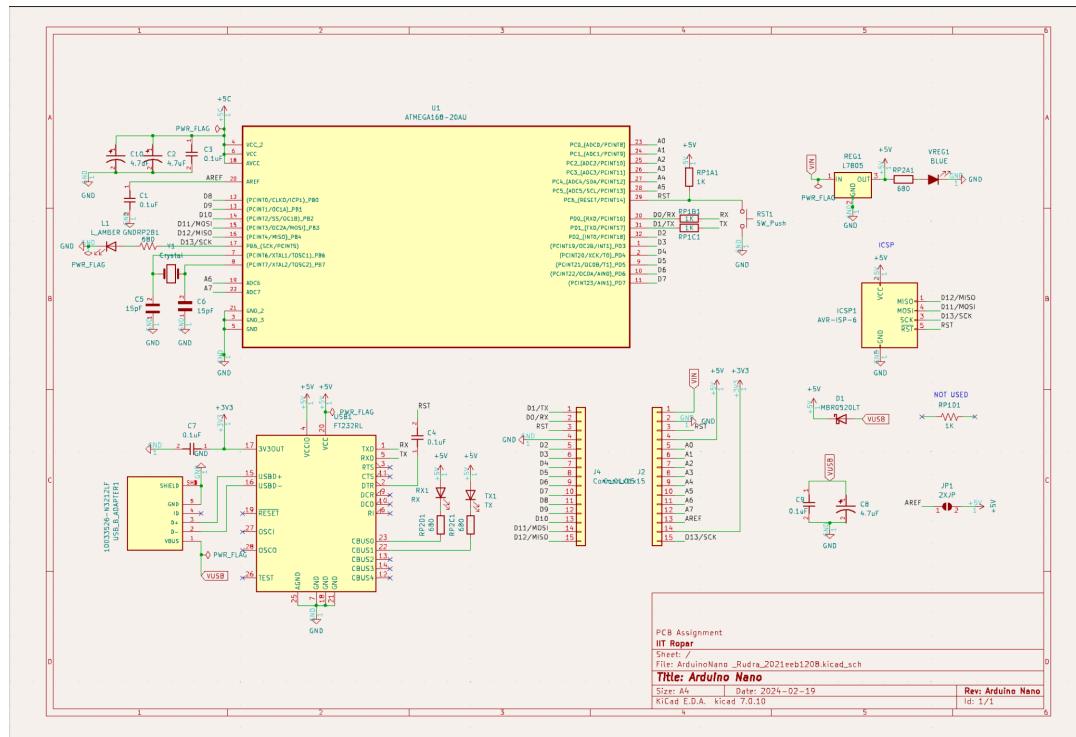


DESIGNING ARDUINO NANO!

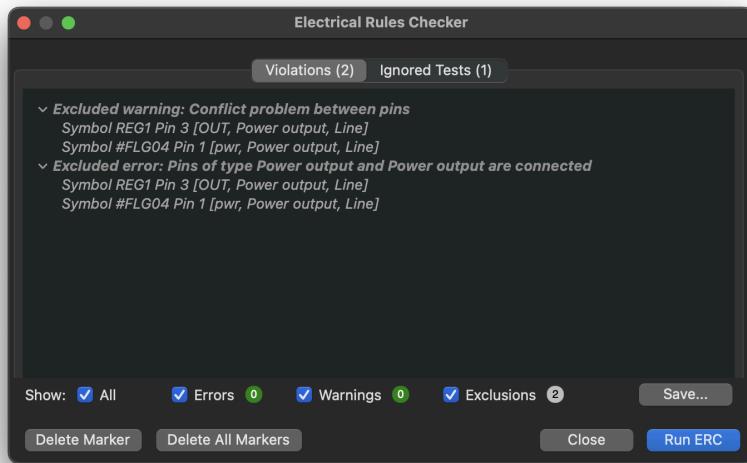
Optimisation:

1. Efforts were made to reduce track lengths in order to mitigate the expenses associated with fabrication.
2. Inner copper layers were incorporated to alleviate congestion in the top and bottom layers.
3. The dimensions of the board surpassed the necessary measurements due to initial hands-on experimentation.
4. The utilisation of drill holes facilitated the routing of inner layers, thereby optimising space allocation and track length.

