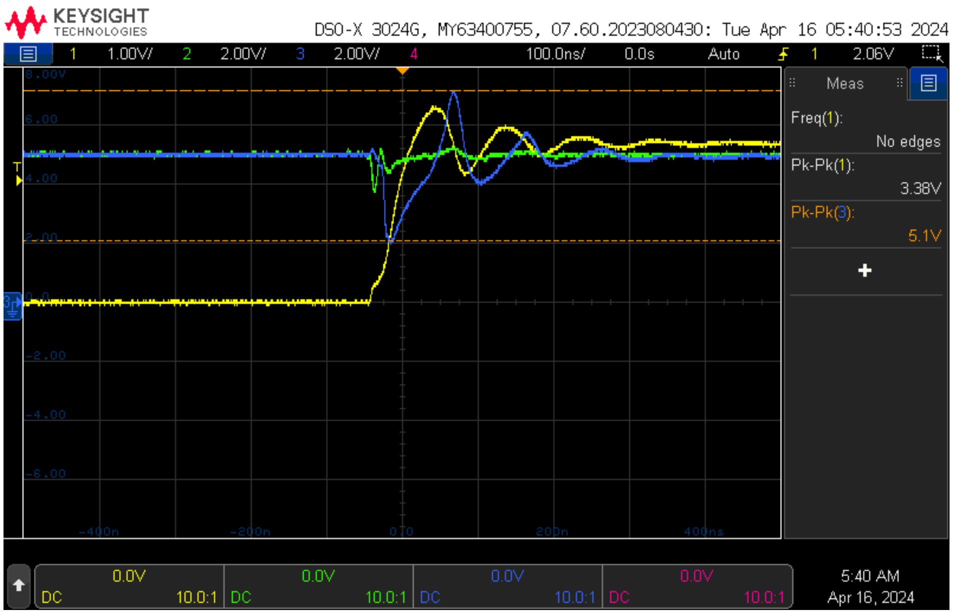

Figure 17 Golden PCB R1(yellow) Q_HI (green) and 5v (blue)

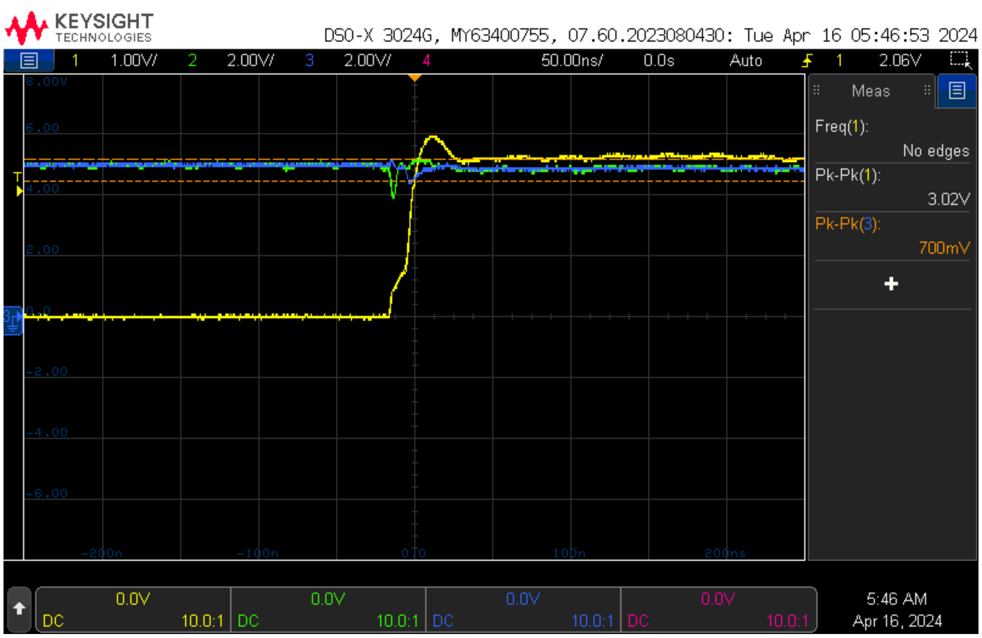

Figure 18 Commercial Arduino R1(yellow) Q_HI (green) and 5v (blue)

Noticeably here, in the golden PCB, the peak-to-peak noise is about 5 volts! This is much higher than that of the commercial Arduino which is 700mV. I asked TA's about this but they were unsure as to why this was happening. This I suppose could be due to the fact that I had to use jumper cables when connecting the shield due to improper layout of one of the header pins. Using jumper wires would lead to increased loop inductances leading to larger L terms and more noise since V = L di/dt. However, this is only part of the reason for such a high noise level. There was negligible difference in noise on the Quiet HI lines when all these were being measured though. Perhaps the artifacts of measurement introduced by three probes measuring at the same time could have something to do with it.

