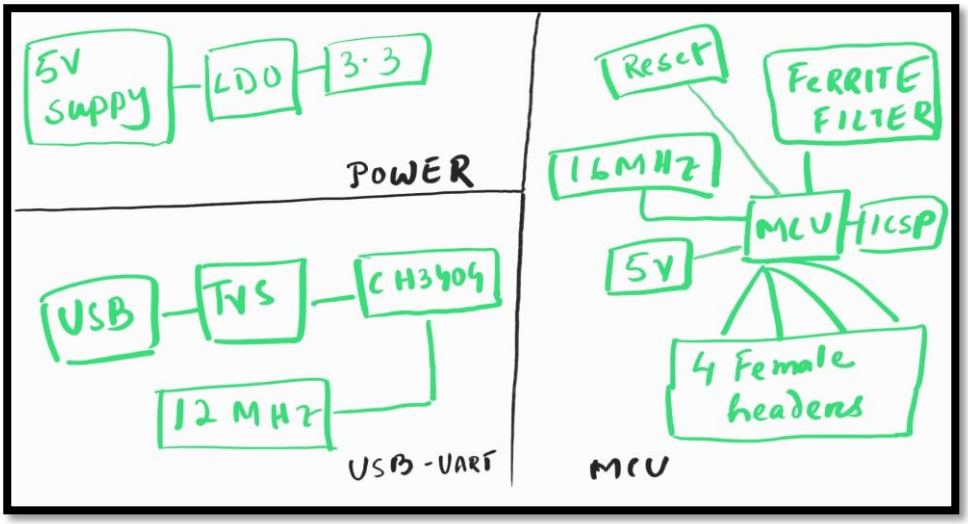
GOAL- BRING UP OF BOARD3

The seven steps for this Project- :

1. Completing the plan of record (POR)

- Some previous schematics were checked from the internet to get the glimpse of arduino schematics. However, taking the responsibility of my own design on which part is needed to add and which are not based on the consideration on my own requirements and also looked at the datasheet for each component and adding them based on the requirements. Some of the requirements are non-commodity such as the ATMEGA 328 and CH340G which was needed to look beforehand whether the parts were still available before inserting those on schematics and ordering later.
- Three separate functional parts were designed one for power functioning, one for USB to UART and the last one for microcontroller.
- Also some additional features we needed to add have been taken into consideration beforehand such as the Ferrite filter on AVCC pin, Current sense resistor in series with the power line, and one alternative power other than USB port.
- In addition to make sure of the risk reduction steps for the layout part and getting accurate results for the board some points were also noted beforehand like the indicator LEDs were connected to make sure that the boards is getting power after turning on, keeping the decoupling caps very close to the power pins of the ICs, labelling for pins and parts for better user understanding and correct footprints for 6, 8, 10 header sockets.



Rough Block Diagram before schematic design

2. Completing the preliminary bill of materials (BOM)

The list of materials including many commodity parts such as capacitors, resistors, LEDs, LDO and power connector, also non-commodity parts such as ATMEGA microcontroller, CH340G are included as the preliminary bill of materials.

3. Completing the final schematic capture and final BOM

Ultimately, the schematic was drawn with the planned approach along with certain changes wherever needed. Later the final BOM was taken from the schematic set itself.

4. Completing the board layout and order all the parts.

After the schematic, the board layout part was completed with better routing as possible and short traces of the cross under path connections to reduce the noise as much as possible.

5. Completing the assembly.

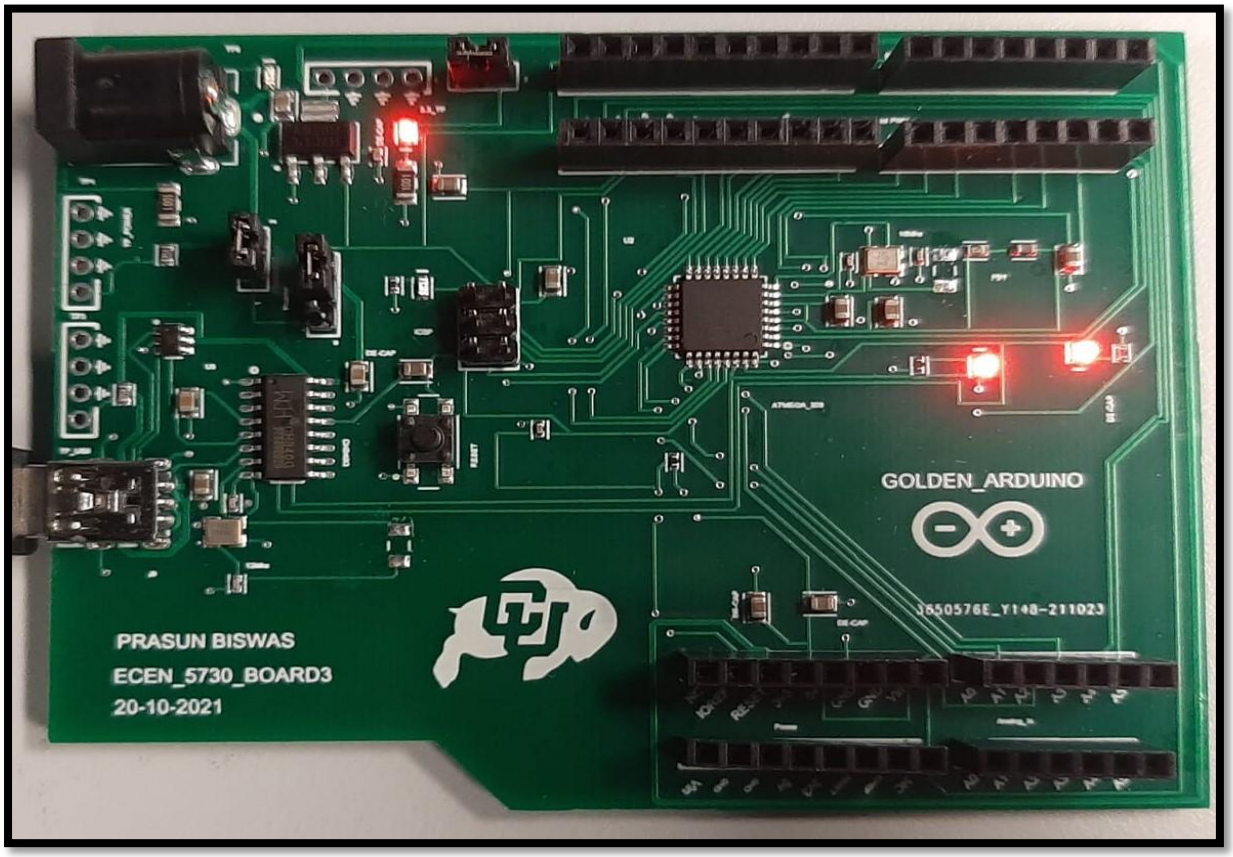
6. Completing the bring up, Troubleshoot, and final test.