Schematic and Footprint:



Schematic of Arduino Nano Board



Electrical Requirement Check (or ERC)

Symbol : Footprint Assignments	
1	C1 - 0.1uF : Capacitor_Tantalum_SMD:CP_EIA-2012-12_Kemet-R
2	C2 - 4.7uF : Capacitor_Tantalum_SMD:CP_EIA-2012-12_Kemet-R
3	C3 - 0.1uF : Capacitor_Tantalum_SMD:CP_EIA-2012-12_Kemet-R
4	C4 - 0.1uF : Capacitor_Tantalum_SMD:CP_EIA-2012-12_Kemet-R
5	C5 - 15pF : Capacitor_Tantalum_SMD:CP_EIA-2012-12_Kemet-R
6	C6 - 15pF : Capacitor_Tantalum_SMD:CP_EIA-2012-12_Kemet-R
7	C7 - 0.1uF : Capacitor_Tantalum_SMD:CP_EIA-2012-12_Kemet-R
8	C8 - 4.7uF : Capacitor_Tantalum_SMD:CP_EIA-2012-12_Kemet-R
9	C9 - 0.1uF : Capacitor_Tantalum_SMD:CP_EIA-2012-12_Kemet-R
10	C10 - 4.7uF : Capacitor_Tantalum_SMD:CP_EIA-2012-12_Kemet-R
11	D1 - MBR0520LT : Diode_SMD:D_SOD-123
12	ICSP1 - AVR-ISP-6 : Connector_PinHeader_2.54mm:PinHeader_2x03_P2.54mm_Vertical
13	J2 - Conn_01x15 : Connector_PinHeader_2.54mm:PinHeader_1x15_P2.54mm_Vertical
14	J4 - Conn_01x15 : Connector_PinHeader_2.54mm:PinHeader_1x15_P2.54mm_Vertical
15	JP1 - 2XJP : Jumper:SolderJumper-2_P1.3mm_Open_Pad1.0x1.5mm
16	L1 - L_AMBER : LED_SMD:LED_0805_2012Metric
17	REG1 - L7805 : Package_TO_SOT_SMD:SOT-223-3_TabPin2
18	RPIA1 - 1K : Resistor_SMD:R_0805_2012Metric
19	RP1B1 - 1K : Resistor_SMD:R_0805_2012Metric
20	RP1C1 - 1K : Resistor_SMD:R_0805_2012Metric
21	RP1D1 - 1K : Resistor_SMD:R_0805_2012Metric
22	RP2A1 - 680 : Resistor_SMD:R_0805_2012Metric
23	RP2B1 - 680 : Resistor_SMD:R_0805_2012Metric
24	RP2C1 - 680 : Resistor_SMD:R_0805_2012Metric
25	RP2D1 - 680 : Resistor_SMD:R_0805_2012Metric
26	RST1 - SW_Push : Button_Switch_SMD:SW_SPST_B3U-1000P-B
27	RX1 - RX : LED_SMD:LED_0805_2012Metric
28	TX1 - TX : LED_SMD:LED_0805_2012Metric
29	U1 - ATMEGA168-20AU : ATMEGA168-20AU:QFP80P900X900X120-32N
30	USB1 - FT232RL : Package_SO:SSOP-28_5.3x10.2mm_P0.65mm
31	USB_B_ADAPTER1 - 10033526-N3212LF : 10033526-N3212LF:AMPHENOL_10033526-N3212LF
32	VREG1 - BLUE : LED_SMD:LED_0805_2012Metric
33	Y1 - Crystal : Crystal:Crystal_HC49-4H_Vertical

