EE308 Power Electronic

Experiment 3: Arduino Nano PCB using KiCAD7

Group Number: 4

Student Name: Krishna Kumar and Parau Majhi

Entry Number: 2021EEB1183 and 2021EEB1191

Date: 21-02-2024

AIM:

- Basic of PCB design using KiCAD7
- PCB design of Arduino Nano in KiCAD7

Introduction:

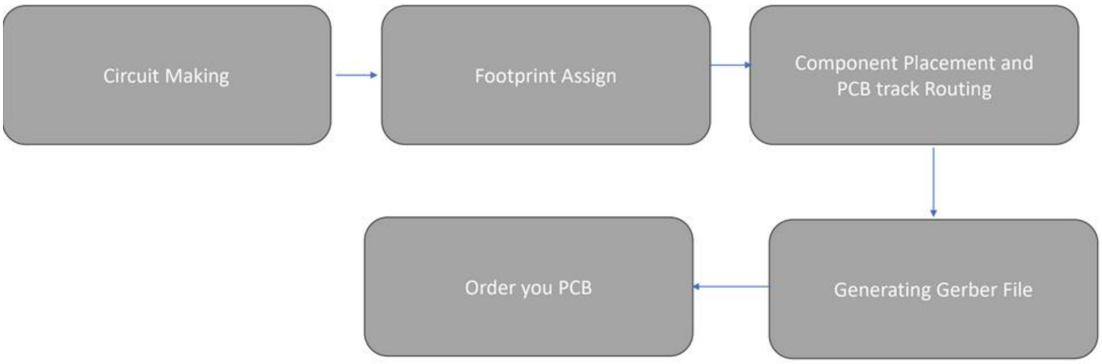
KiCad is an open-source software suite for creating electronic circuit schematics, printed circuit boards (PCBs), and associated part descriptions. KiCad supports an integrated design workflow in which a schematic and corresponding PCB are designed together, as well as standalone workflows for special uses. KiCad also includes several utilities to help with circuit and PCB design, including a PCB calculator for determining electrical properties of circuit structures, a Gerber viewer for inspecting manufacturing files, a 3D viewer for visualizing the finished PCB, and an integrated SPICE simulator for inspecting circuit behavior.

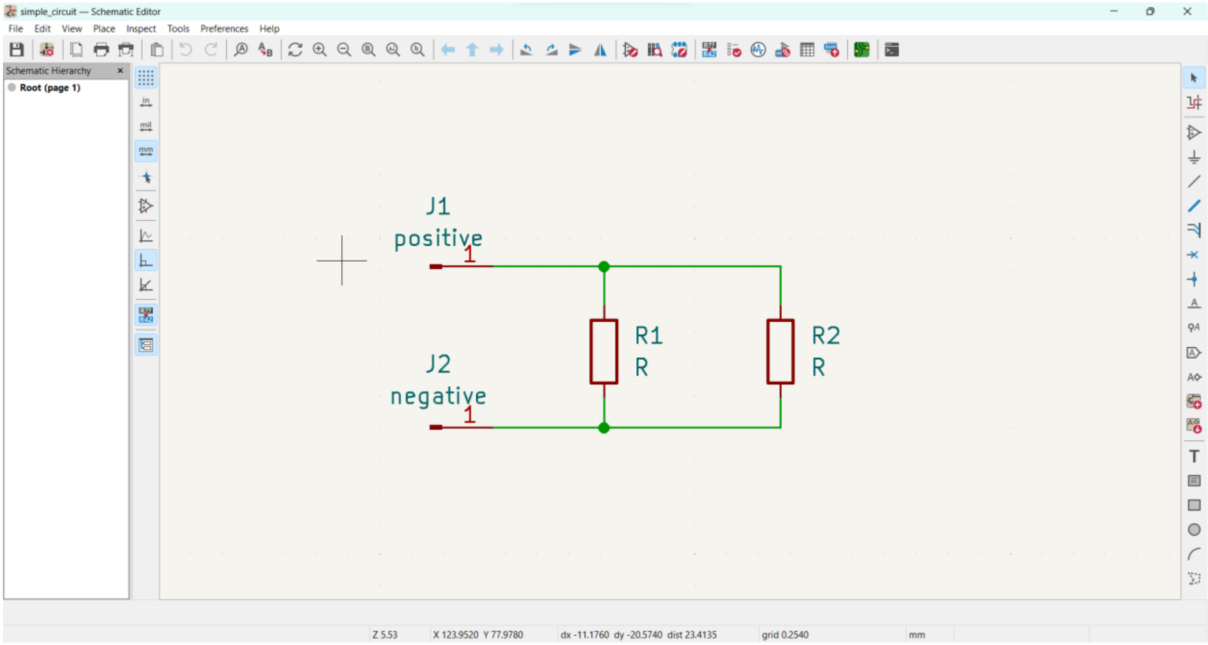
KiCad runs on all major operating systems and a wide range of computer hardware. It supports PCBs with up to 32 copper layers and is suitable for creating designs of all complexities. KiCad is developed by a volunteer team of software and electrical engineers around the world with a mission of creating free and open-source electronics design software suitable for professional designers.