Finally after the board was assembled with components we have tested the required testing areas along with the board such as the boot loading process being the first step and it was successfully done following the proper steps for boot loading. Furthermore, Blink code was also uploaded in the code to make sure that code is uploading and running fine on my board. There are some soft errors on my board which I found while testing and analyzed where it could be improved further in the designing process. (I have mentioned the recommended areas where it can be improved in the later part of this report).

7. Completing the documentation.

✓ Assembled board component list

Parts	Description	Designator	Footprint	LibRef	Quantity	LCSC Part#
22uF	MULTILAYER CERA	C1	0805_Passive_Capac	C_22uF_0805	1	C45783
22uF	MULTILAYER CERA	C2	0805_Passive_Capac	C_22uF_0805	1	C45783
10uF	10uF ±20% 6.3V X5R	C3	0402_Passive_Capac	C_10uF_0402	1	C15525
22uF	MULTILAYER CERA	C4	0805_Passive_Capac	C_22uF_0805	1	C45783
22pF	MULTILAYER CERA	C5	0402_Passive_Capac	C_22pF_0402	1	C1555
22pF	MULTILAYER CERA	C6	0402_Passive_Capac	C_22pF_0402	1	C1555
22uF	MULTILAYER CERA	C7	0805_Passive_Capac	C_22uF_0805	1	C45783
22uF	MULTILAYER CERA	C8	0805_Passive_Capac	C_22uF_0805	1	C45783
22uF	MULTILAYER CERA	C9	0805_Passive_Capac	C_22uF_0805	1	C45783
22uF	MULTILAYER CERA	C10	0805_Passive_Capac	C_22uF_0805	1	C45783
1uF	MULTILAYER CERA	C11	0402_Passive_Capac	C_1uF_0402	1	C52923
22pF	MULTILAYER CERA	C12	0402_Passive_Capac	C_22pF_0402	1	C1555
22pF	MULTILAYER CERA	C13	0402_Passive_Capac	C_22pF_0402	1	C1555
22uF	MULTILAYER CERA	C14	0805_Passive_Capac	C_22uF_0805	1	C45783
22uF	MULTILAYER CERA	C15	0805_Passive_Capac	C_22uF_0805	1	C45783
22uF	MULTILAYER CERA	C16	0805_Passive_Capac	C_22uF_0805	1	C45783
22uF	MULTILAYER CERA	C17	0805_Passive_Capac	C_22uF_0805	1	C45783
22uF	MULTILAYER CERA	C18	0805_Passive_Capac	C_22uF_0805	1	C45783
SRV05-4-P-T7	TVS SOT-23-6 ROHS	D1	TVS	D_TVS_Diode	1	C85364
FB_10uH_0603	10 uH, 30 Ω, 1.85 Ohm	FB1	0603_PASSIVE_Capa	FB_10uH_0603	1	C1035
NRPN032PAEN-RC	Connector Header Th	J1	ICSP	J_ICSP	1	C124384
PPTC101LFBN-RC	10 Position Header Co	J2	10_PIN_ARDUINO_10	JF_10_PIN_FEMALE_	1	C225507
PPTC101LFBN-RC	10 Position Header Co	J3	10_PIN_ARDUINO_10	JF_10_PIN_FEMALE_	1	C225507
PPTC081LFBN-RC	8 Position Header Cor	J4	8_PIN_ARDUINO_100	JF_8_PIN_FEMALE_	1	C27438
PPTC081LFBN-RC	8 Position Header Cor	J5	8_PIN_ARDUINO_100	JF_8_PIN_FEMALE_	1	C27438
PPTC061LFBN-RC	6 Position Header Cor	J6	6_PIN_ARDUINO_100	JF_6_PIN_FEMALE_	1	C40877
PPTC061LFBN-RC	6 Position Header Cor	J7	6_PIN_ARDUINO_100	JF_6_PIN_FEMALE_	1	C40877
3 PIN	3 2.54mm Straight 1 H	J8	3_PIN_100mil	J_3_PIN_HEADER	1	C49257
USB Mini	Through Hole USB C	J9	USB_MINI_B	J_USB_B_Mini	1	C46398
PPTC081LFBN-RC	8 Position Header Cor	J10	8_PIN_ARDUINO_100	JF_8_PIN_FEMALE_	1	C27438
PPTC081LFBN-RC	8 Position Header Cor	J11	8_PIN_ARDUINO_100	JF_8_PIN_FEMALE_	1	C27438
KT-0603R	LIGHT EMITTING DI	LED1	LED_0603	LED_RED_0603	1	C2286
KT-0603R	LIGHT EMITTING DI	LED2	LED_0603	LED_RED_0603	1	C2286
KT-0603R	LIGHT EMITTING DI	LED3	LED_0603	LED_RED_0603	1	C2286
KT-0603R	LIGHT EMITTING DI	LED4	LED_0603	LED_RED_0603	1	C2286
KT-0603R	LIGHT EMITTING DI	LED5	LED_0603	LED_RED_0603	1	C2286
Power Jack	Power Barrel Connec	P1	Power_Jack	P_Power Jack	1	C136744
500m	CHIP RESISTOR - S	R1	0603_PASSIVE_Resi	R_500m_0603	1	C45312
1k	CHIP RESISTOR - S	R2	1206_Passive_Resitor	R_1K_1206	1	C4410
1k	CHIP RESISTOR - S	R3	1206_Passive_Resitor	R_1K_1206	1	C4410
1k	1 kOhms chip resistor	R4	0402_Passive_Resist	R_1K_0402	1	C11702
10k	10 kOhms ±5% 0.25W	R5	1206_Passive_Resitor	R_10KOhm_1206	1	C17902
1k	1 kOhms chip resistor	R6	0402_Passive_Resist	R_1K_0402	1	C11702
10k	CHIP RESISTOR - S	R7	0402_Passive_Resist	R_10KOhm_0402	1	C25744
10k	10 kOhms ±5% 0.25W	R8	1206_Passive_Resitor	R_10KOhm_1206	1	C17902
1k	1 kOhms chip resistor	R9	0402_Passive_Resist	R_1K_0402	1	C11702
500m	CHIP RESISTOR - S	R10	0603_PASSIVE_Resi	R_500m_0603	1	C45312
500m	CHIP RESISTOR - S	R11	0603_PASSIVE_Resi	R_500m_0603	1	C45312
on_off_switch	2Pin Header	SW1	2Pin-Header-Conn	SW_2Pin_100mil_Sw	1	C124375
on_off_switch	2Pin Header	SW2	2Pin-Header-Conn	SW_2Pin_100mil_Sw	1	C124375
TL3305AF160QG	Tactile Switch SPST-	SW3	Switch_JLC	SW_Push_Button_JL	1	C174049
TP_5V	Test Point 300 mil cer	TP1	TP10x_Probe	TP_10x_Probe	1	
TP_5V	Test Point 300 mil cer	TP2	TP10x_Probe	TP_10x_Probe	1	
TP_5V	Test Point 300 mil cer	TP3	TP10x_Probe	TP_10x_Probe	1	
AMS1117-3.3	LOW DROPOUT RE	U1	LDO	PS_AMS_LDO_3v3	1	C6186
ATMEGA328P-ANR	AVR AVR® ATmega	U2	ATMega328P	U_ATMega328P	1	C14877
CH340G	Transceiver USB 2.0	U3	USB to serial	U_USBtoSerial	1	C14267
X322516MLB4SI	±10ppm 16000000Hz	Y1	Crystal resonator	X_16Mhz_Res	1	C13738
X322512MSB4SI	SMD Crystal Resonat	Y2	Crystal resonator	X_12MHz_Res	1	C9002

