PCB DESIGN WORKSHOP

By

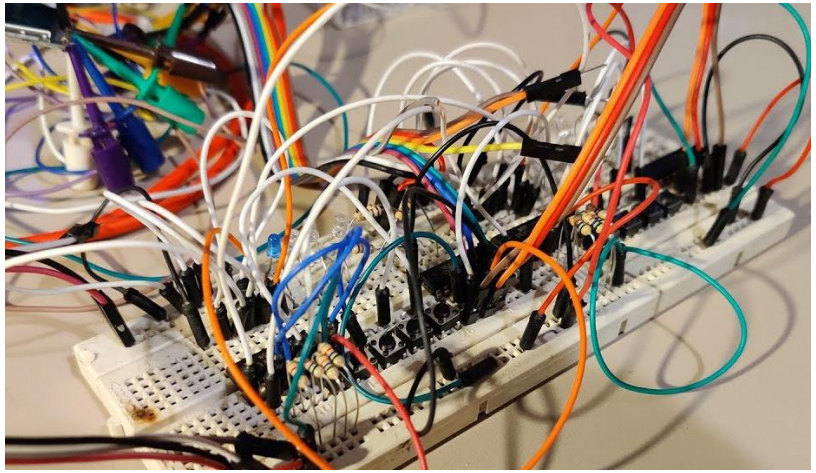
Y-Dyuthi

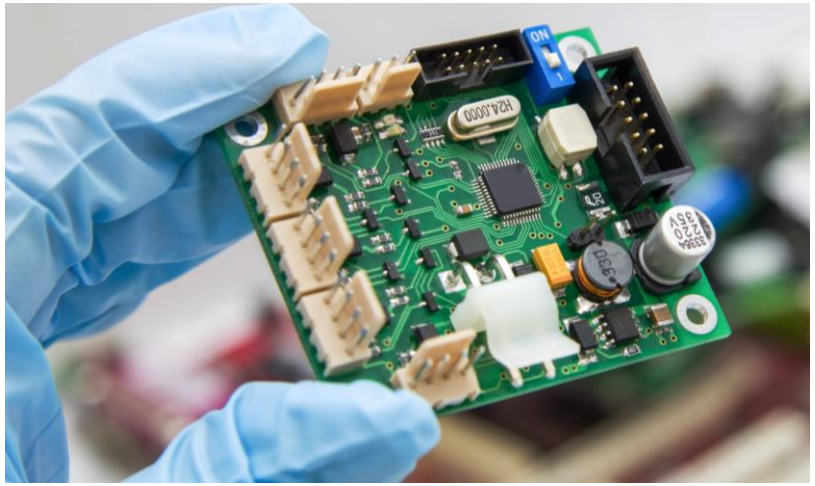
DAY – 1

Introduction to KiCad

What is a PCB?

A PCB (Printed Circuit Board) provides a more reliable and **permanent electrical connection** compared to the messy and unreliable connections of a breadboard.

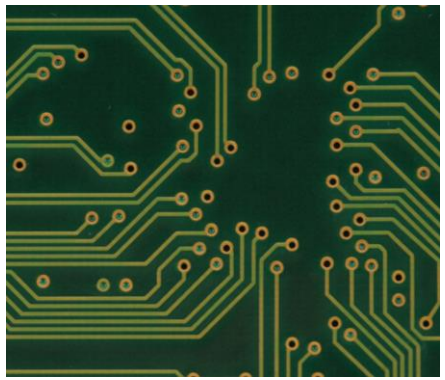




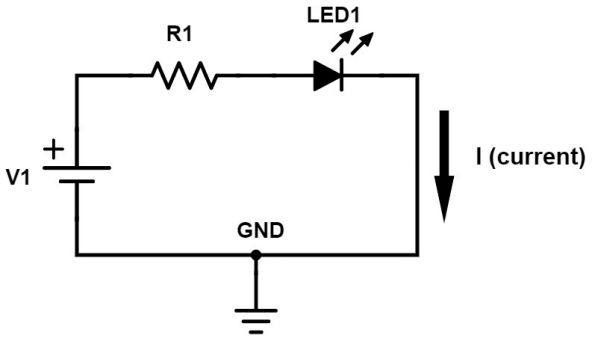
For textbook definitions https://en.wikipedia.org/wiki/Printed_circuit_board

What is a PCB ?

Printed



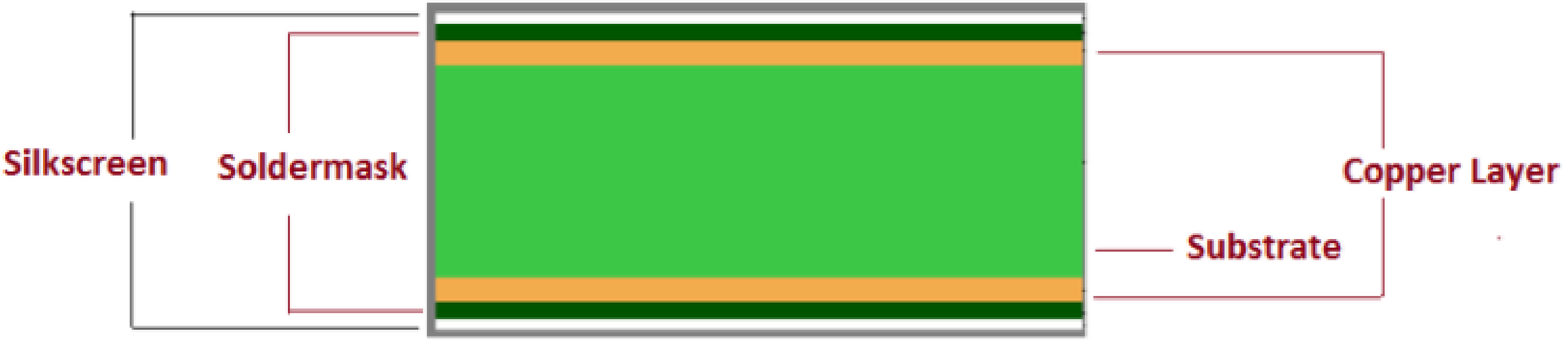
Circuit



Board

For textbook definitions https://en.wikipedia.org/wiki/Printed_circuit_board

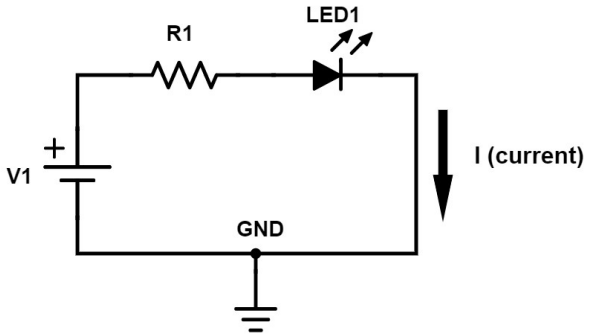
PCB Structure



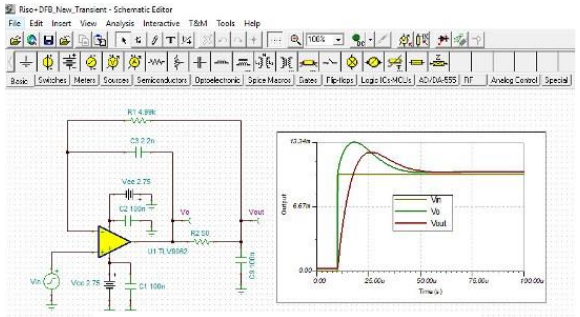
For textbook definitions https://en.wikipedia.org/wiki/Printed_circuit_board

PCB making: Process flow

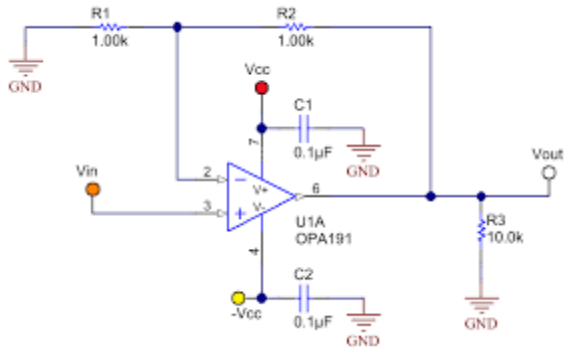
Circuit design



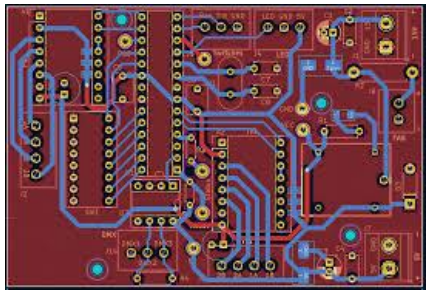
Verification (simulation, analysis)



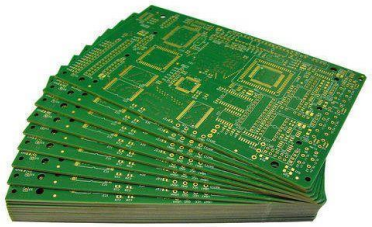
Schematic design (assign footprints)