Basic Concepts and Workflow:

The typical workflow in KiCad consists of two main tasks: drawing a schematic and laying out a circuit board.

The schematic is a symbolic representation of the circuit: which components are used and what connections are made between them. Schematic symbols are pictorial representations of electronic components in a schematic, such as a zigzag or rectangle for a resistor or a triangle for an opamp. The schematic contains symbols for every component in the design, with wires connecting pins in the symbols. The schematic is typically drawn first, before laying out the circuit board.

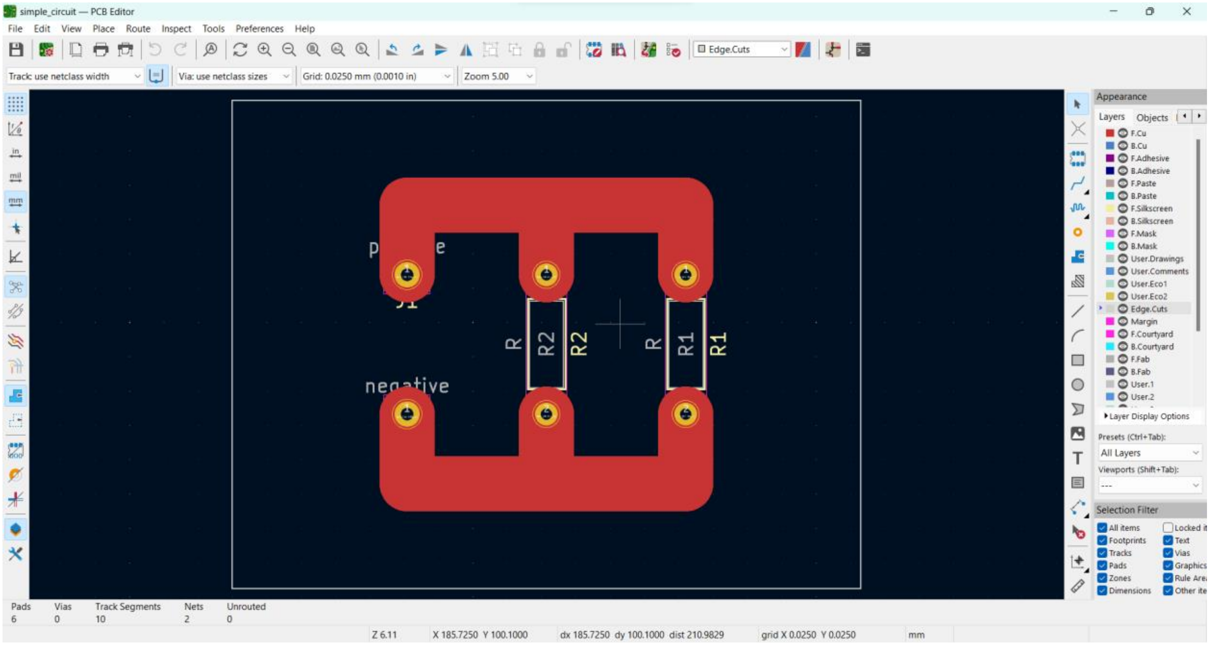



simple_circuit — Schematic Editor
File Edit View Place Inspect Tools Preferences Help
Schematic Hierarchy
Root (page 1)
J1
positive
1
J2
negative
1
R1
R
R2
R

Z 5.53 X 123.9520 Y 77.9780 dx -11.1760 dy -20.5740 dist 23.4135 grid 0.2540 mm

Figure 1: Schematic

The board is the physical realization of the schematic, with component footprints positioned on the board and copper tracks making the connections described in the schematic. Footprints are a set of copper pads that match the pins on a physical component. When the board is manufactured and assembled, the component will be soldered onto its corresponding footprint on the circuit board.



Track: use netclass width | Via: use netclass sizes | Grid: 0.0250 mm (0.0010 in) | Zoom 5.00

Pads	Vias	Track Segments	Nets	Unrouted
6	0	10	2	0

Z 6.11 X 185.7250 Y 100.1000 dx 185.7250 dy 100.1000 dist 210.9829 grid X 0.0250 Y 0.0250 mm

Figure 2: PCB layout and Routing

The board is the physical realization of the schematic, with component footprints positioned on the board and copper tracks making the connections described in the schematic. Footprints are a set of copper pads that match the pins on a physical component. When the board is manufactured and assembled, the component will be soldered onto its

corresponding footprint on the circuit board. KiCad has separate windows for drawing the schematic ("Schematic Editor"), laying out the board ("PCB Editor"), and editing symbols and footprints ("Symbol Editor" and "Footprint Editor"). KiCad comes with a large library of high quality, user contributed symbols and footprints, but it is also simple to create new symbols and footprints or modify existing symbols and footprints.

The board is the physical realization of the schematic, with component footprints positioned on the board and copper tracks making the connections described in the schematic. Footprints are a set of copper pads that match the pins on a physical component. When the board is manufactured and assembled, the component will be soldered onto its corresponding footprint on the circuit board. KiCad has separate windows for drawing the schematic ("Schematic Editor"), laying out the board ("PCB Editor"), and editing symbols and footprints ("Symbol Editor" and "Footprint Editor"). KiCad comes with a large library of high quality, user contributed symbols and footprints, but it is also simple to create new symbols and footprints or modify existing symbols and footprints.