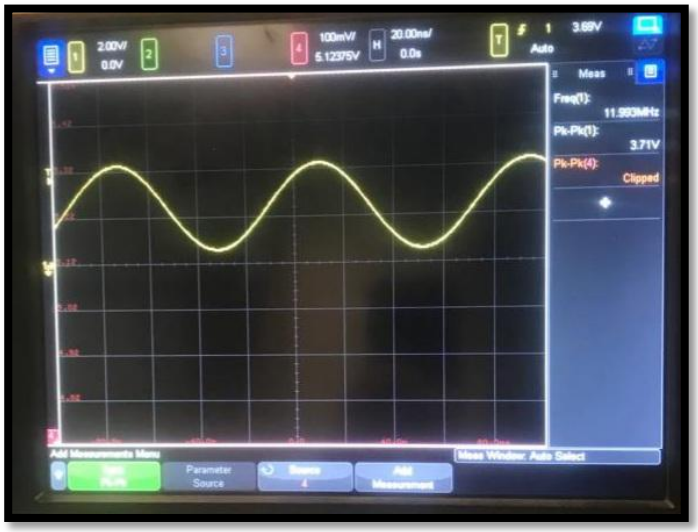


Assembled board-

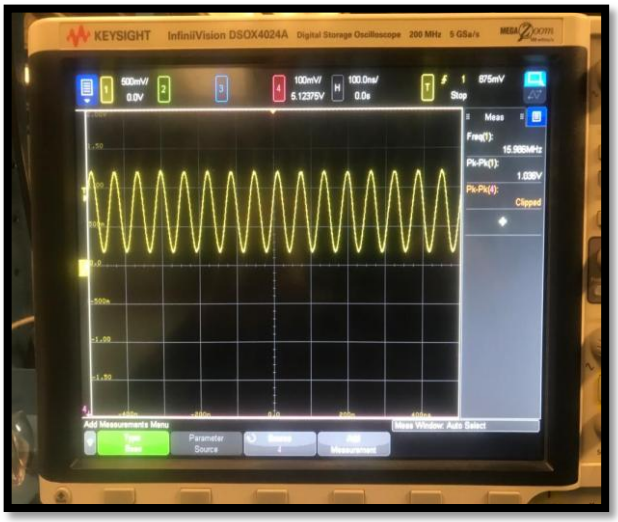
- **In case of functioning of the board-**

✓ At first the communication with USB was not happening as there was a resistor in parallel to both the oscillator which was preventing it to oscillate, later the resistor was removed and the communication over the USB to the CH340G chip was successful which has been verified with the RX TX LEDs on the Board while communicating and finally uploading a simple sketch of blink.

✓ Moreover it was also verified that there was no shorts on the power-ground rails on the bare board, and the crystals were also functioning as shown:

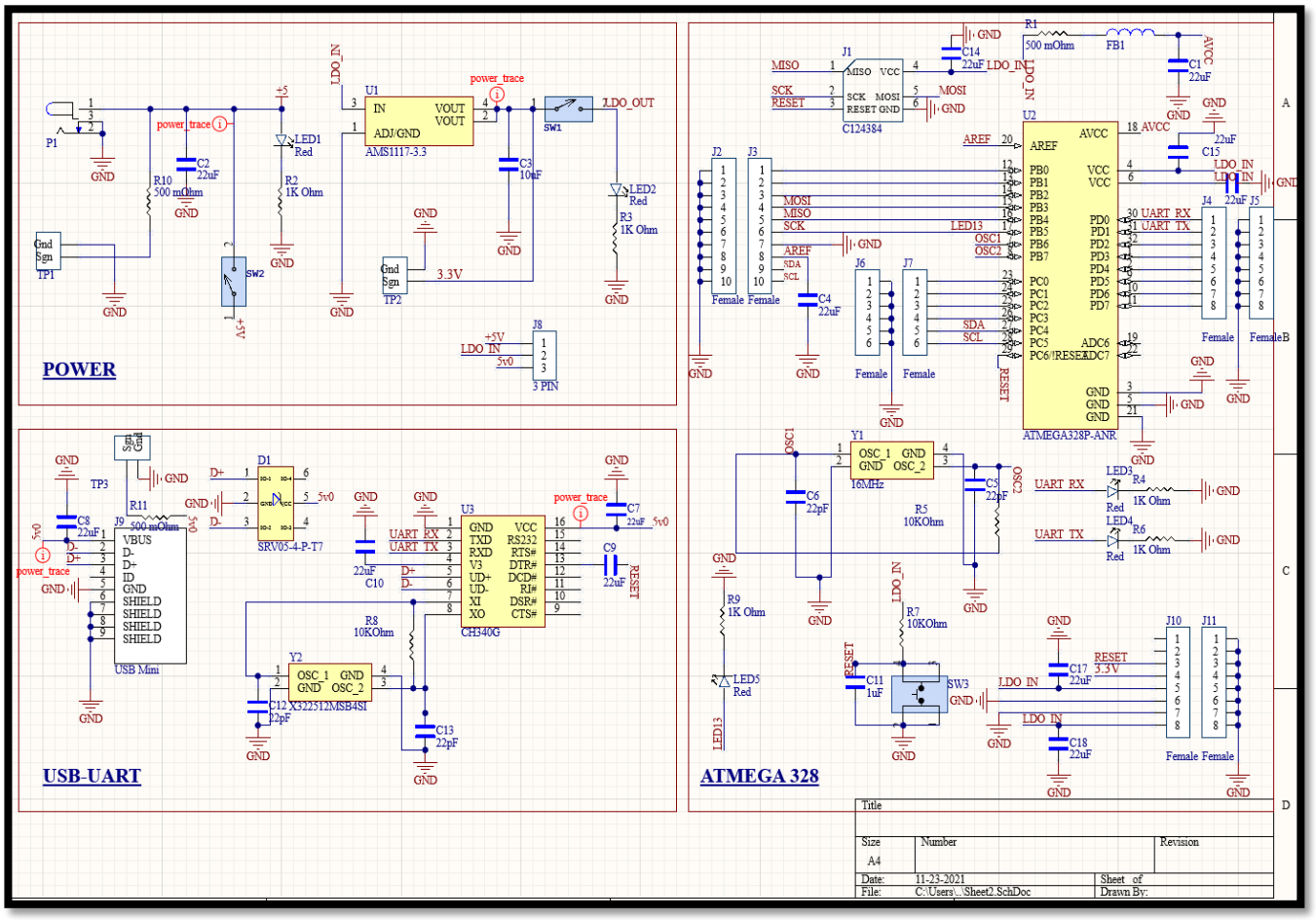


12 MHz crystal

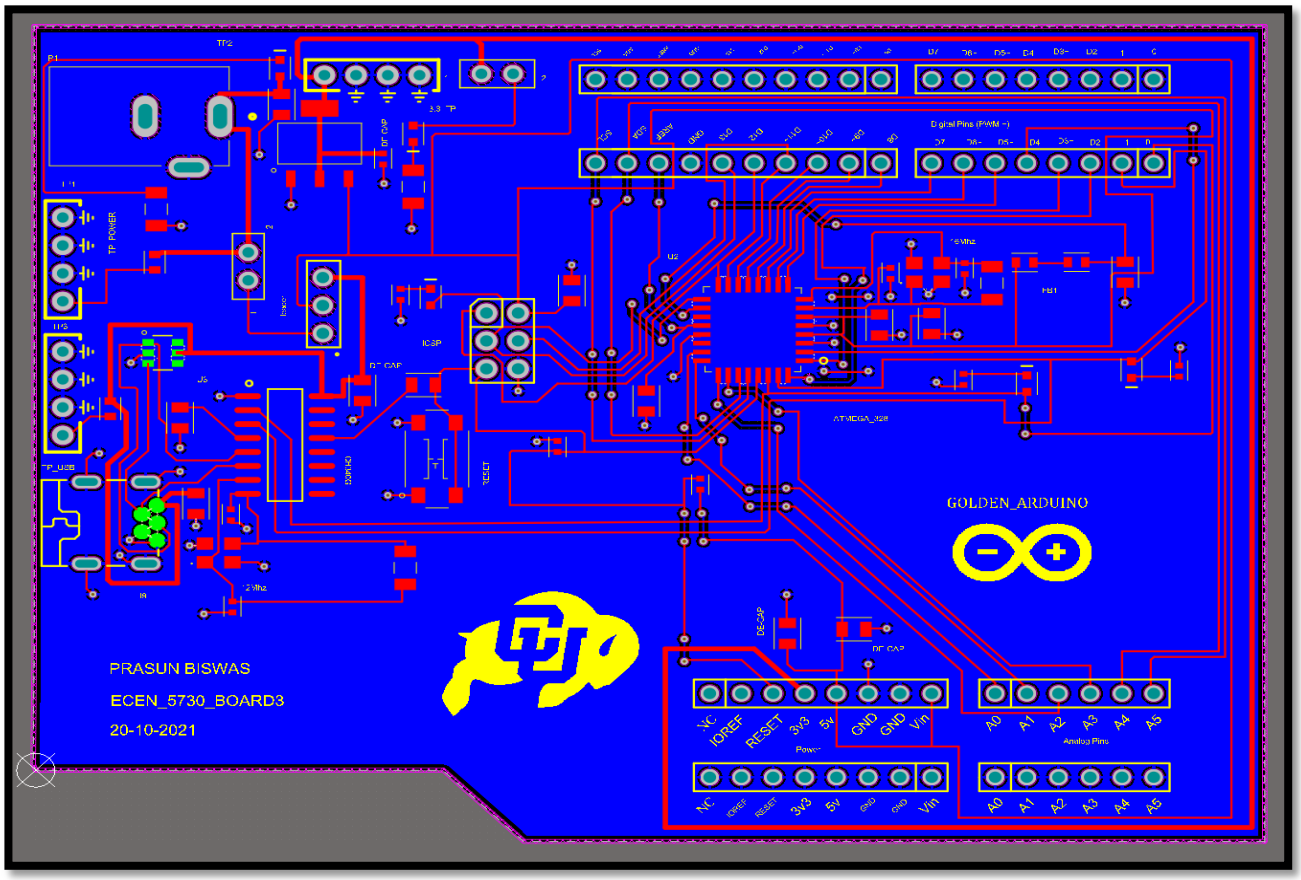


16 MHz crystal

SCHEMATIC

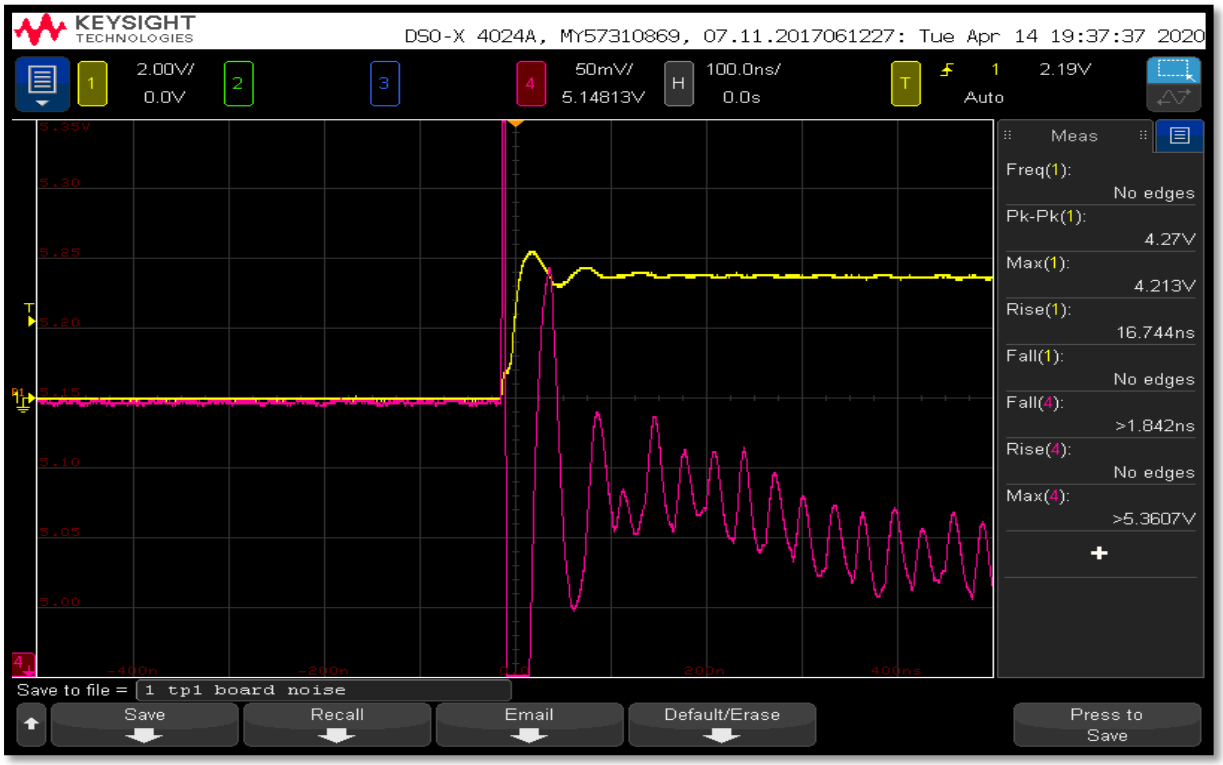