Inrush Current

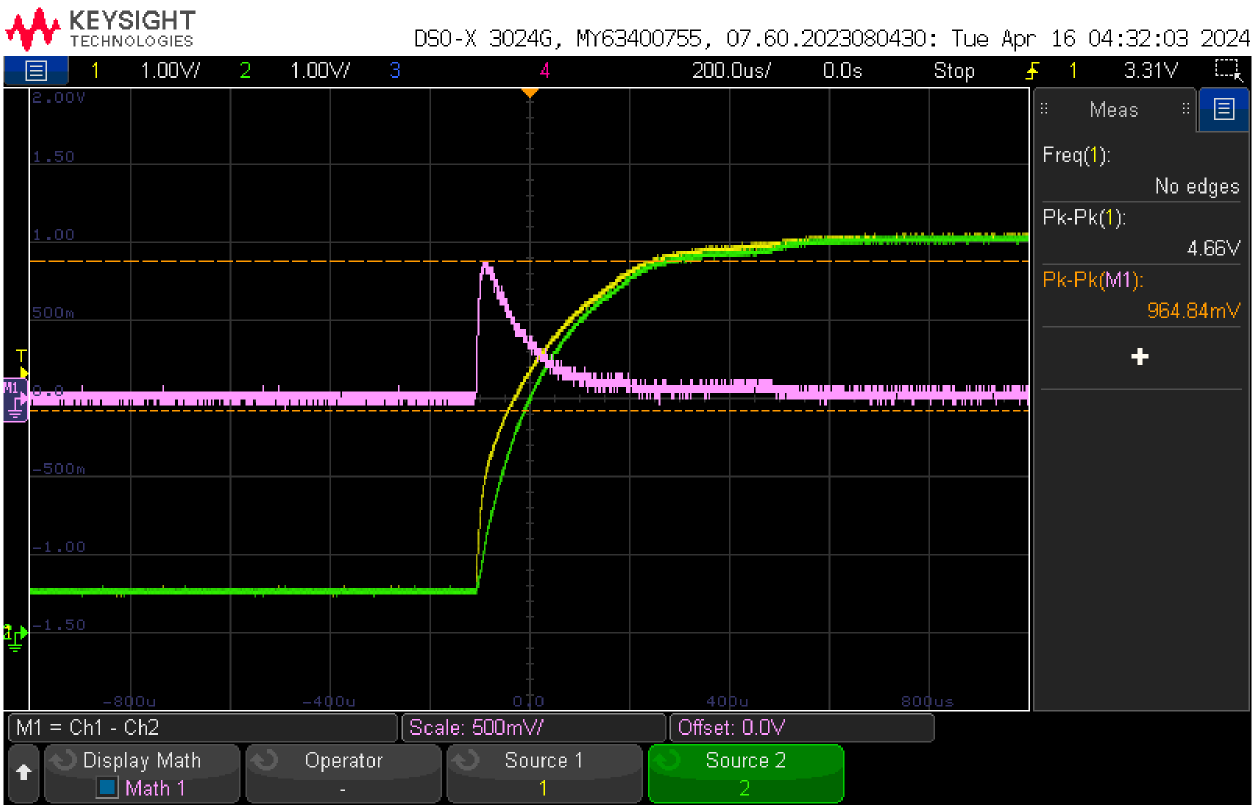

Figure 19 Inrush current for the Golden PCB

Using the 0.5-ohm resistor, we can calculate the inrush current of this board. Since we measure a peak voltage of 964.84mV. Using ohms law,

I = V/R = 964.84mV/0.5Ohms = 1.929 Amps.

Comparison of EMI noise

In comparing the EMI noise from the two board. Firstly, I moved the victim loop around the board to find out where would the most noise be induced. I found that it was when the victim loop was as close as possible to the switching pin. In comparing the two boards, the golden PCB had around 160mV peak to peak noise induced whereas in the same configuration, the commercial Arduino had close to 420mV peak to peak noise. This could be due to several factors. One could be that the material used in the commercial PCB is more prone to noise or emit more noise due to different process manufacturing techniques. Also we can see that the rise time of the signal in the commercial PCB is nearly half as short which leads to more mutually inductive coupling and hence a larger amount of voltage being induced.