PCB Design of Arduino Nano in KiCAD7:

The Arduino Nano is a small, complete, and breadboard-friendly board based on the ATmega328 (Arduino Nano 3.x). It has more or less the same functionality of the Arduino Duemilanove, but in a different package. It lacks only a DC power jack, and works with a Mini-B USB cable instead of a standard one.

The design flow, we had followed are as following:

1. Open KiCAD and open a new project from File
2. Making Circuit Schematic
3. Annotation and Footprint Assign
4. Component Placement and
5. PCB track Routing
6. Generating Gerber File
7. Saved it and Order it

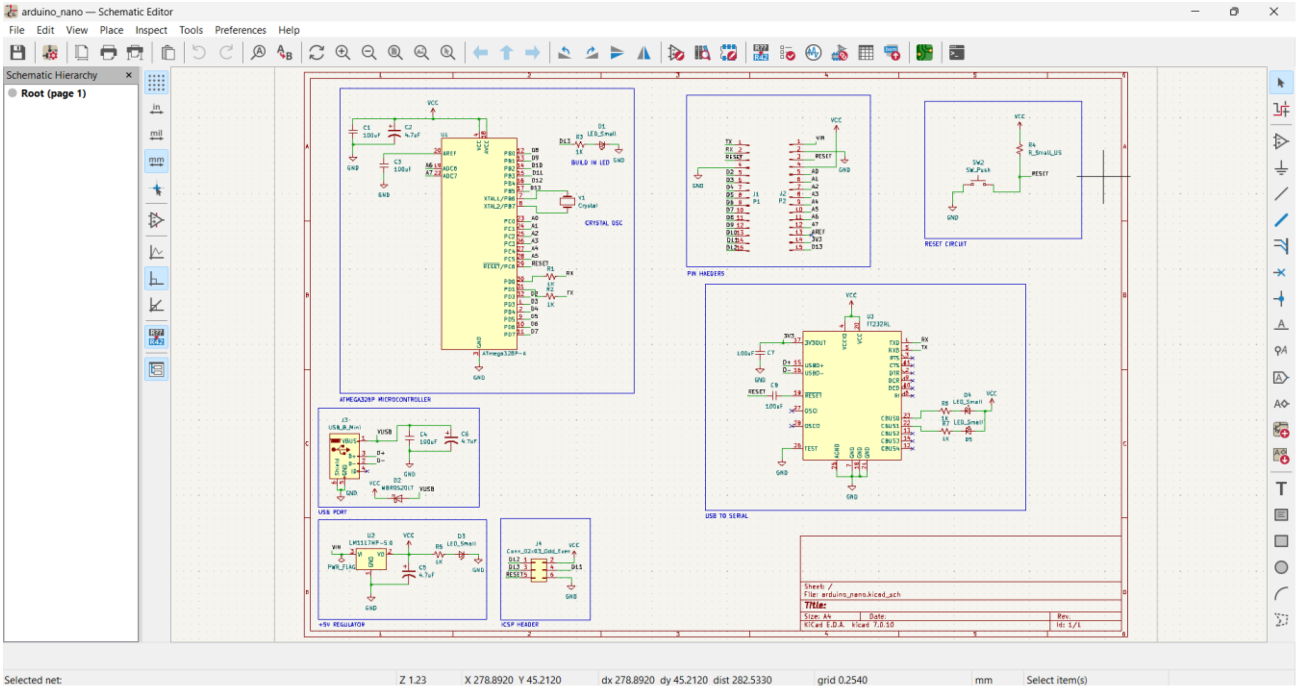
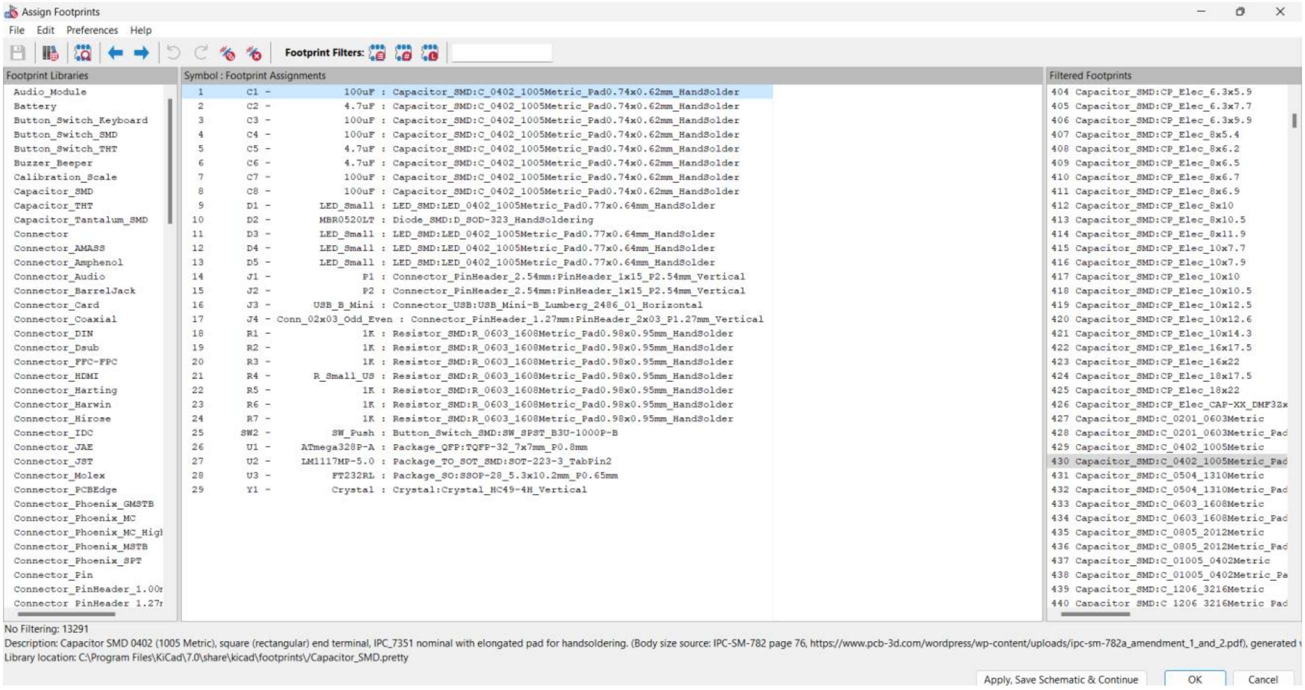