Layout-



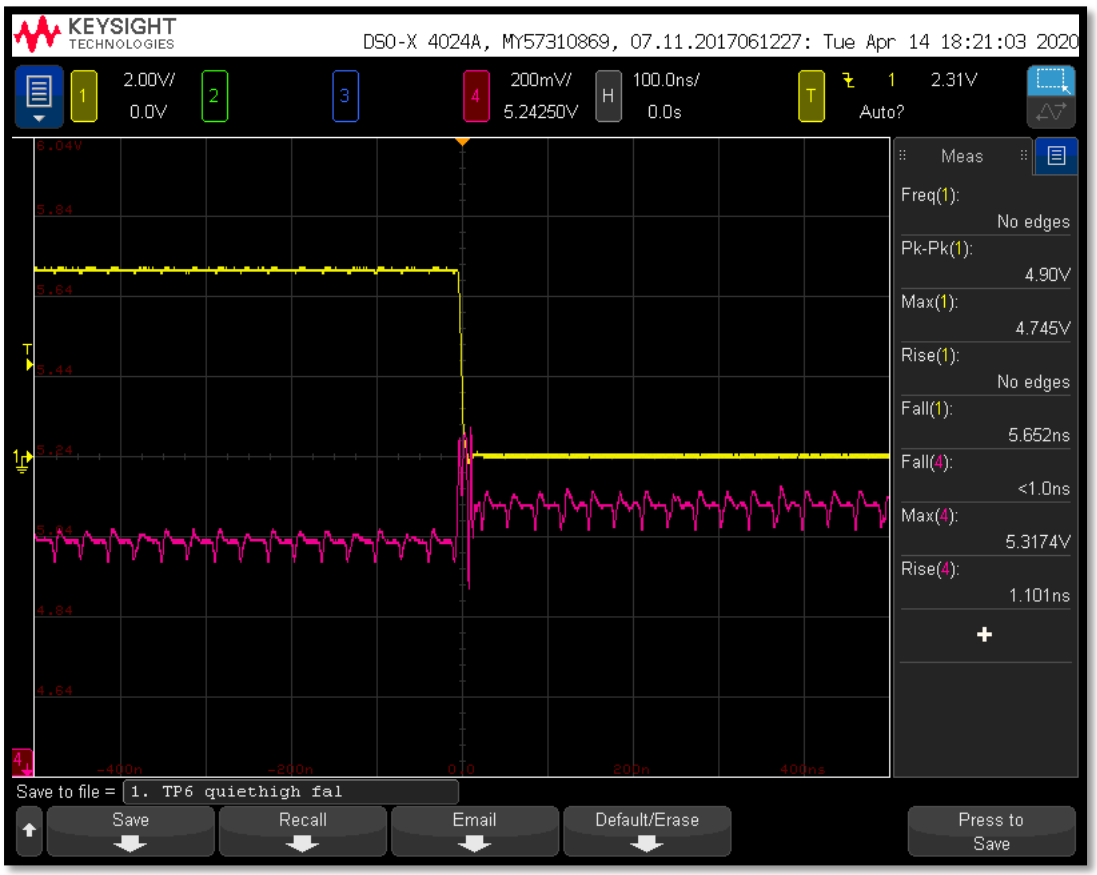
➢ Measurements on My arduino Board and then comparison with the Commercial Board