Placement & Layout



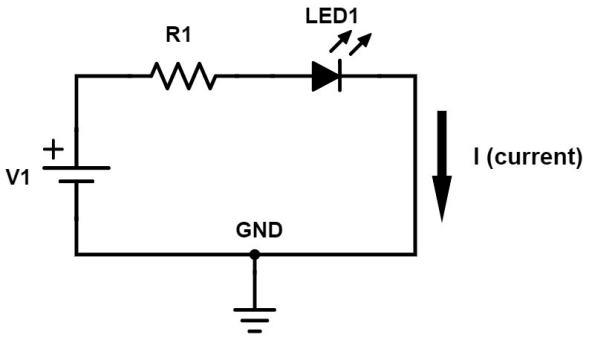
Fabrication

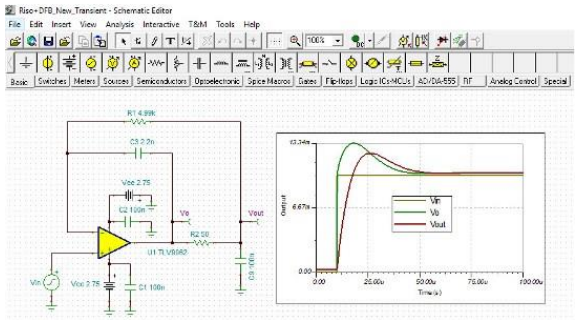


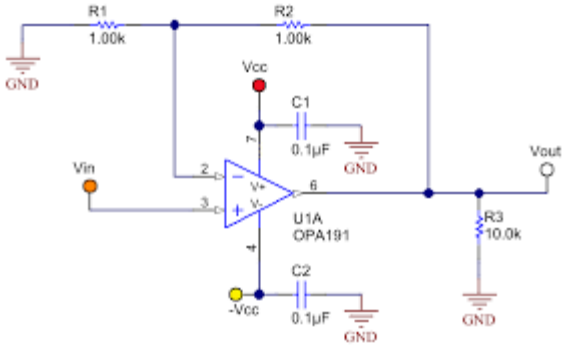
Assembly

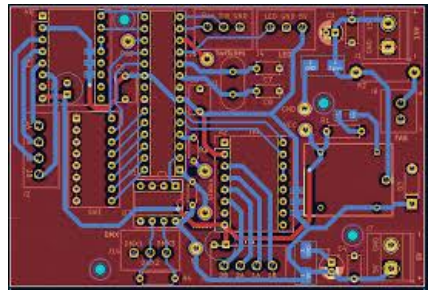
What will we focus on ?

Circuit design → **Verification (simulation, analysis)** → **Schematic design (assign footprints)** → **Placement & Layout**









We will use **LTspice**

We will use **KiCad**

PCB Design Tools

Numerous CAD tools are available for PCB designing, including both licensed and free open-source options.

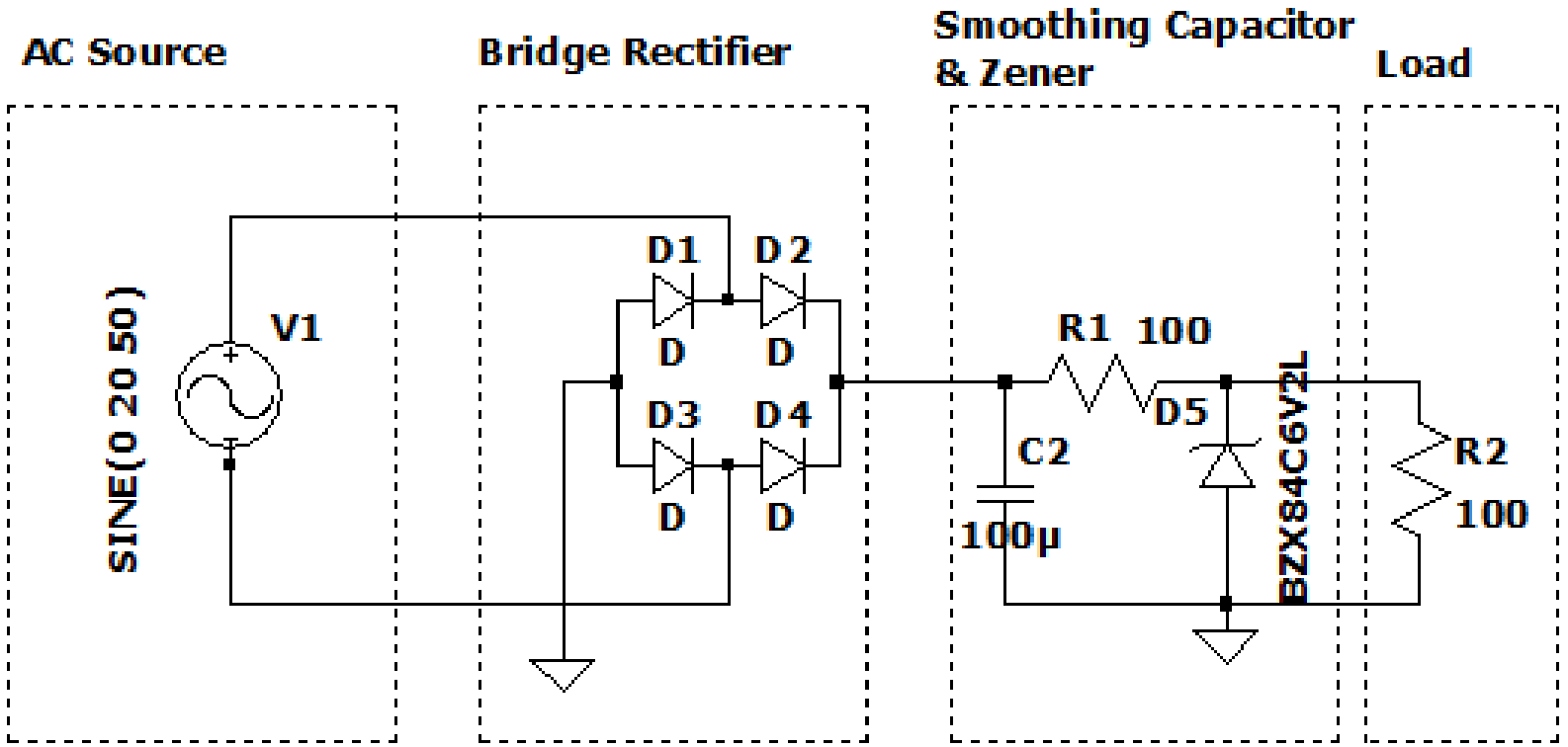
Why KiCad ?

Open-source, cross-platform, no license fees and a lot of community support.
Plus, who doesn't love free stuff?