arduino_nano — Schematic Editor
File Edit View Place Inspect Tools Preferences Help
Schematic Hierarchy
Root (page 1)
ATMEGA328P MICROCONTROLLER
CRYSTAL OSC
BUILD IN LED
PIN HEADERS
RESET CIRCUIT
USB PORT
USB TO SERIAL
+5V REGULATOR
ICSP HEADER
Sheet: /
File: arduino_nano.kicad_sch
Title:
Size: A4 Date: Rev:
KiCad E.D.A. kicad 7.0.10 Id: 1/1
Selected net: Z 1.23 X 278.8920 Y 45.2120 dx 278.8920 dy 45.2120 dist 282.5330 grid 0.2540 mm Select item(s)

Figure 3: Schematic of Arduino Nano



Footprint Libraries

```
Audio_Module
Battery
Button_Switch_Keyboard
Button_Switch_SMD
Button_Switch_THT
Buzzer_Beeper
Calibration_Scale
Capacitor_SMD
Capacitor_THT
Capacitor_Tantalum_SMD
Connector
Connector_AMASS
Connector_Amphenol
Connector_Audio
Connector_BarrelJack
Connector_Card
Connector_Coaxial
Connector_DIN
Connector_Dsub
Connector_FFC-FPC
Connector_HDMI
Connector_Harting
Connector_Harwin
Connector_Hirose
Connector_IDC
Connector_JAE
Connector_JST
Connector_Molex
Connector_PCBEdge
Connector_Phoenix_GMSTB
Connector_Phoenix_MC
Connector_Phoenix_MC_Higl
Connector_Phoenix_MSTB
Connector_Phoenix_SPT
Connector_Pin
Connector_PinHeader_1.00r
Connector_PinHeader_1.27r
```

Symbol : Footprint Assignments

```
 1    C1  -             100uF : Capacitor_SMD:C_0402_1005Metric_Pad0.74x0.62mm_HandSolder
 2    C2  -             4.7uF : Capacitor_SMD:C_0402_1005Metric_Pad0.74x0.62mm_HandSolder
 3    C3  -             100uF : Capacitor_SMD:C_0402_1005Metric_Pad0.74x0.62mm_HandSolder
 4    C4  -             100uF : Capacitor_SMD:C_0402_1005Metric_Pad0.74x0.62mm_HandSolder
 5    C5  -             4.7uF : Capacitor_SMD:C_0402_1005Metric_Pad0.74x0.62mm_HandSolder
 6    C6  -             4.7uF : Capacitor_SMD:C_0402_1005Metric_Pad0.74x0.62mm_HandSolder
 7    C7  -             100uF : Capacitor_SMD:C_0402_1005Metric_Pad0.74x0.62mm_HandSolder
 8    C8  -             100uF : Capacitor_SMD:C_0402_1005Metric_Pad0.74x0.62mm_HandSolder
 9    D1  -         LED_Small : LED_SMD:LED_0402_1005Metric_Pad0.77x0.64mm_HandSolder
10    D2  -         MBR0520LT : Diode_SMD:D_SOD-323_HandSoldering
11    D3  -         LED_Small : LED_SMD:LED_0402_1005Metric_Pad0.77x0.64mm_HandSolder
12    D4  -         LED_Small : LED_SMD:LED_0402_1005Metric_Pad0.77x0.64mm_HandSolder
13    D5  -         LED_Small : LED_SMD:LED_0402_1005Metric_Pad0.77x0.64mm_HandSolder
14    J1  -                P1 : Connector_PinHeader_2.54mm:PinHeader_1x15_P2.54mm_Vertical
15    J2  -                P2 : Connector_PinHeader_2.54mm:PinHeader_1x15_P2.54mm_Vertical
16    J3  -        USB_B_Mini : Connector_USB:USB_Mini-B_Lumberg_2486_01_Horizontal
17    J4  - Conn_02x03_Odd_Even : Connector_PinHeader_1.27mm:PinHeader_2x03_P1.27mm_Vertical
18    R1  -                1K : Resistor_SMD:R_0603_1608Metric_Pad0.98x0.95mm_HandSolder
19    R2  -                1K : Resistor_SMD:R_0603_1608Metric_Pad0.98x0.95mm_HandSolder
20    R3  -                1K : Resistor_SMD:R_0603_1608Metric_Pad0.98x0.95mm_HandSolder
21    R4  -        R_Small_US : Resistor_SMD:R_0603_1608Metric_Pad0.98x0.95mm_HandSolder
22    R5  -                1K : Resistor_SMD:R_0603_1608Metric_Pad0.98x0.95mm_HandSolder
23    R6  -                1K : Resistor_SMD:R_0603_1608Metric_Pad0.98x0.95mm_HandSolder
24    R7  -                1K : Resistor_SMD:R_0603_1608Metric_Pad0.98x0.95mm_HandSolder
25   SW2  -           SW_Push : Button_Switch_SMD:SW_SPST_B3U-1000P-B
26    U1  -       ATmega328P-A : Package_QFP:TQFP-32_7x7mm_P0.8mm
27    U2  -       LM1117MP-5.0 : Package_TO_SOT_SMD:SOT-223-3_TabPin2
28    U3  -           FT232RL : Package_SO:SSOP-28_5.3x10.2mm_P0.65mm
29    Y1  -           Crystal : Crystal:Crystal_HC49-4H_Vertical
```