✓ **TP1- when I/O switching was there for my arduino board**



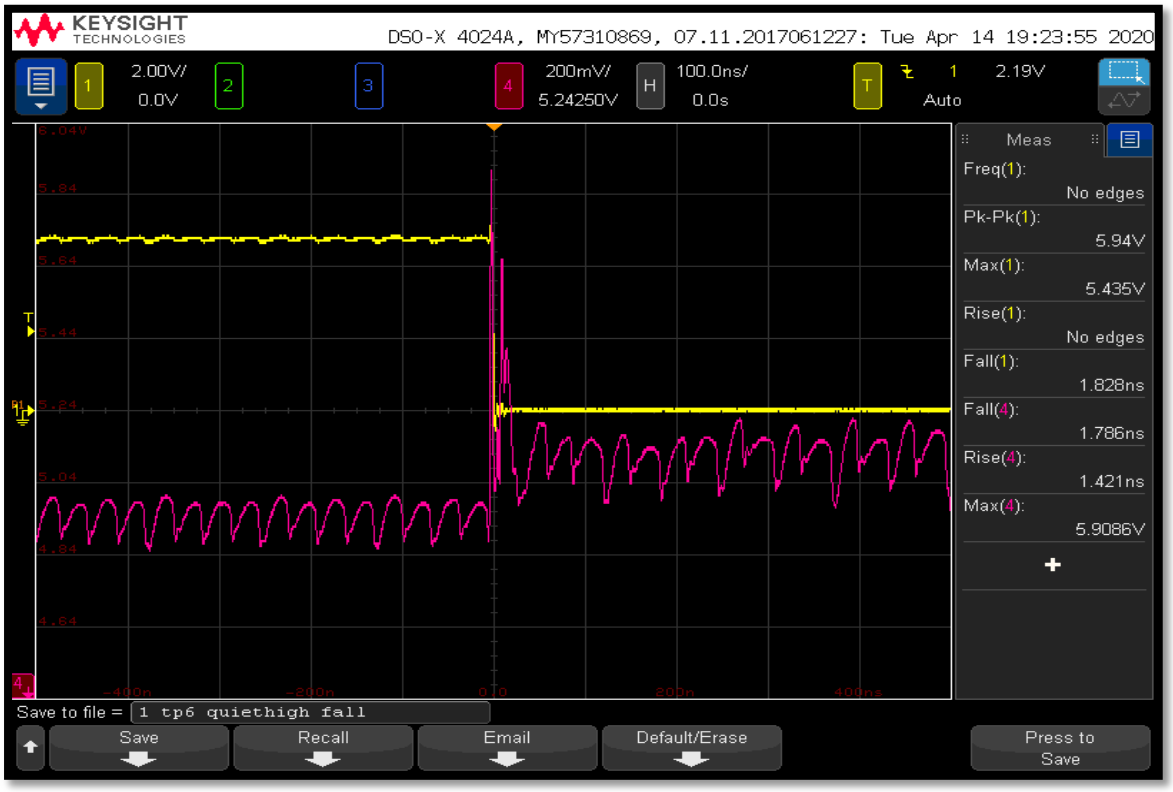
✓ **TP1- when I/O switching was there for my commercial board**

➢ With comparison to my board and commercial board for TP1 board noise when I/O switching was present, The self Aggressor noise on TP1 for my board is comparatively much less than what was coming for the commercial one because of the decoupling capacitor placement on my board which was reducing the noise much better coming from the die to the board.



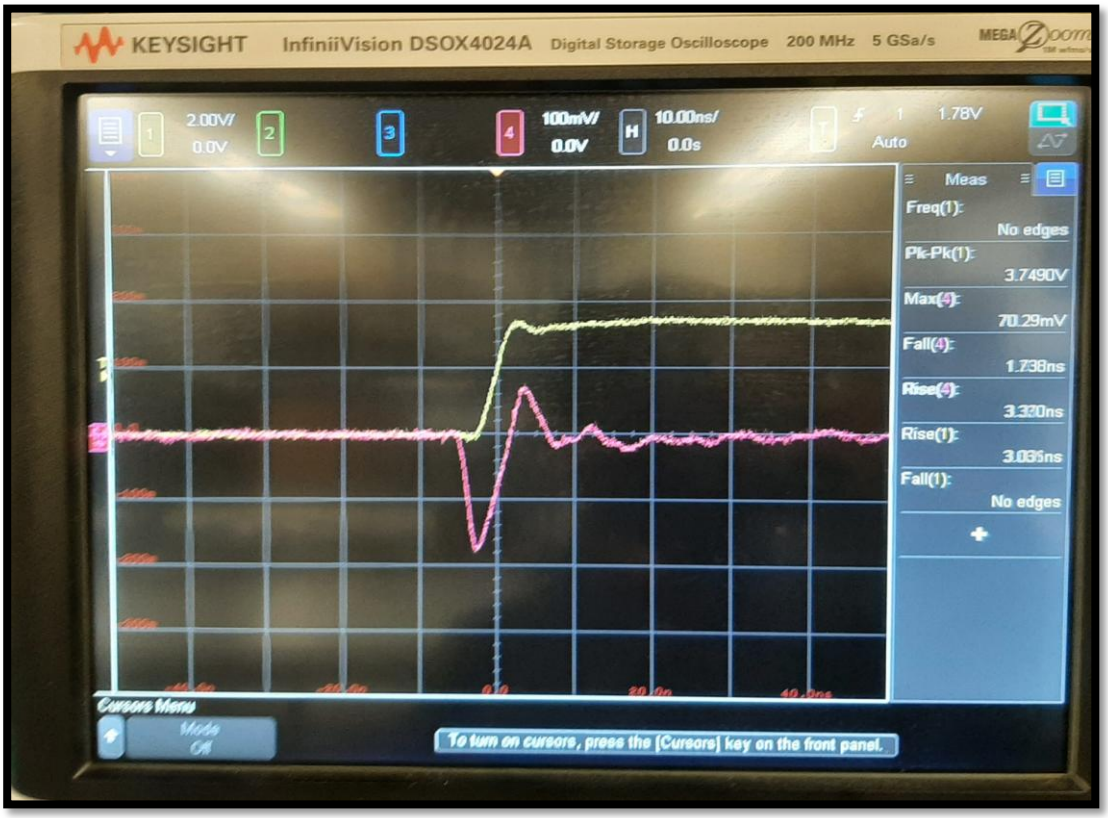
TP6 QIUET HIGH Noise when I/O was switching on my board-:

Here the fall time is less than 1 ns



TP6 QIUET HIGH Noise when I/O was switching on commercial board-:

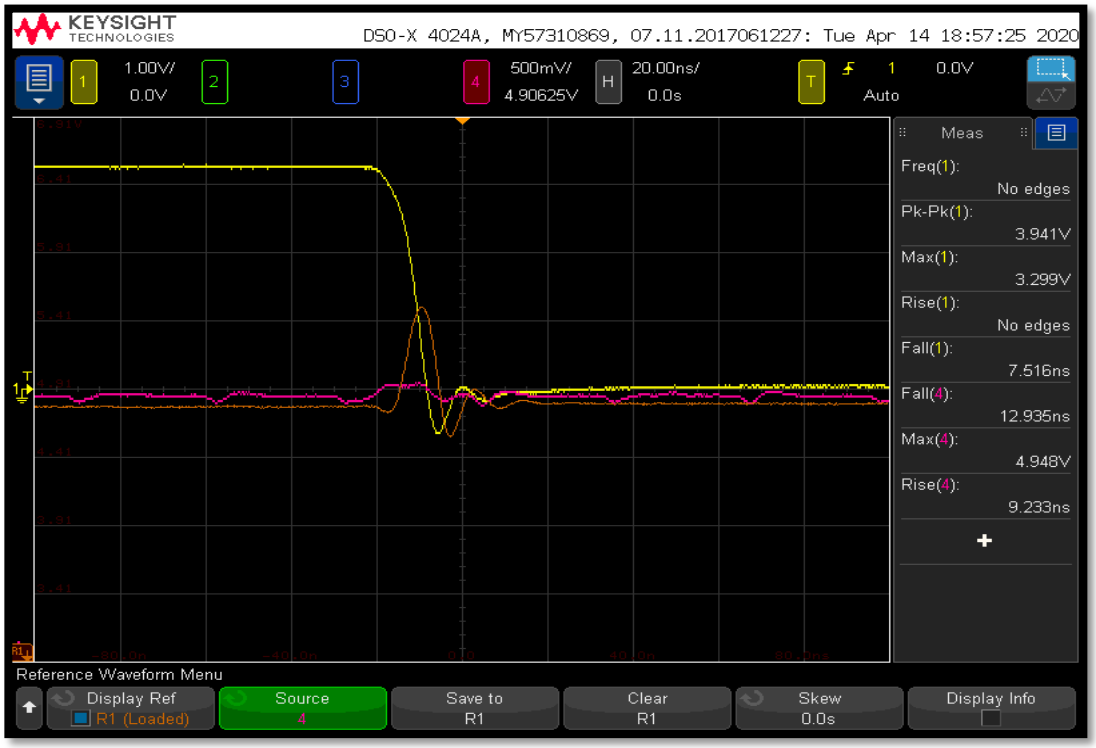
Fall Time- 1.786 ns



TP5 QUIET LOW Noise when I/O was switching on my board

TP5 QUIET LOW Noise when I/O was switching on commercial board

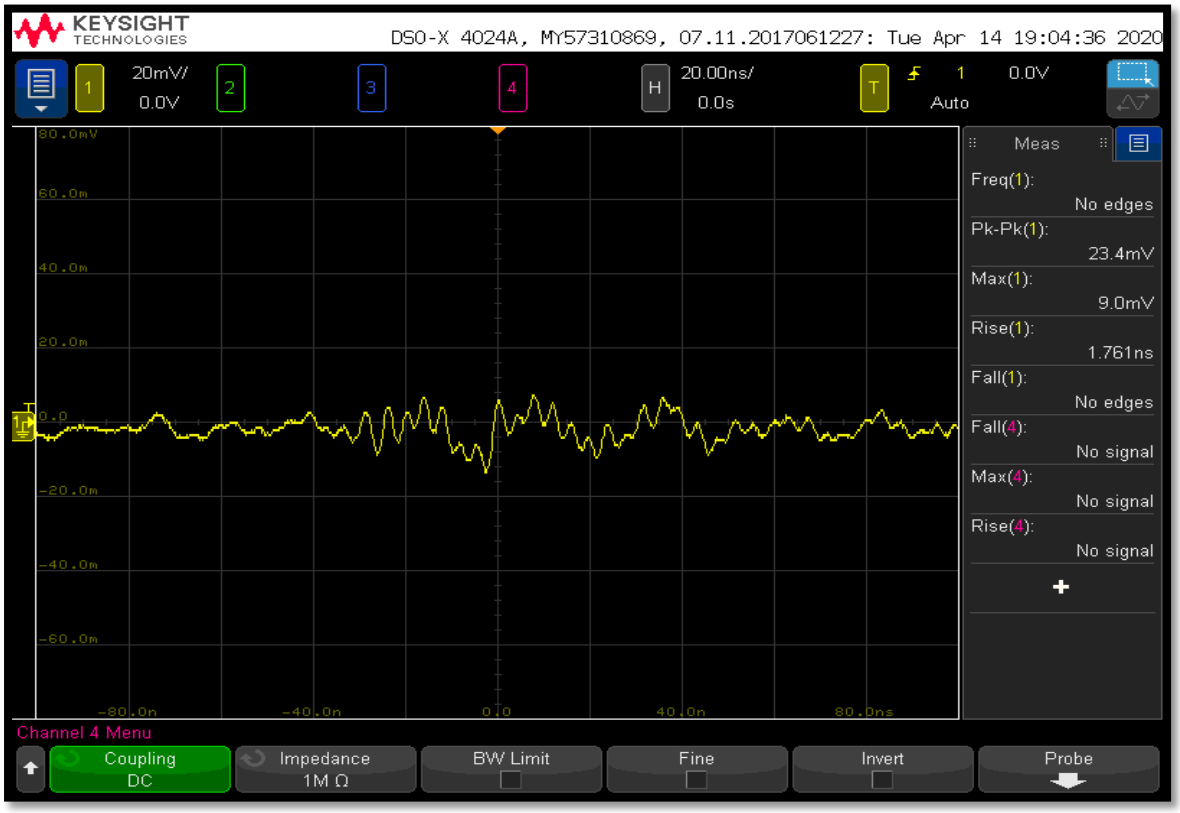
- With comparison to my board and commercial board for quiet low signal when I/O switching was there, the ground bounce noise was around twice as much for commercial board compared to my board.