Get Kicad: https://www.kicad.org/download/

Enough Theory.. Let’s get started

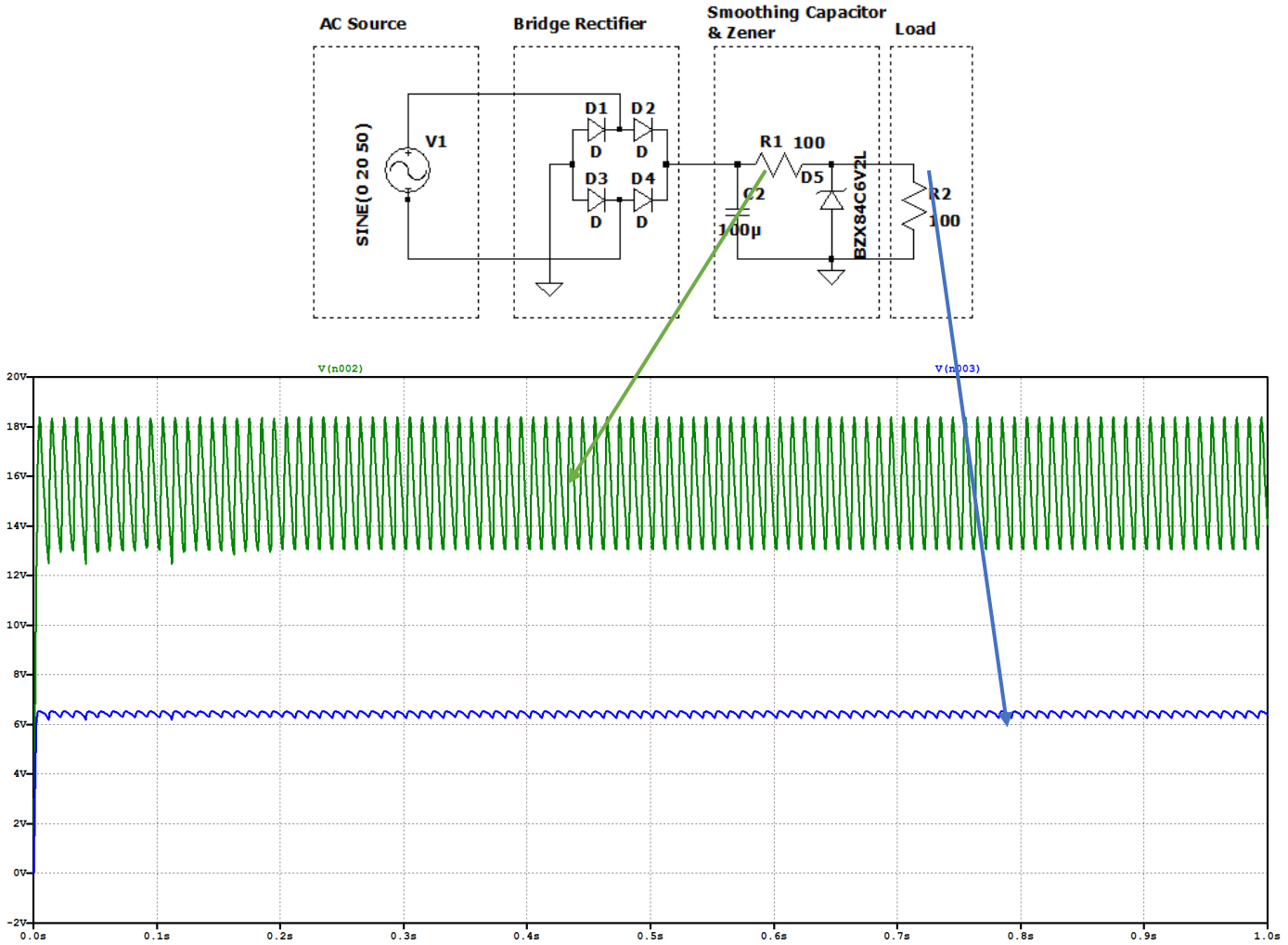
Making an AC to DC convertor



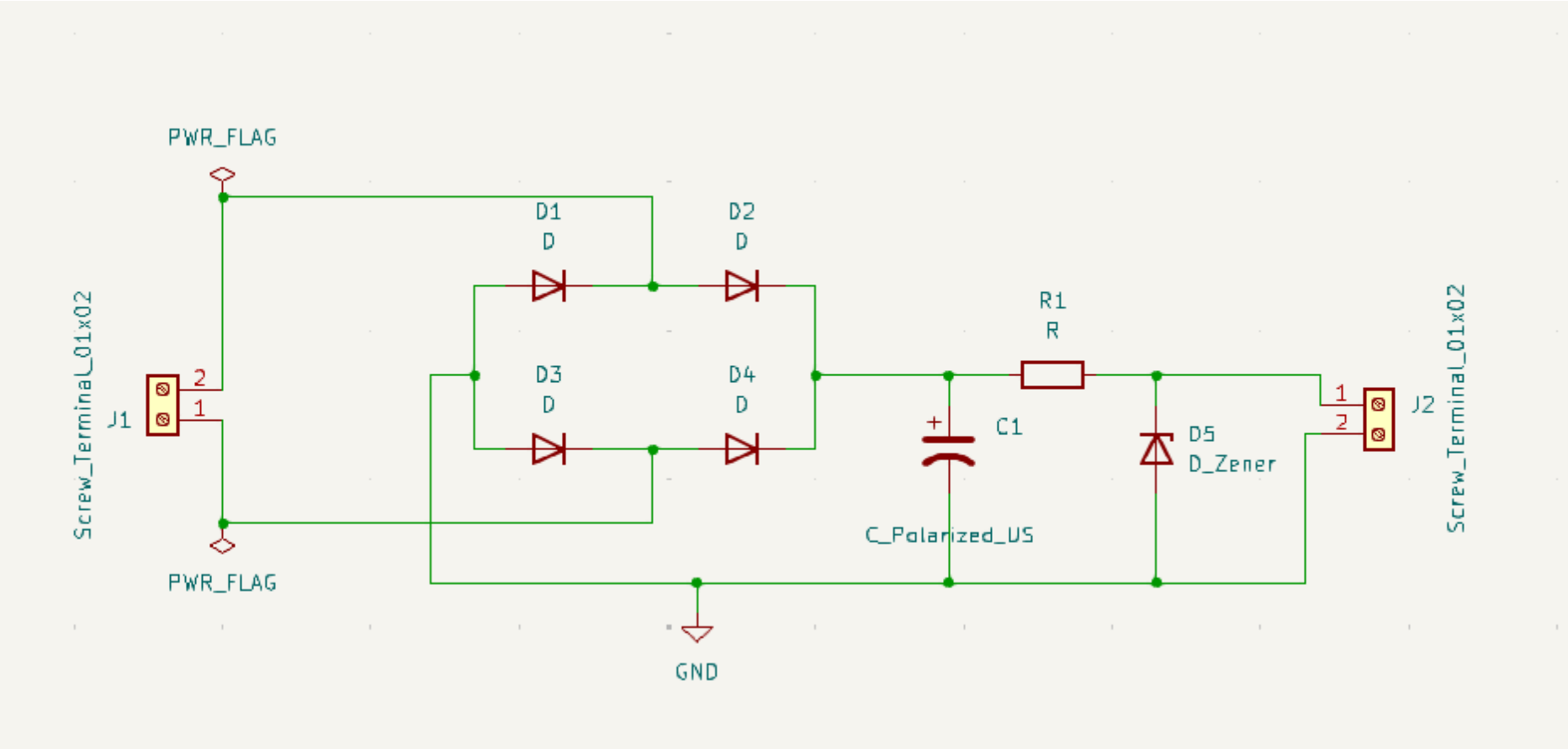
https://www.electronics-tutorials.ws/diode/diode_6.html

Simulation and Verification

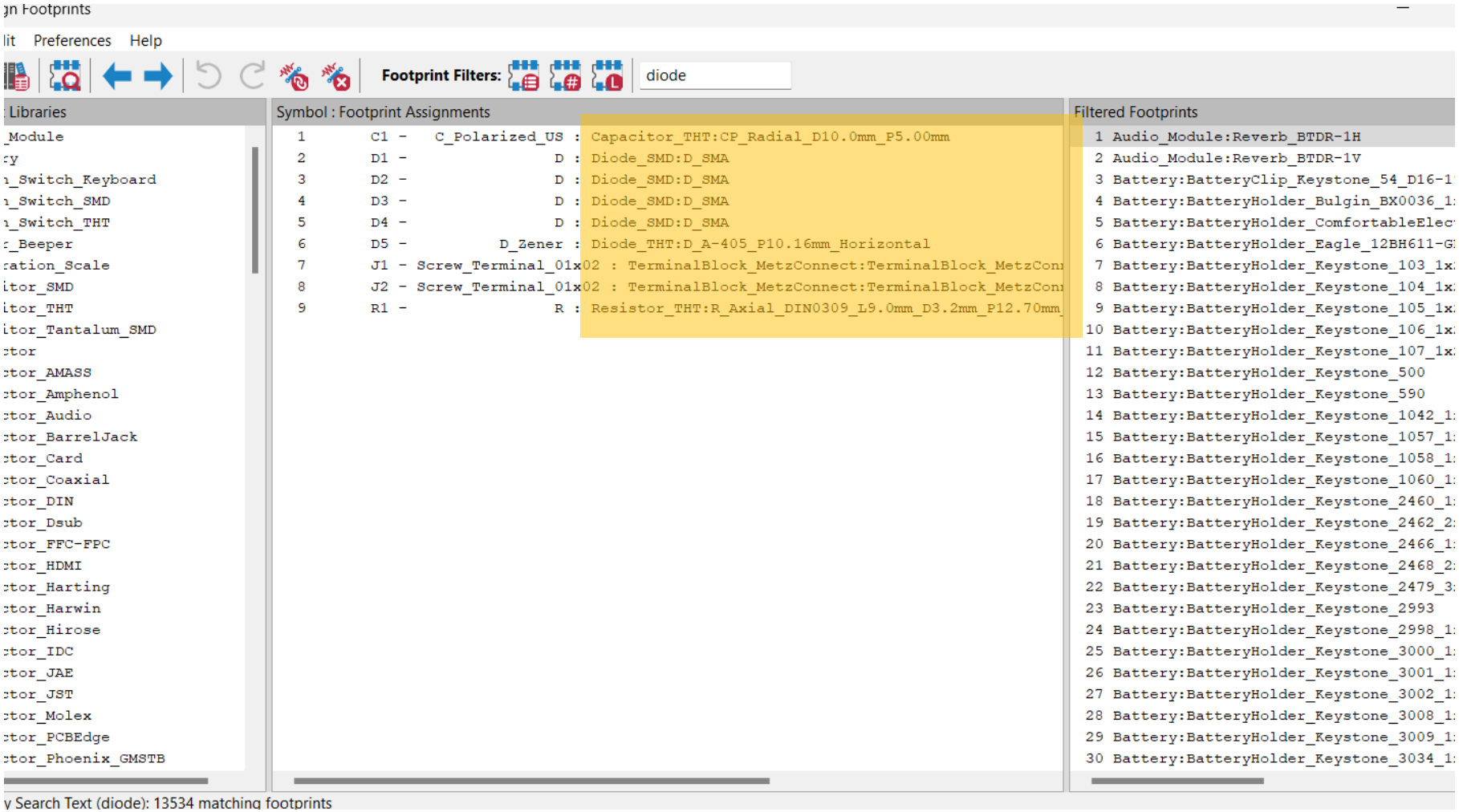
Verify the functionality using some simulator (Ltspice),
Note: a theoretical simulation is not possible for all designs some time you have to go with the hunch