Filtered Footprints

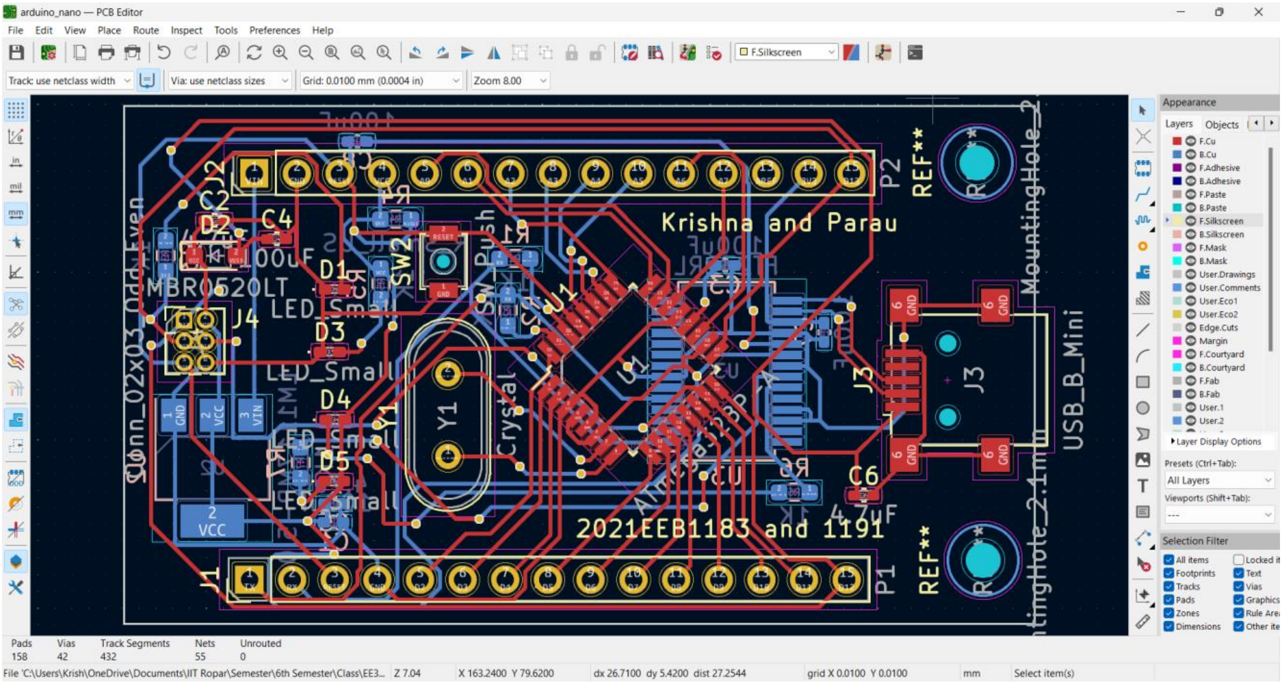
```
404 Capacitor_SMD:CP_Elec_6.3x5.9
405 Capacitor_SMD:CP_Elec_6.3x7.7
406 Capacitor_SMD:CP_Elec_6.3x9.9
407 Capacitor_SMD:CP_Elec_8x5.4
408 Capacitor_SMD:CP_Elec_8x6.2
409 Capacitor_SMD:CP_Elec_8x6.5
410 Capacitor_SMD:CP_Elec_8x6.7
411 Capacitor_SMD:CP_Elec_8x6.9
412 Capacitor_SMD:CP_Elec_8x10
413 Capacitor_SMD:CP_Elec_8x10.5
414 Capacitor_SMD:CP_Elec_8x11.9
415 Capacitor_SMD:CP_Elec_10x7.7
416 Capacitor_SMD:CP_Elec_10x7.9
417 Capacitor_SMD:CP_Elec_10x10
418 Capacitor_SMD:CP_Elec_10x10.5
419 Capacitor_SMD:CP_Elec_10x12.5
420 Capacitor_SMD:CP_Elec_10x12.6
421 Capacitor_SMD:CP_Elec_10x14.3
422 Capacitor_SMD:CP_Elec_16x17.5
423 Capacitor_SMD:CP_Elec_16x22
424 Capacitor_SMD:CP_Elec_18x17.5
425 Capacitor_SMD:CP_Elec_18x22
426 Capacitor_SMD:CP_Elec_CAP-XX_DMF3Zx
427 Capacitor_SMD:C_0201_0603Metric
428 Capacitor_SMD:C_0201_0603Metric_Pad
429 Capacitor_SMD:C_0402_1005Metric
430 Capacitor_SMD:C_0402_1005Metric_Pad
431 Capacitor_SMD:C_0504_1310Metric
432 Capacitor_SMD:C_0504_1310Metric_Pad
433 Capacitor_SMD:C_0603_1608Metric
434 Capacitor_SMD:C_0603_1608Metric_Pad
435 Capacitor_SMD:C_0805_2012Metric
436 Capacitor_SMD:C_0805_2012Metric_Pad
437 Capacitor_SMD:C_01005_0402Metric
438 Capacitor_SMD:C_01005_0402Metric_Pa
439 Capacitor_SMD:C_1206_3216Metric
440 Capacitor_SMD:C_1206_3216Metric_Pad
```