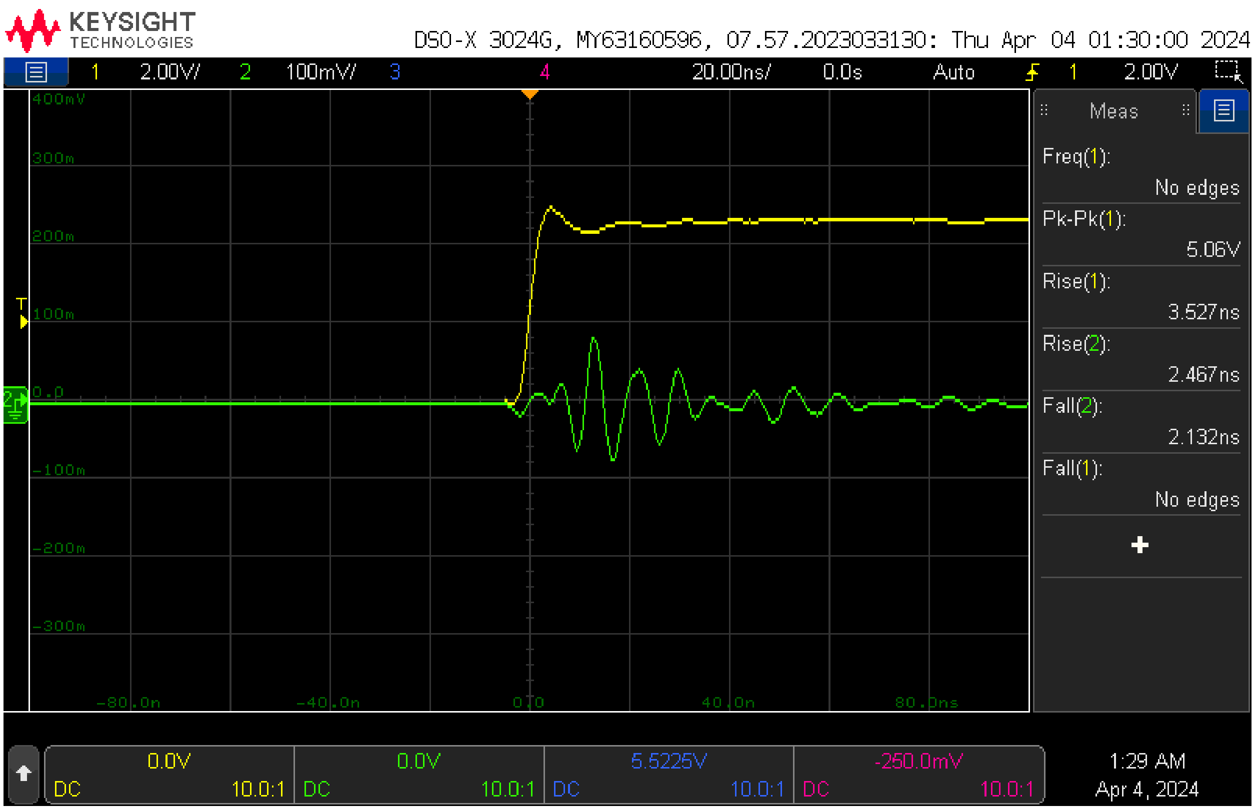

Figure 20 Golden PCB d13(yellow) victim loop (green)