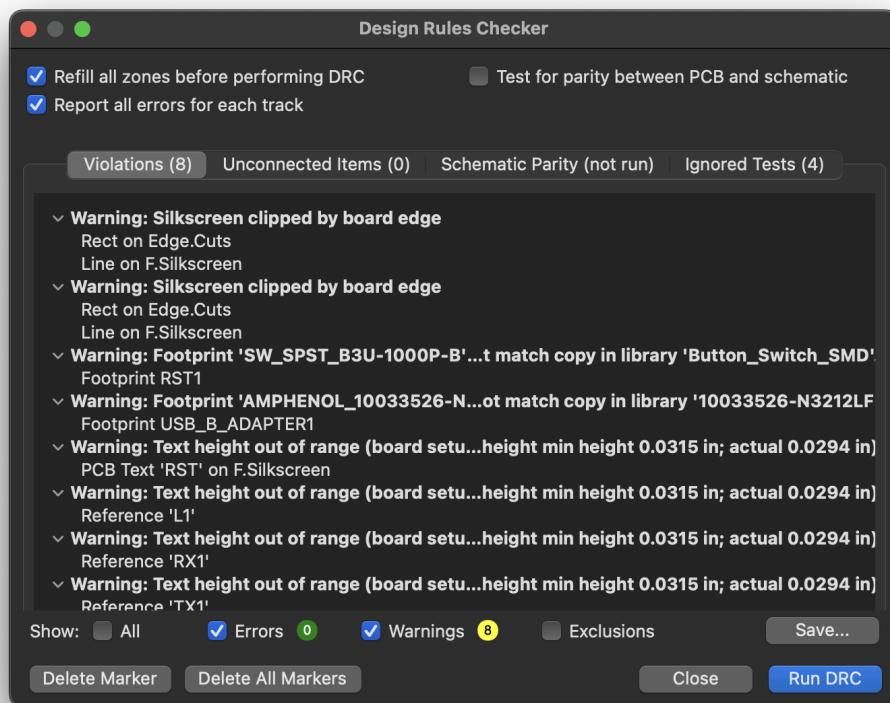
Footprint Assignment of Components

Note: Following instructions from Nikhil sir and based on my case, I have opted for the standard set of resistors and capacitors available in the default footprint library. All other components have been selected from the library according to provided specifications, ensuring compatibility with the provided or expected datasheet requirements. He also advised me to optimise the design to the fullest extent possible until no further improvements are feasible.

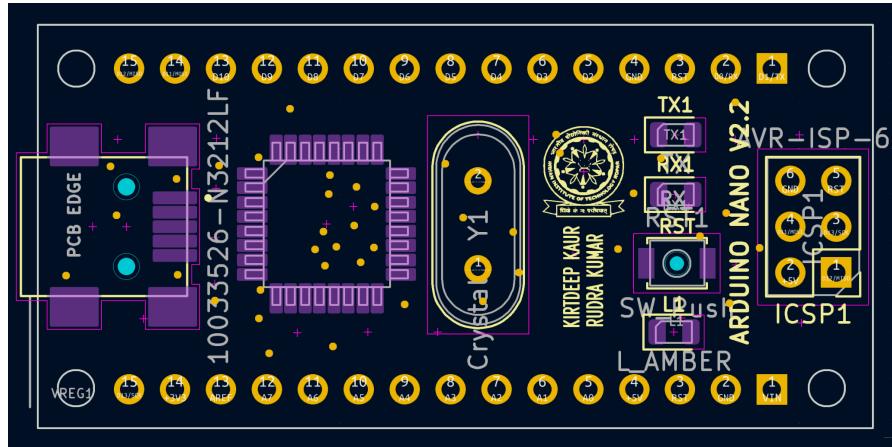
Footprints of the following were externally added by me: Diode, ATMEGA168-20AU chip, USB B adapter.