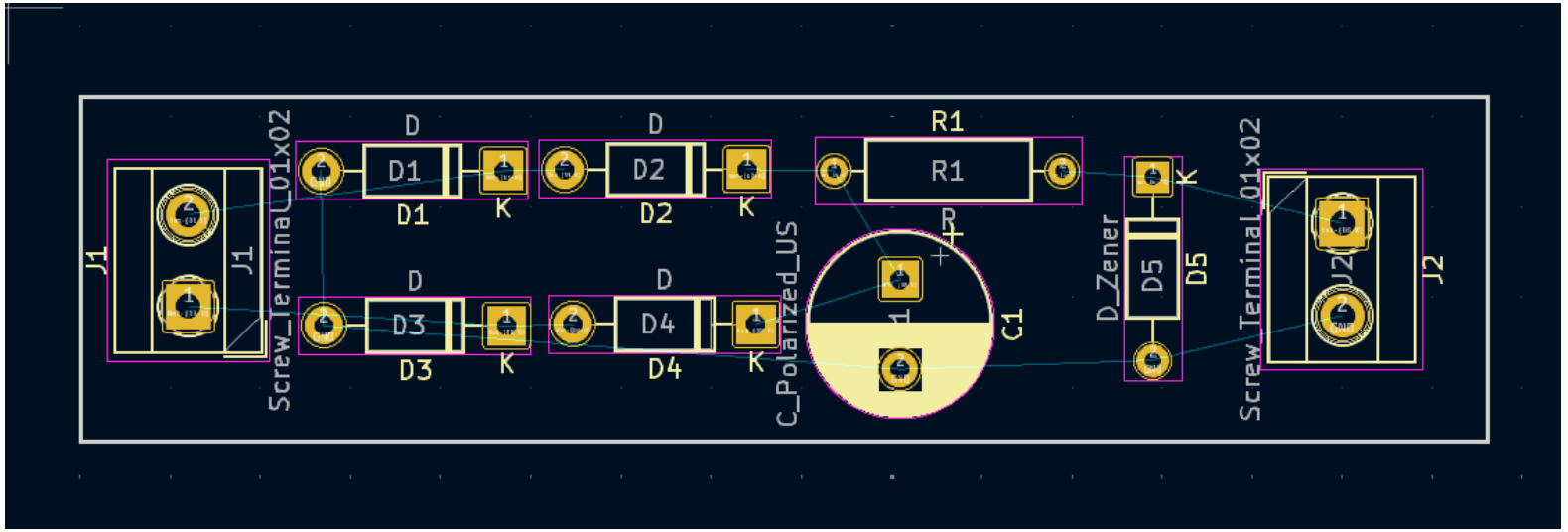
Develop the schematic

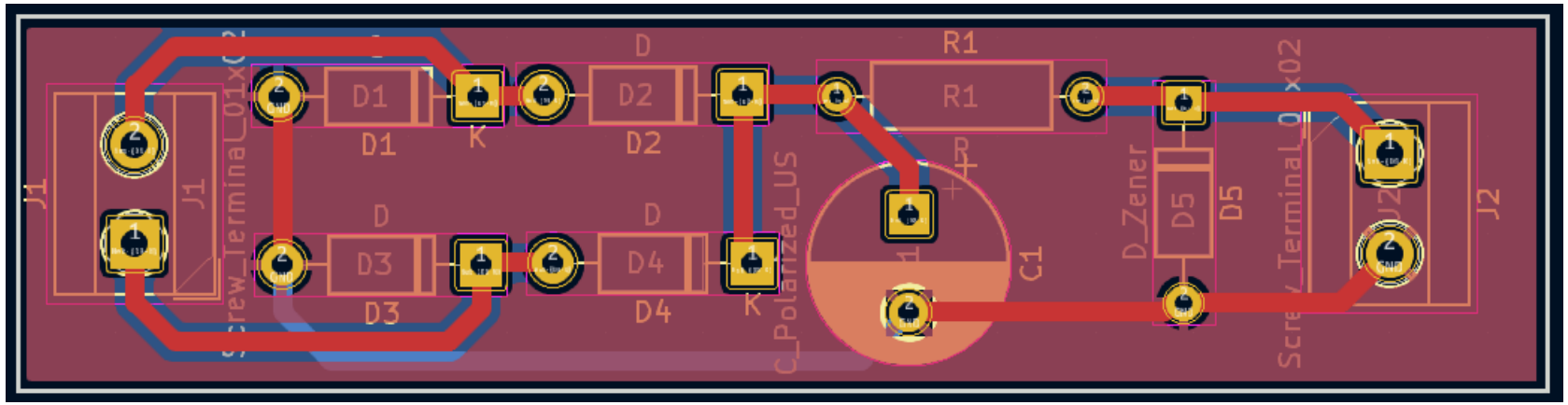


Footprint assignment

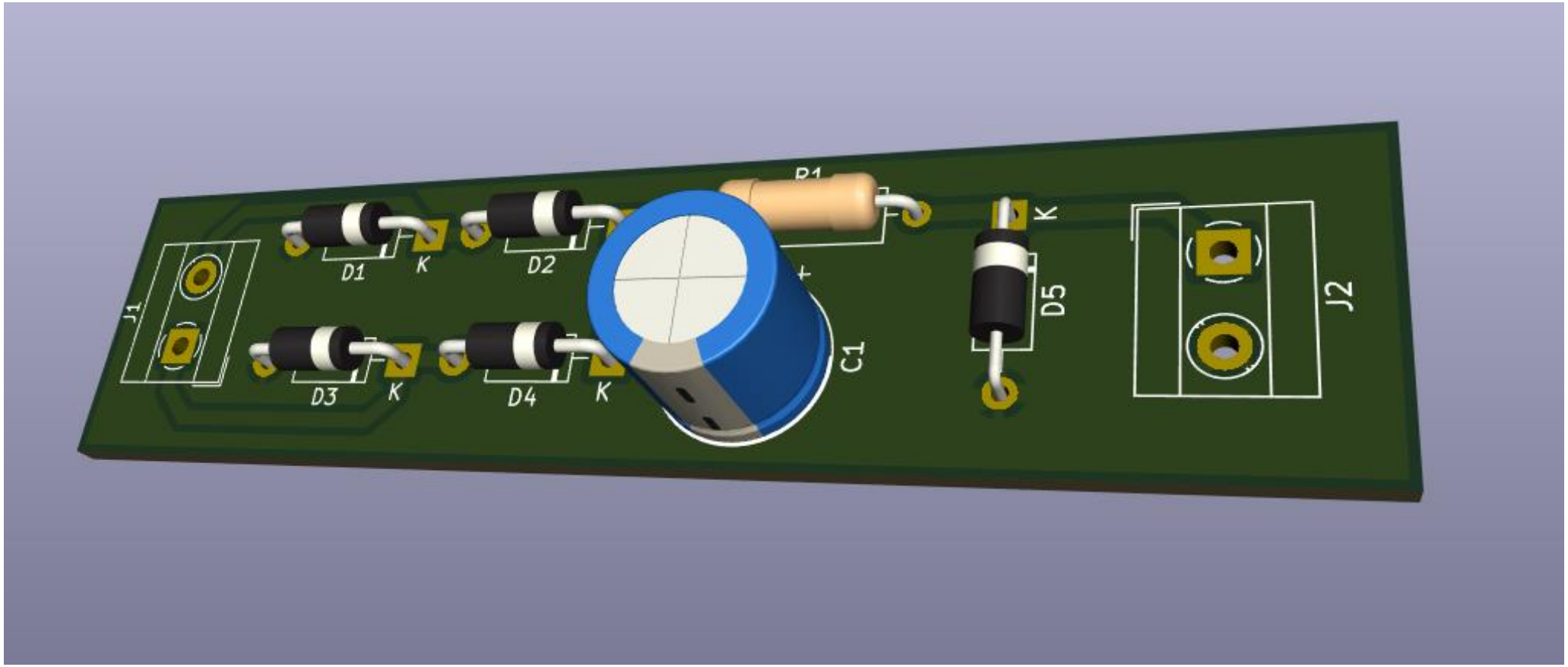


Placement & Layout





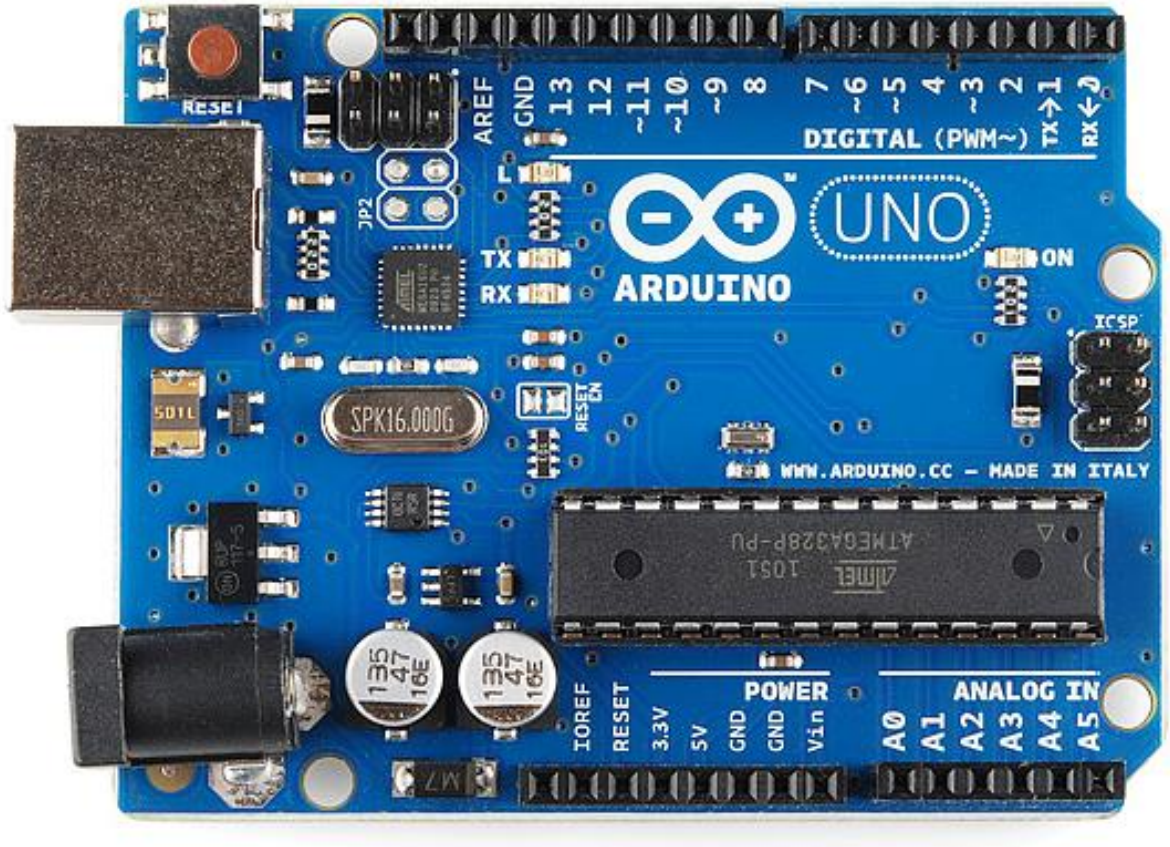
Here’s the result!