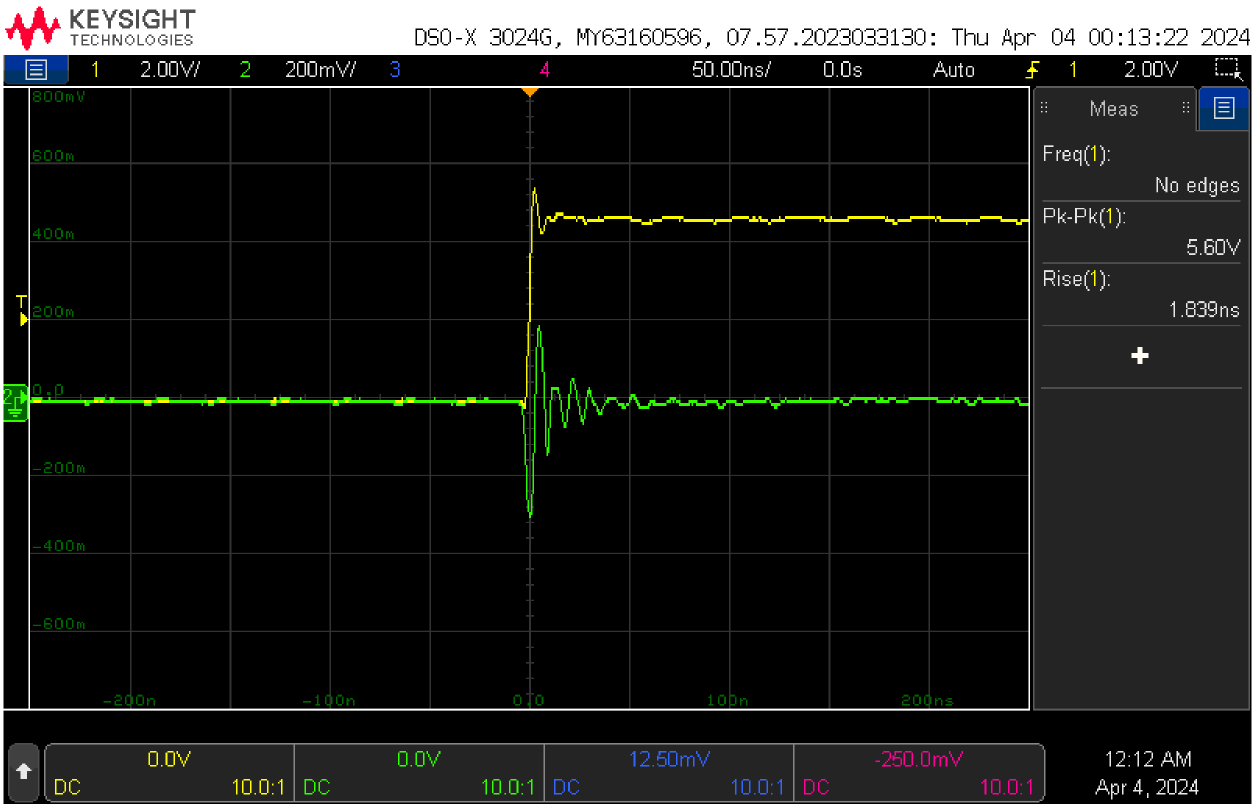

Figure 21 Commercial Arduino d13(yellow) victim loop (green)

The last thing to be talked about is the reset circuit which does indeed function as it should. The key here is that the series capacitor for the DTR line needed to be higher than that of the capacitor parallel to the MCU reset pin. Holding the reset button does indeed hold it in reset and of course when it is released, the MCU can be programmed from the Arduino IDE as normal.

Overall, we need to keep in mind that the commercial Arduino board are mass produced and are made to be as small as possible. Also, at its core, a different IC is used as it is a larger DIP package compared to the smaller surface mounted package we use. Since our components are more spread out and there are overall less components, this would lead to difference in the load capacitance on the power rails, pins etc. Differences in capacitance would lead to different levels of noise being induced.

In comparing the performance of the golden PCB designed to the commercial arduino, it is clear from the previous screenshots that the Golden PCB wins out in most categories. I could only imagine how much better the performance of the golden PCB would be if I did not have to use jumper cables.

Things which worked well

Things which worked well, and I would like to do in my future designs:

- Placing decoupling capacitors as close as possible to the IC as possible
- Using jumper switched to isolate parts of the board during bring up. This was great so I can verify each subsection's functionality before connecting the rest of the board.\
- When doing the layout, rotating the MCU several times and observing the rats nest, I was able to minimize the amount of cross-unders needed.
- Doing the schematic piece by piece was much more efficient , neater and made me feel much less overwhelmed as it was a circuit with more components than previously.
- Adding test points for the digital signals was useful to check to see if there was any issues in connectivity between the ch340 and the mcu

Soft Errors

Everything worked as intended for this design fortunately and there were no hard errors. The biggest error, which was the upside-down header pin, I personally considered hard but it did not affect the operation of the board so by definition it isn't a hard error but a soft one.

Other soft errors include.

- Not using more indicator LEDs. I thought it would just be unnescessary but it is helpful in certain areas like the RX and TX lines when they are active
- Not connecting an LED to PD7. This is useful since the example code for blinky has an LED connected there so you can quickly tell if it is working instead of connecting an external LED or using an oscilloscope.