No Filtering: 13291

Description: Capacitor SMD 0402 (1005 Metric), square (rectangular) end terminal, IPC_7351 nominal with elongated pad for handsoldering. (Body size source: IPC-SM-782 page 76, https://www.pcb-3d.com/wordpress/wp-content/uploads/ipc-sm-782a_amendment_1_and_2.pdf), generated

Library location: C:\Program Files\KiCad\7.0\share\kicad\footprints\/Capacitor_SMD.pretty

Figure 4: Footprints od given schematic


arduino_nano — PCB Editor

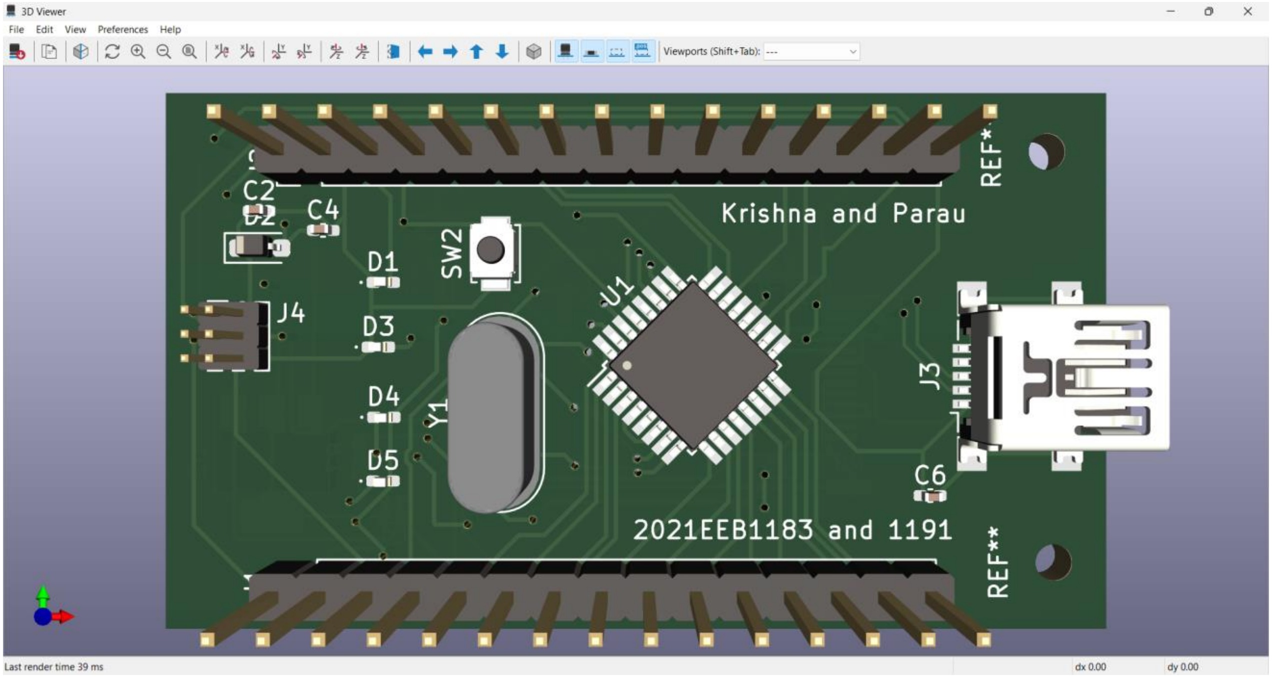
Track: use netclass width | Via: use netclass sizes | Grid: 0.0100 mm (0.0004 in) | Zoom 8.00