Note: In the process of Electrical Rules Check (ERC), errors related to power flags have been intentionally excluded. This exclusion has been duly considered, resulting in the absence of any errors or warnings in footprint assignments and connections. During the Design Rule Check (DRC) process, violations related to the silkscreen can be disregarded due to optimization constraints on the board dimensions.

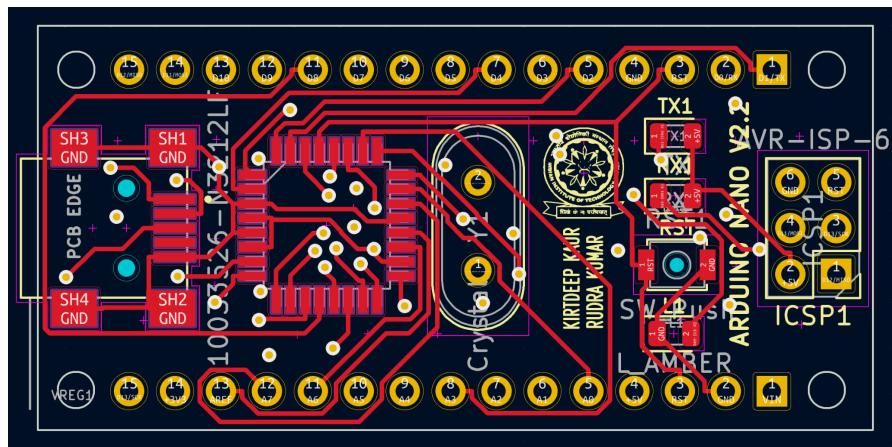
PCB Layout:



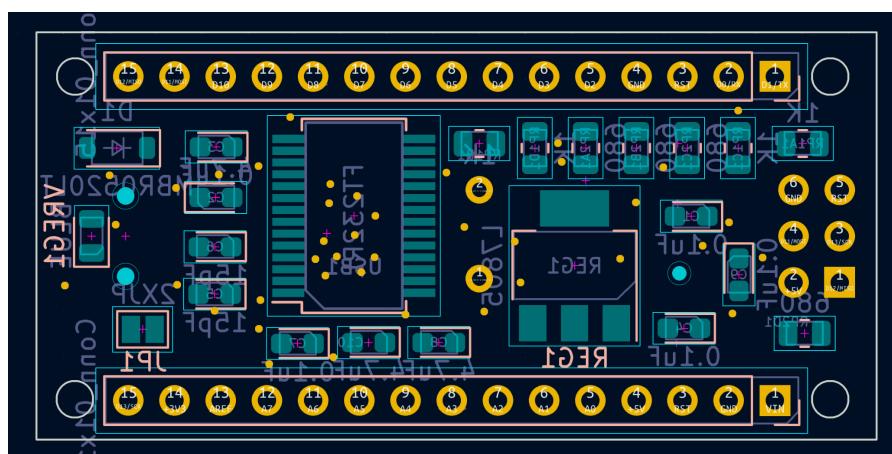
Design Requirement Check (or DRC)