DAY – 2

Now to more complex things

Here’s what we’ll be designing



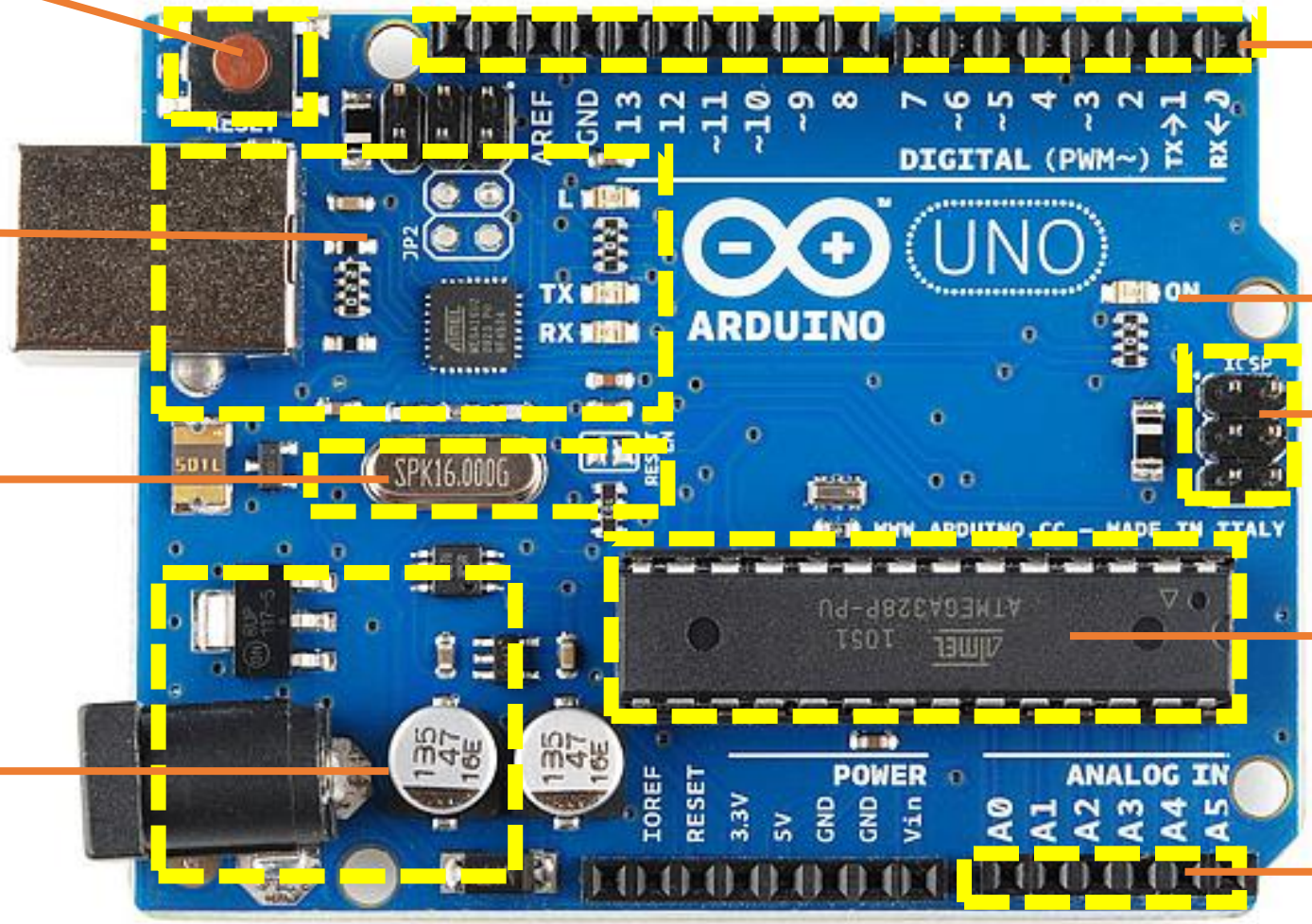
We will make a development board like Arduino (but a simpler version)

Reset Pushbutton
With debounce capacitors

USB
For – Powering the circuit, programming the Atmega chip, communication (Rx and Tx)

Clock circuit
A quartz crystal oscillator circuit of frequency 16MHz

Power supply
An external power supply, with voltage regulators and decoupling capacitors



I/O pins Digital

Indicator LED's
for power, Tx, Rx

AVR - ISP
Alternative program connector option

Atmega (microcontroller IC)
The brain of this board

I/O pins Analog

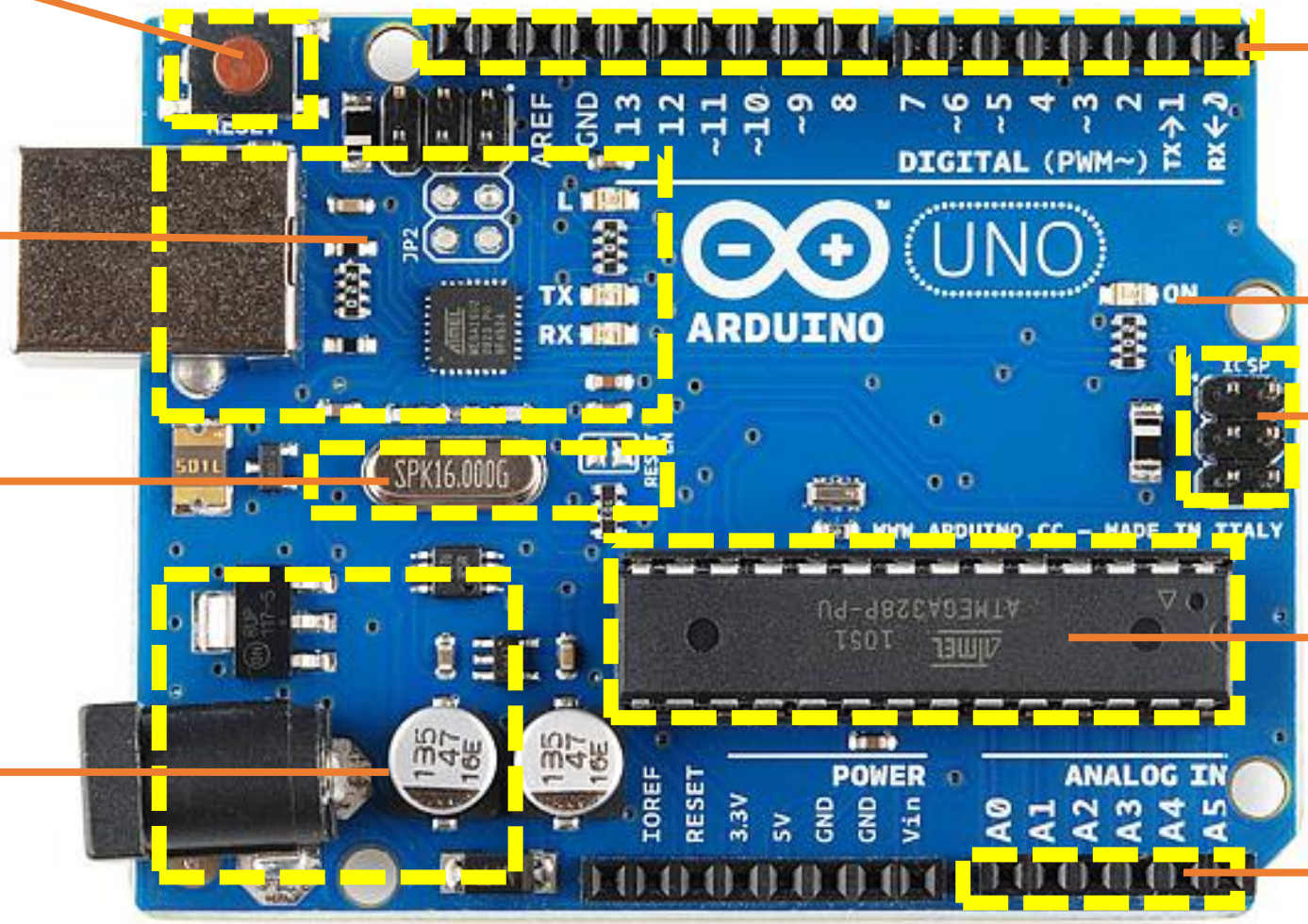
We will make a simple version

✓ **Reset Pushbutton**
With debounce capacitors

USB ✗
For – Powering the circuit, programming the Atmega chip, communication (Rx and Tx)

✓ **Clock circuit**
A quartz crystal oscillator circuit of frequency 16MHz

✓ **Power supply**
An external power supply, with voltage regulators and decoupling capacitors