F.Silkscreen

Pads	Vias	Track Segments	Nets	Unrouted
158	42	432	55	0

File 'C:\Users\Krish\OneDrive\Documents\IIT Ropar\Semester\6th Semester\Class\EE3... Z 7.04 X 163.2400 Y 79.6200 dx 26.7100 dy 5.4200 dist 27.2544 grid X 0.0100 Y 0.0100 mm Select item(s)

Figure 5: Placement and Routing part


3D Viewer

Viewports (Shift+Tab): ---

Last render time 39 ms dx 0.00 dy 0.00

Figure 6: 3D view of Arduino Nano (front view)

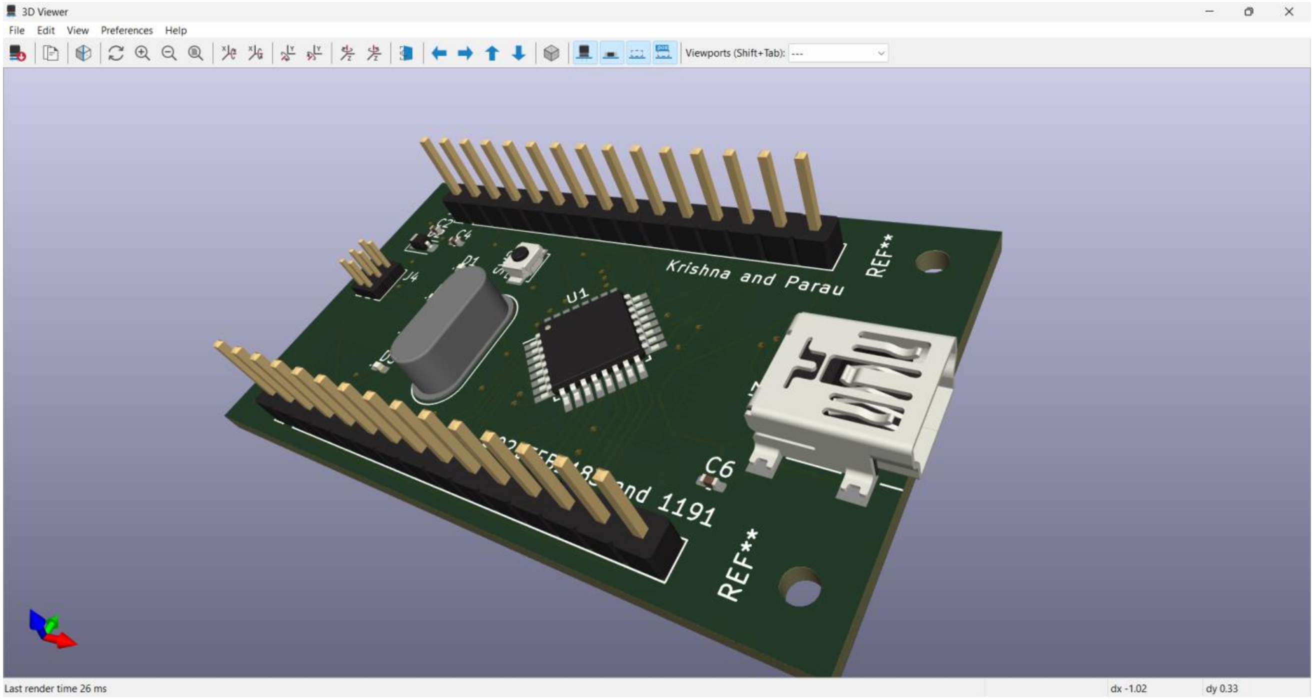


Figure 7: 3D view of Arduino Nano (side view)