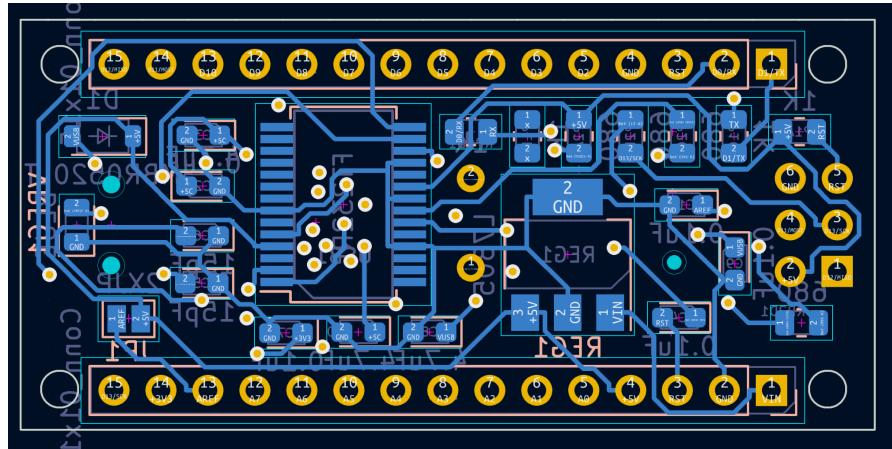
Front Assembly View



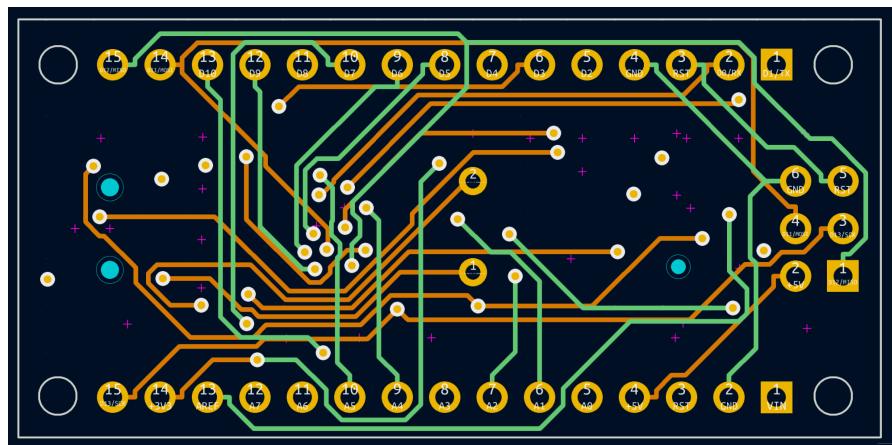
Front Layer



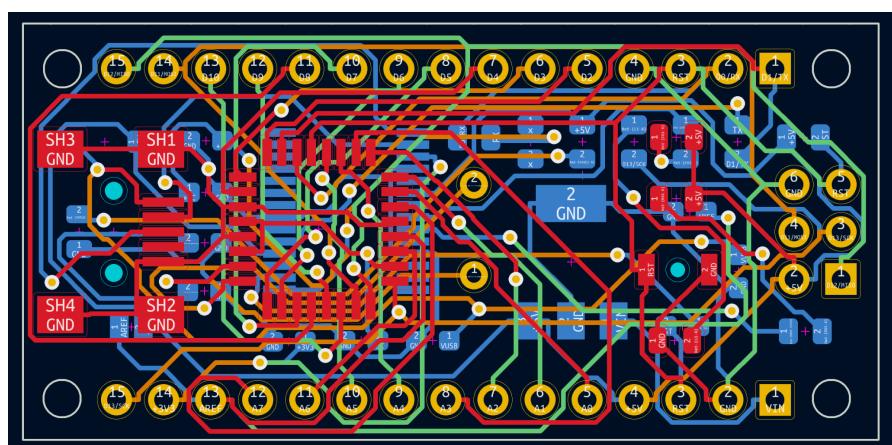
Back Assembly View