✓ **I/O pins Digital**

✓ **Indicator LED's**
for power, Tx, Rx

✓ **AVR - ISP**
Alternative program connector option

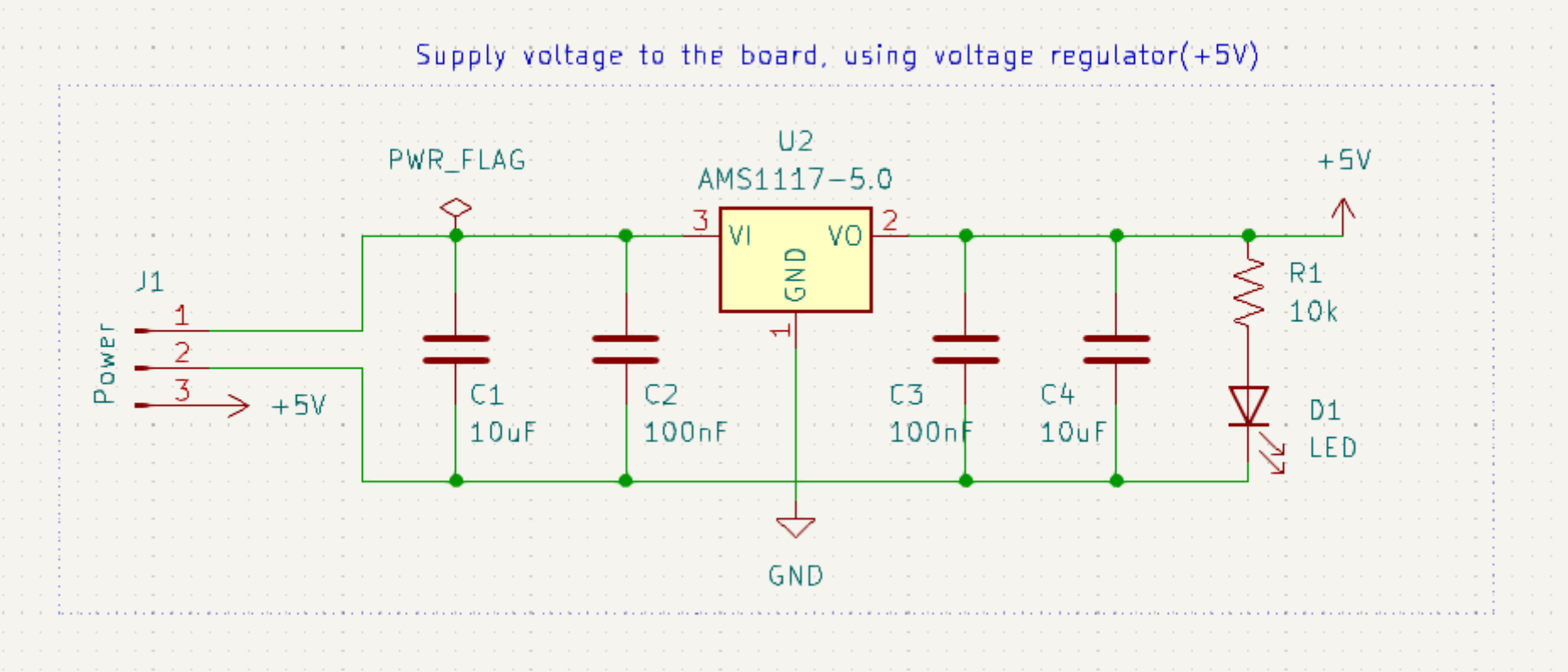
✓ **Atmega (microcontroller IC)**
The brain of this board

✓ **I/O pins Analog**

Let’s make the schematic!

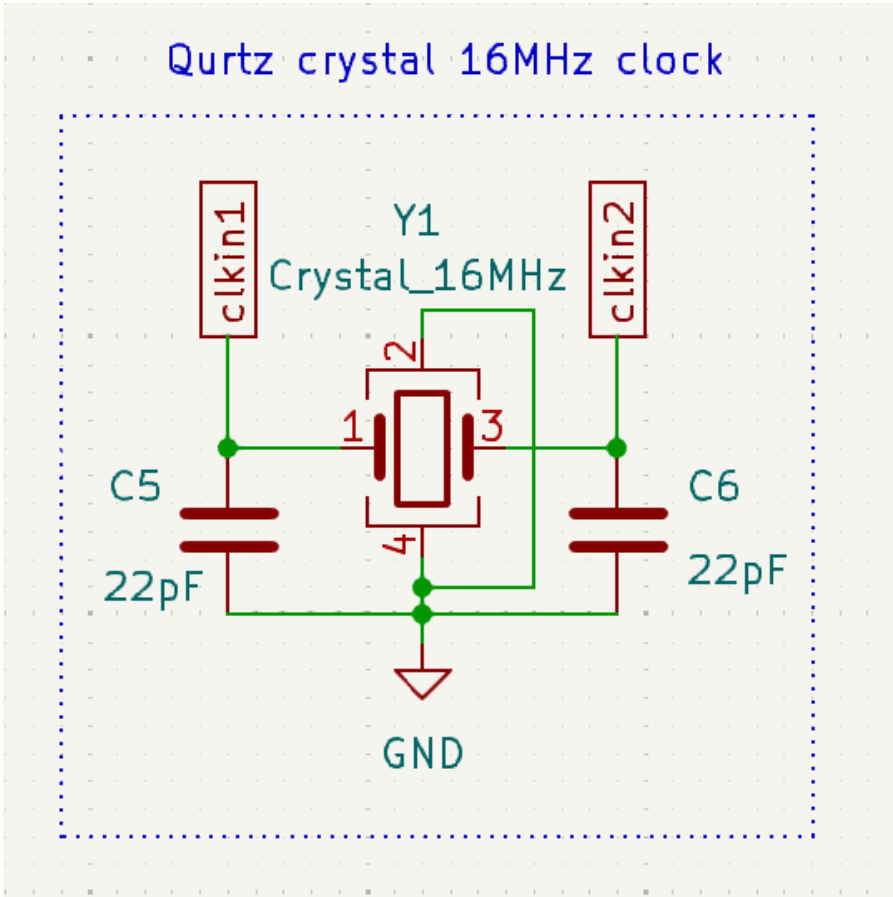
Reference: https://www.hackster.io/simon-mugo/simple-standalone-atmega328p-microcontroller-72efcc
https://www.youtube.com/watch?v=aVUqaB0IMh4

Power supply



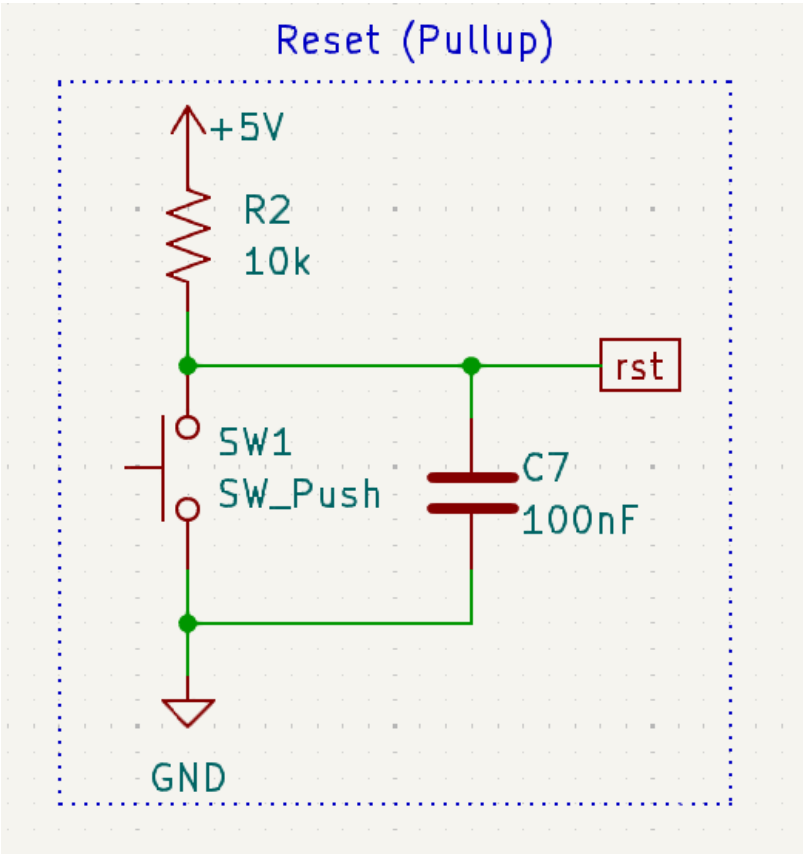
https://www.electrothinks.com/2023/05/ams1117-dc-3-3v-regulator-module.html

Oscillator Circuit (for clock)



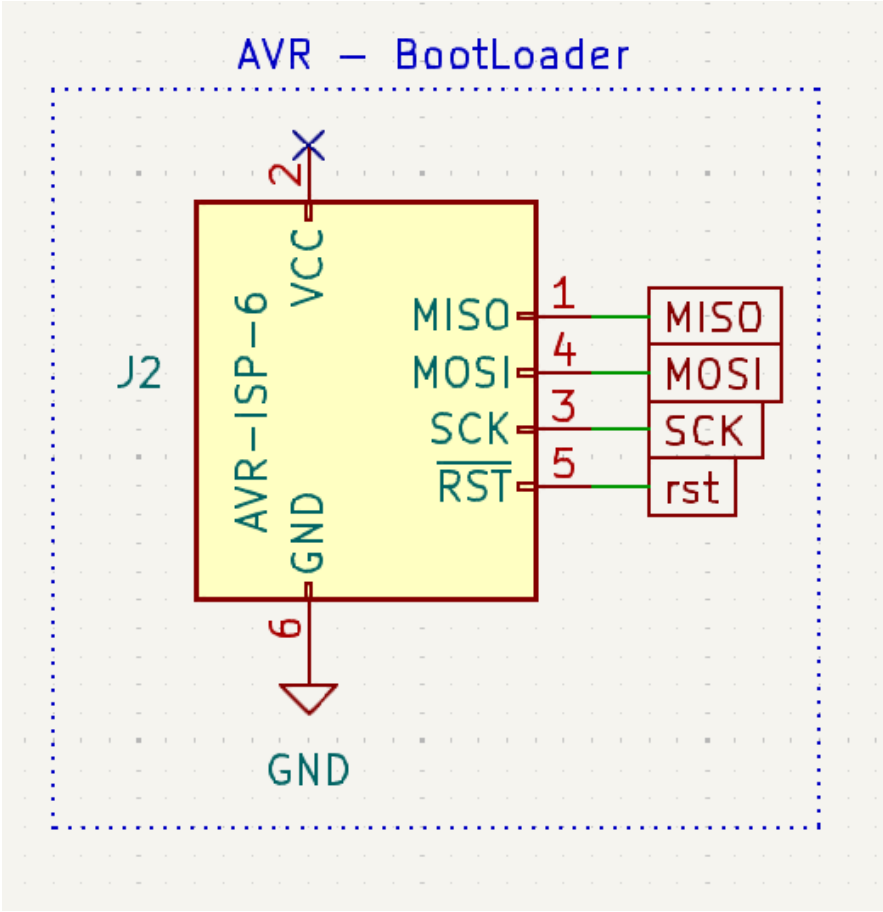
https://www.electronics-tutorials.ws/oscillator/crystal.html