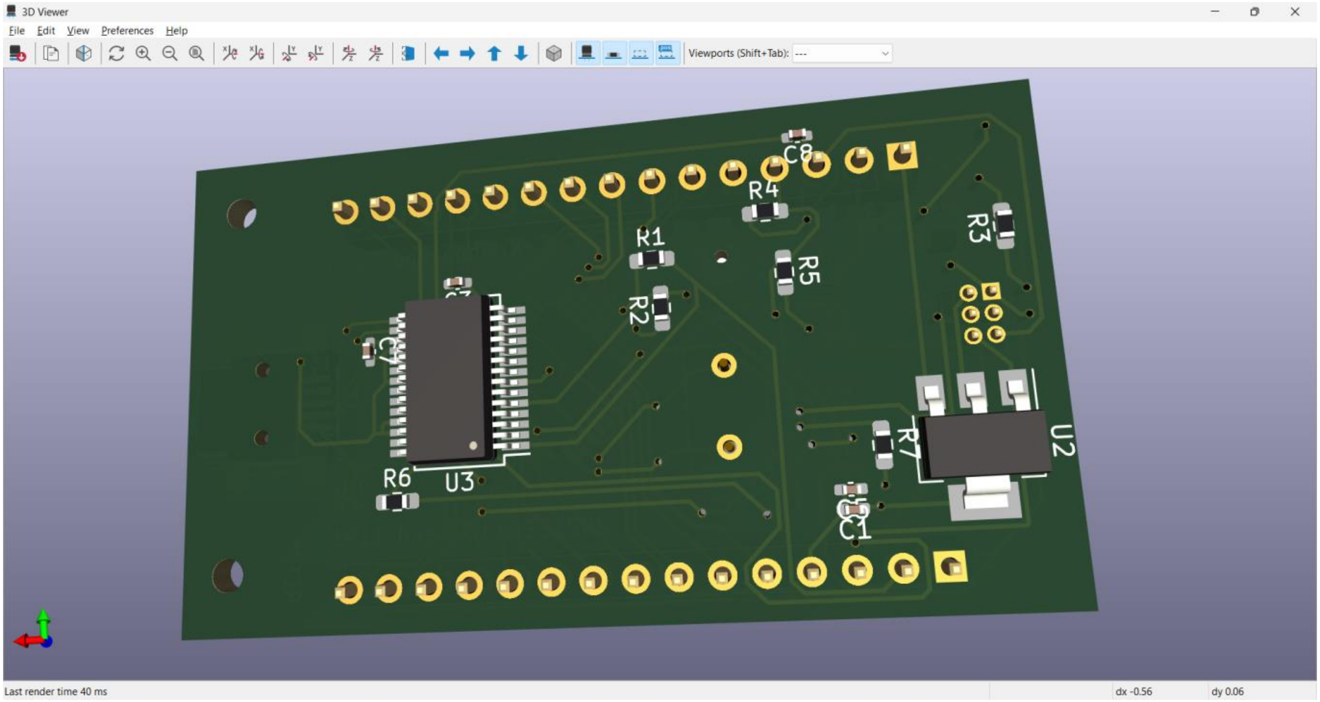


Figure 8: 3D view of Arduino Nano (back view)

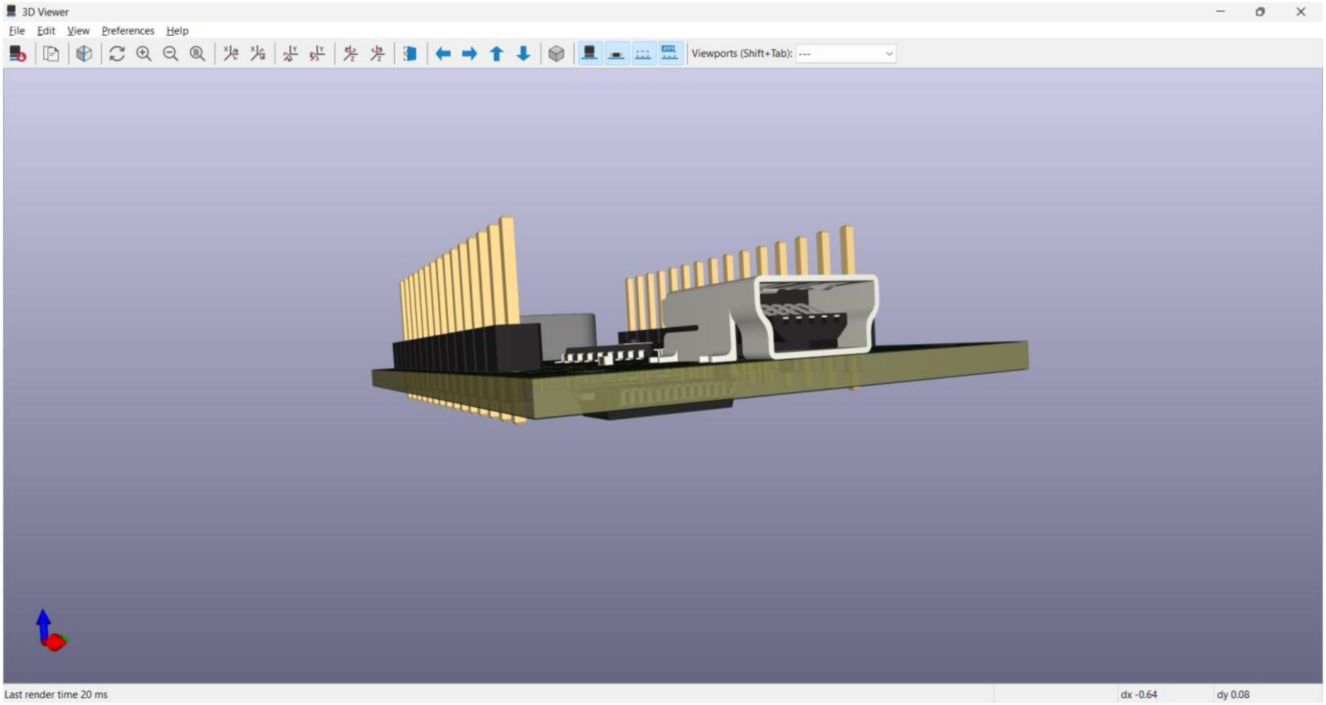


Figure 9: 3D view of Arduino Nano (side-front view)

Conclusion:

The PCB design of a Arduino Nano using KiCad offers valuable insights and practical skills that can be leveraged for various applications. Whether designing power supplies, motor control circuits, or other electronic systems, the knowledge gained from this experience can be applied to enhance convenience, efficiency, and innovation in future endeavours. Embracing the capabilities of KiCad empowers electronics enthusiasts and professionals alike to bring their ideas to life with confidence and precision.