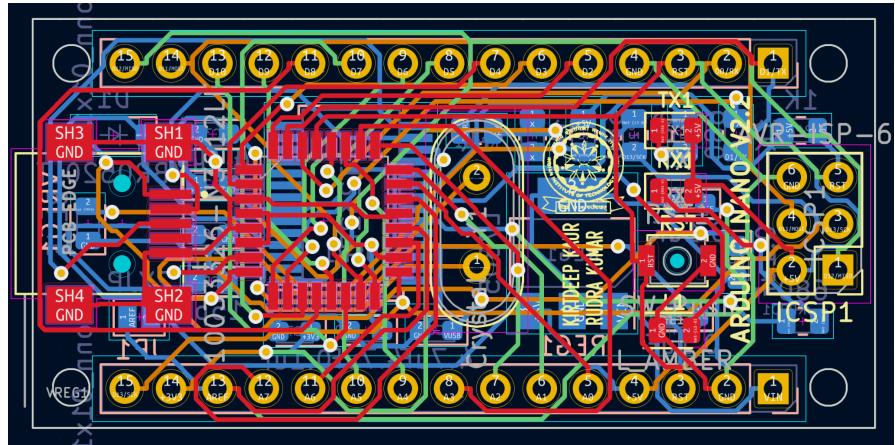
Back Layer



Inner Copper Layer

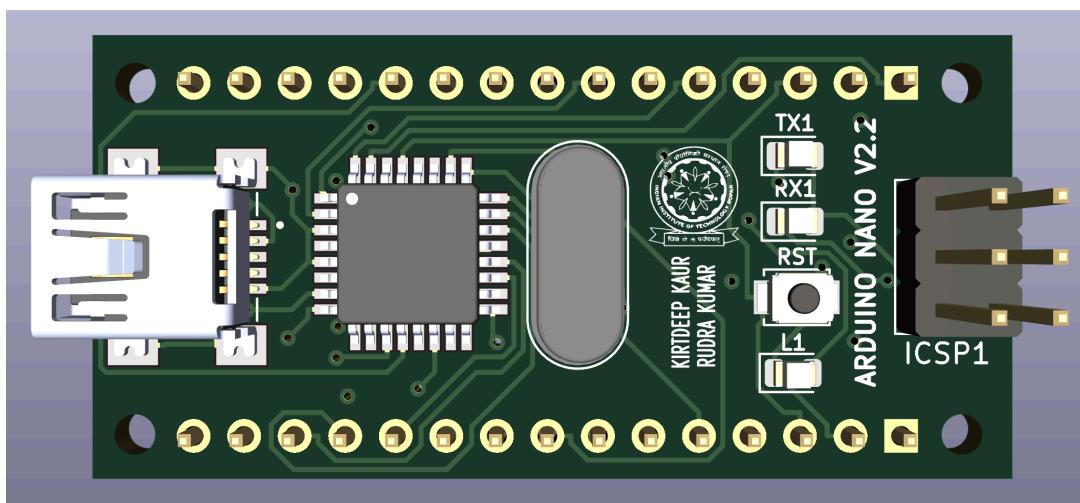


All Copper Layer

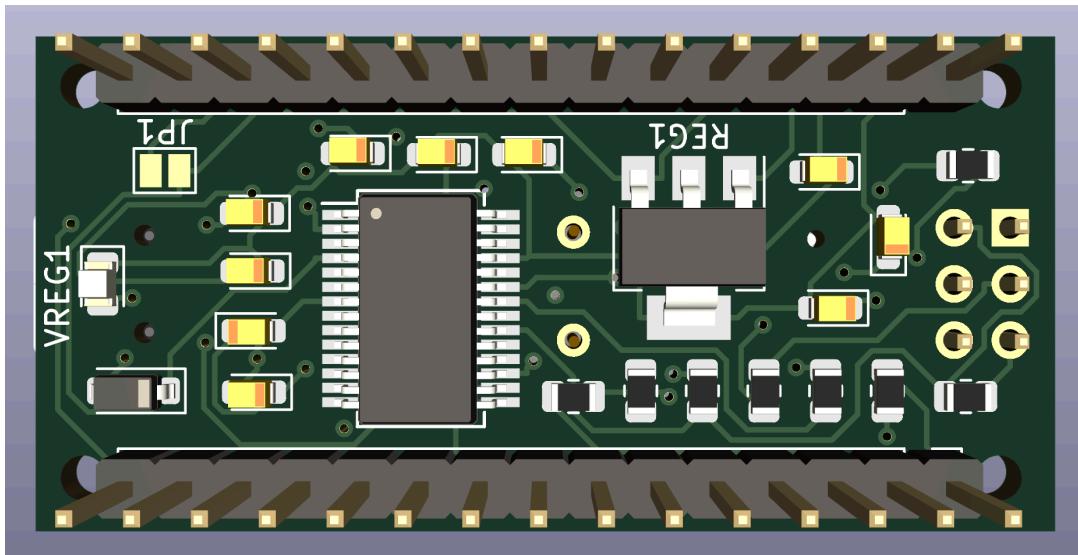