Push Button (for reset signal)



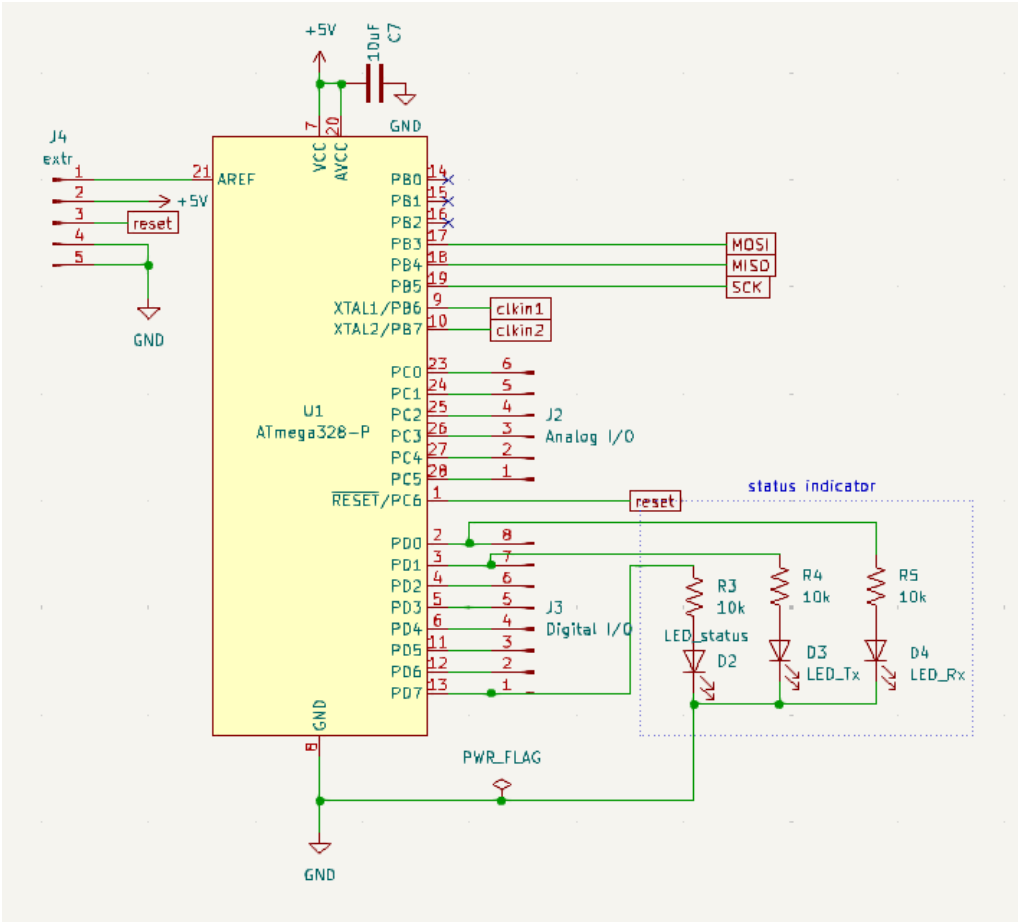
https://www.geeksforgeeks.org/switch-debounce-in-digital-circuits/

AVR ISP (program connector)



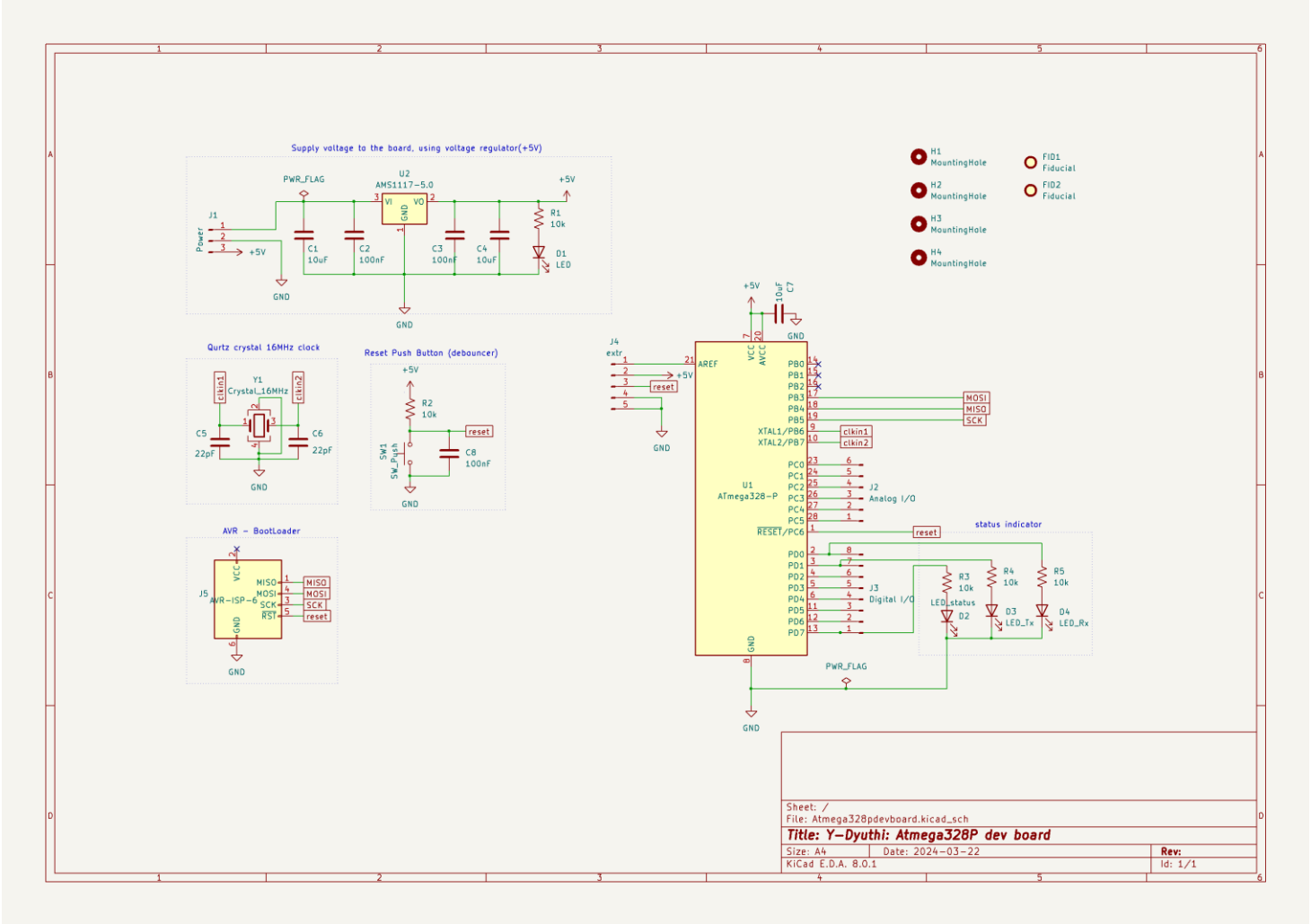
https://docs.arduino.cc/built-in-examples/arduino-isp/ArduinoISP/

Atmega328P



https://ww1.microchip.com/downloads/en/DeviceDoc/Atmel-7810-Automotive-Microcontrollers-ATmega328P_Datasheet.pdf