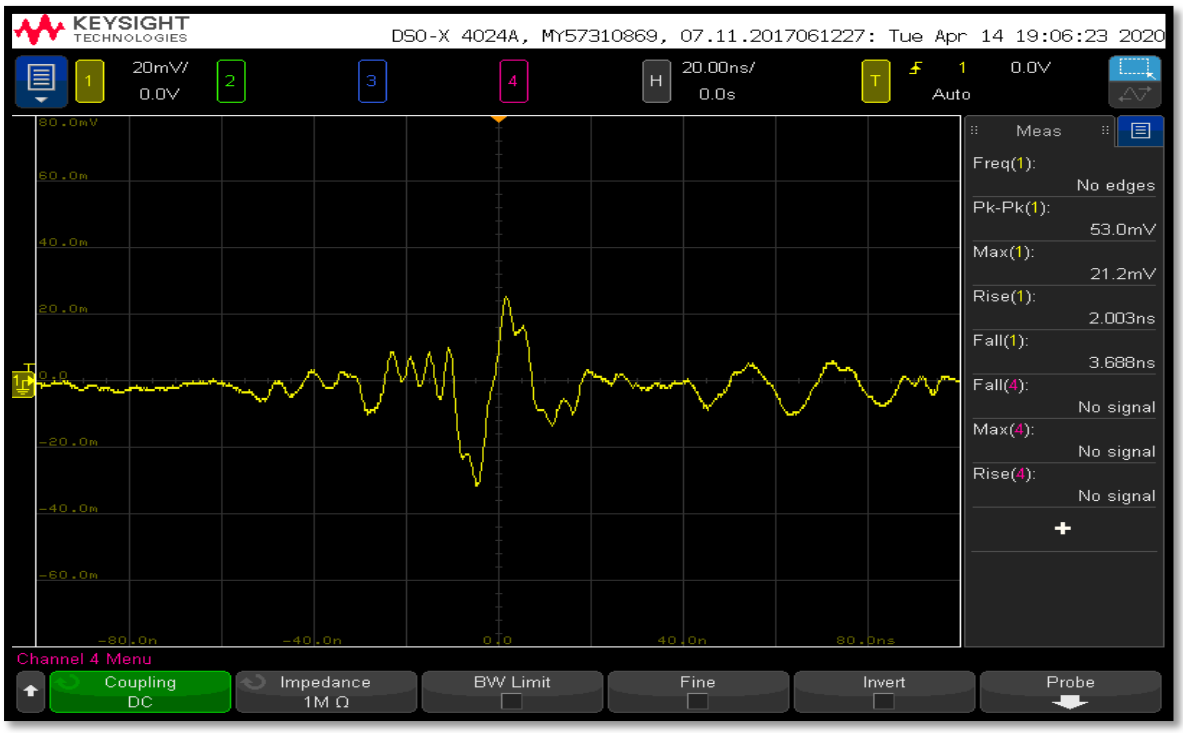
Slammer circuit voltage noise on 5V rail and on the quiet high pin for my board

Slammer circuit voltage noise on 5V rail and on the quiet high pin for commercial board

- With comparison to my board and commercial board for voltage noise on 5V rail to quiet high pin on the die was comparatively less, because the high frequency noise from the board is being filtered out by the decoupling capacitor. Moreover, the noise on the board for commercial one was also very high (~2) due to the high inductance on the board rail.



Near Emitting noise on my board which has a peak to peak voltage of around 23.4 mV



Near Emitting noise on commercial board which has a peak to peak voltage of 53mV

- With comparison to my board and commercial arduino for near emitting noise, it was much less (23mV) to that of in commercial one (53mV) because of the presence of less cross under traces as a result which the radiation will also be much less compared to commercial one.

Emphasizing on what worked for me and what was a mistake that could be done better the next time.

- In this design I have made some soft error such as giving the name of the pins opposite on one die with respect to what should be given originally so the naming would be a consideration next time which will decrease the confusion of connecting wires because of this soft error even though the functionality or the connection internally is correct.

- Moreover, I missed the correct dimension of my board with respect to commercial arduino UNO for the power rail female headers by a few mils which is why I had to struggle a bit to push the noise shield onto it and take the measurements. So I should keep this in mind if using any shield or sensor board on the arduino so that it gets shielded correctly.

- In addition, I have added the resistor in parallel with both the 12 and 16 MHz oscillators, but unfortunately it was not oscillating and the CH340 chip was not running as it should be. So I might consider keeping the connection open for resistor and check if it oscillates else I can solder 10k resistor there later.

- Furthermore, I feel that I have the board size a bit bigger than usual; I could have done that routing between the parts with a smaller board size as well. Also I need to have planning beforehand to start routing when it comes to so many components as many cross-under are used which could be neglected with proper planning for routing and placing of components beforehand.