All Layers

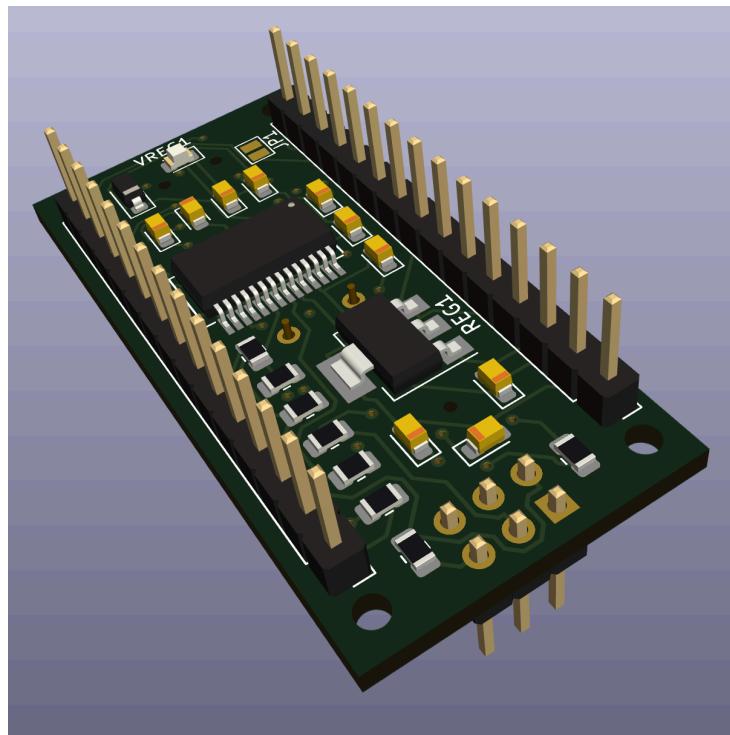
3D View:



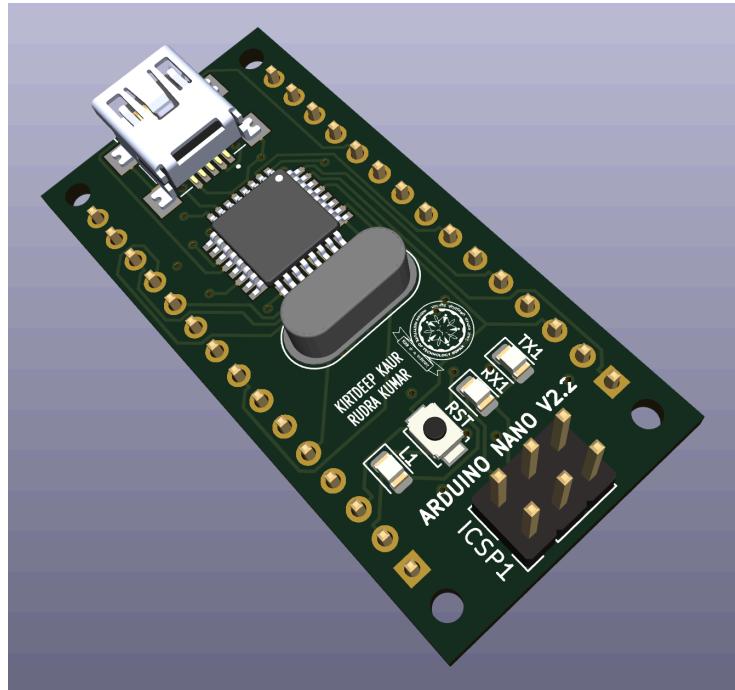
Top View



Bottom View



Slant Bottom View



Slant Top View