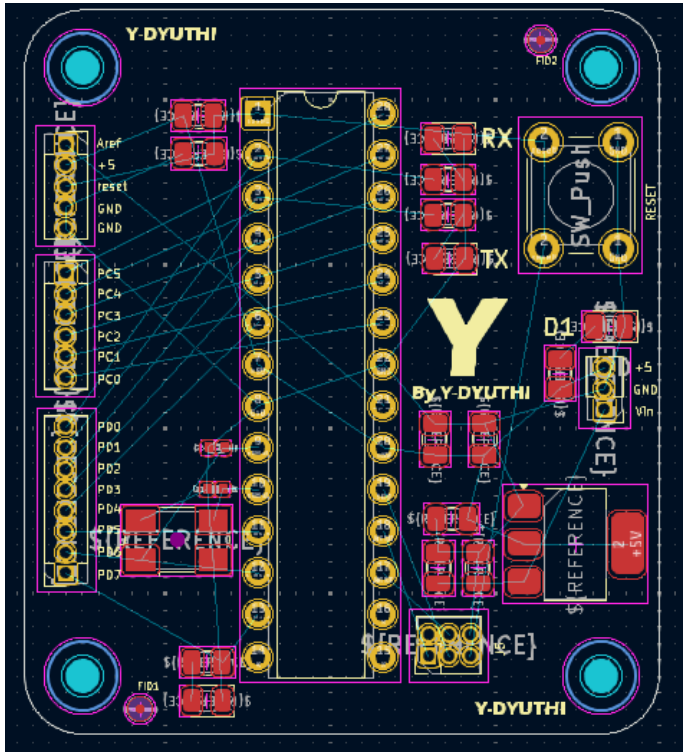
The schematic



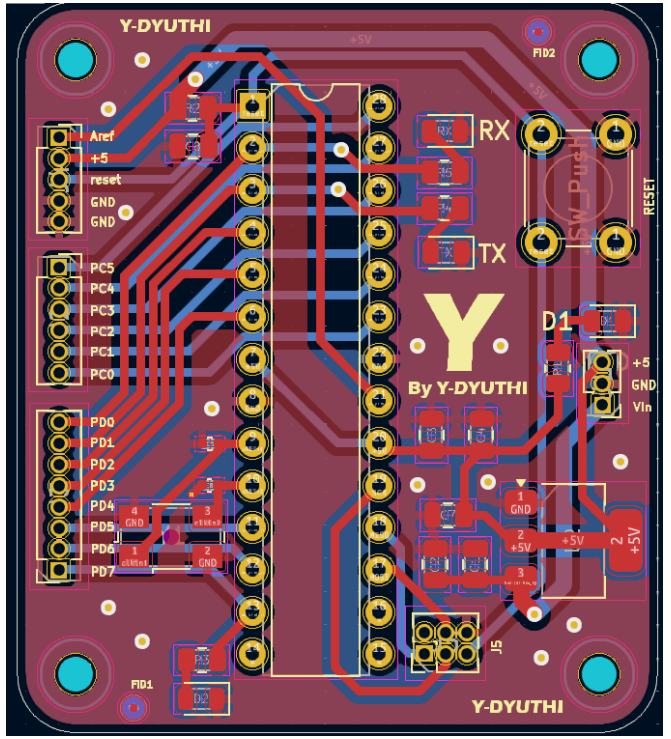
Footprint assignment

1	C1	10uF	Capacitor_SMD:C_0805_2012Metric	
2	C2	100nF	Capacitor_SMD:C_0805_2012Metric	SMD 0805 package Capacitor
3	C3	100nF	Capacitor_SMD:C_0805_2012Metric	
4	C4	10uF	Capacitor_SMD:C_0805_2012Metric	
5	C5	22pF	Capacitor_SMD:C_0402_1005Metric	
6	C6	22pF	Capacitor_SMD:C_0402_1005Metric	SMD 0402 package Capacitor
7	C7	10uF	Capacitor_SMD:C_0805_2012Metric	
8	C8	100nF	Capacitor_SMD:C_0805_2012Metric	
9	D1	LED	LED_SMD:LED_0805_2012Metric	
10	D2	LED_status	LED_SMD:LED_0805_2012Metric	SMD 0805 package LED
11	D3	LED_Tx	LED_SMD:LED_0805_2012Metric	
12	D4	LED_Rx	LED_SMD:LED_0805_2012Metric	
13	FID1	Fiducial	Fiducial:Fiducial_0.5mm_Mask1.5mm	
14	FID2	Fiducial	Fiducial:Fiducial_0.5mm_Mask1.5mm	Fiducials
15	H1	MountingHole	MountingHole:MountingHole_2.1mm	
16	H2	MountingHole	MountingHole:MountingHole_2.1mm	
17	H3	MountingHole	MountingHole:MountingHole_2.1mm	Mounting Hole
18	H4	MountingHole	MountingHole:MountingHole_2.1mm	
19	J1	Power	Connector_PinHeader_1.27mm:PinHeader_1x03_P1.27mm_Vertical	
20	J2	Analog I/O	Connector_PinSocket_1.27mm:PinSocket_1x06_P1.27mm_Vertical	
21	J3	Digital I/O	Connector_PinSocket_1.27mm:PinSocket_1x08_P1.27mm_Vertical	Pin socket with pitch 1.27mm
22	J4	extr	Connector_PinSocket_1.27mm:PinSocket_1x05_P1.27mm_Vertical	
23	J5	AVR-ISP-6	Connector_PinHeader_1.27mm:PinHeader_2x03_P1.27mm_Vertical	
24	R1	10k	Resistor_SMD:R_0805_2012Metric	
25	R2	10k	Resistor_SMD:R_0805_2012Metric	
26	R3	10k	Resistor_SMD:R_0805_2012Metric	
27	R4	10k	Resistor_SMD:R_0805_2012Metric	SMD 0805 package
28	R5	10k	Resistor_SMD:R_0805_2012Metric	
29	SW1	SW_Push	Button_Switch_THT:SW_PUSH_6mm	
30	U1	ATmega328-P	Package_DIP:DIP-28_W7.62mm	
31	U2	AMS1117-5.0	Package_TO_SOT_SMD:SOT-223-3_TabPin2	
32	Y1	Crystal_16MHz	Crystal:Crystal_SMD_0603-4Pin_6.0x3.5mm	

Placement & Layout

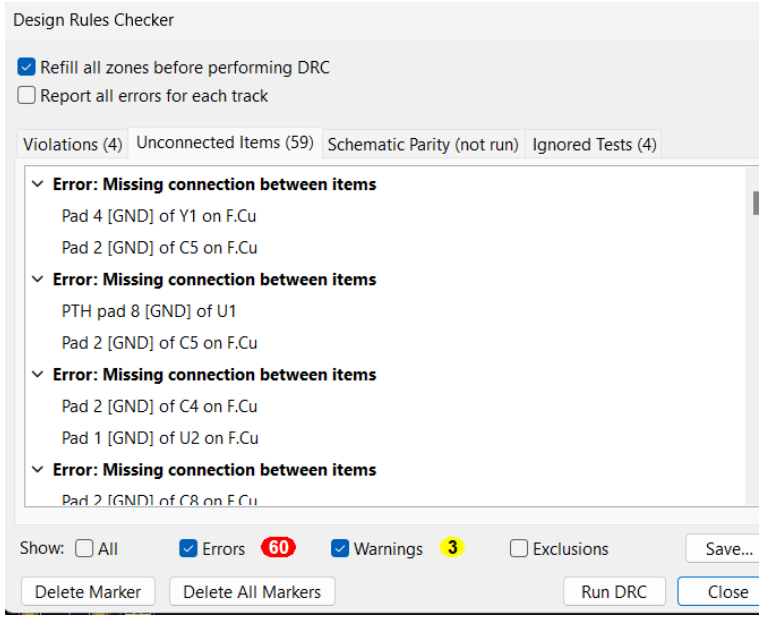


Place the components, draw the border on **edge cut layer**, add markings (eg : +5V , GND, reset etc.) in the **F.silkscreen**, remove unnecessary text.



Set some track widths (**0.2mm, 0.3mm, 0.5mm**) and **via diameters (0.3,0.5,1 mm, hole size 0.2, 0.4, 0.5 mm)** and start routing. Switch between **F.Cu, B.Cu layers** for connecting traces. After this draw a fill zone and fill it with GND plane. (Note: No need to connect GND while routing. Zone filling will take care of that)

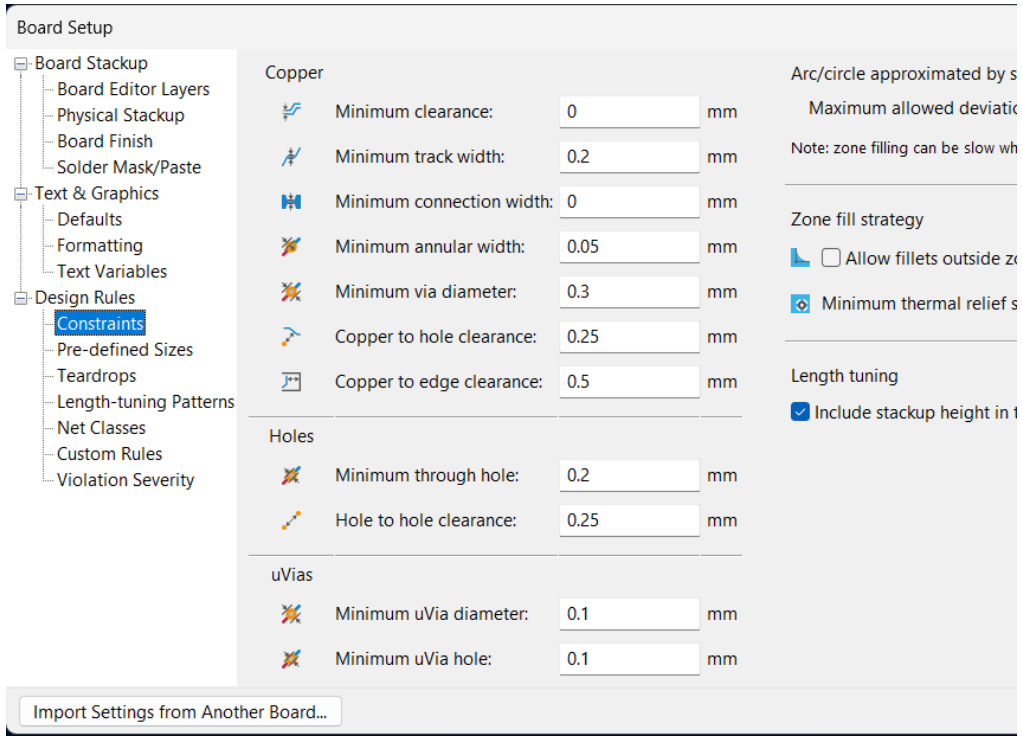
Run DRC (Design Rule Check)



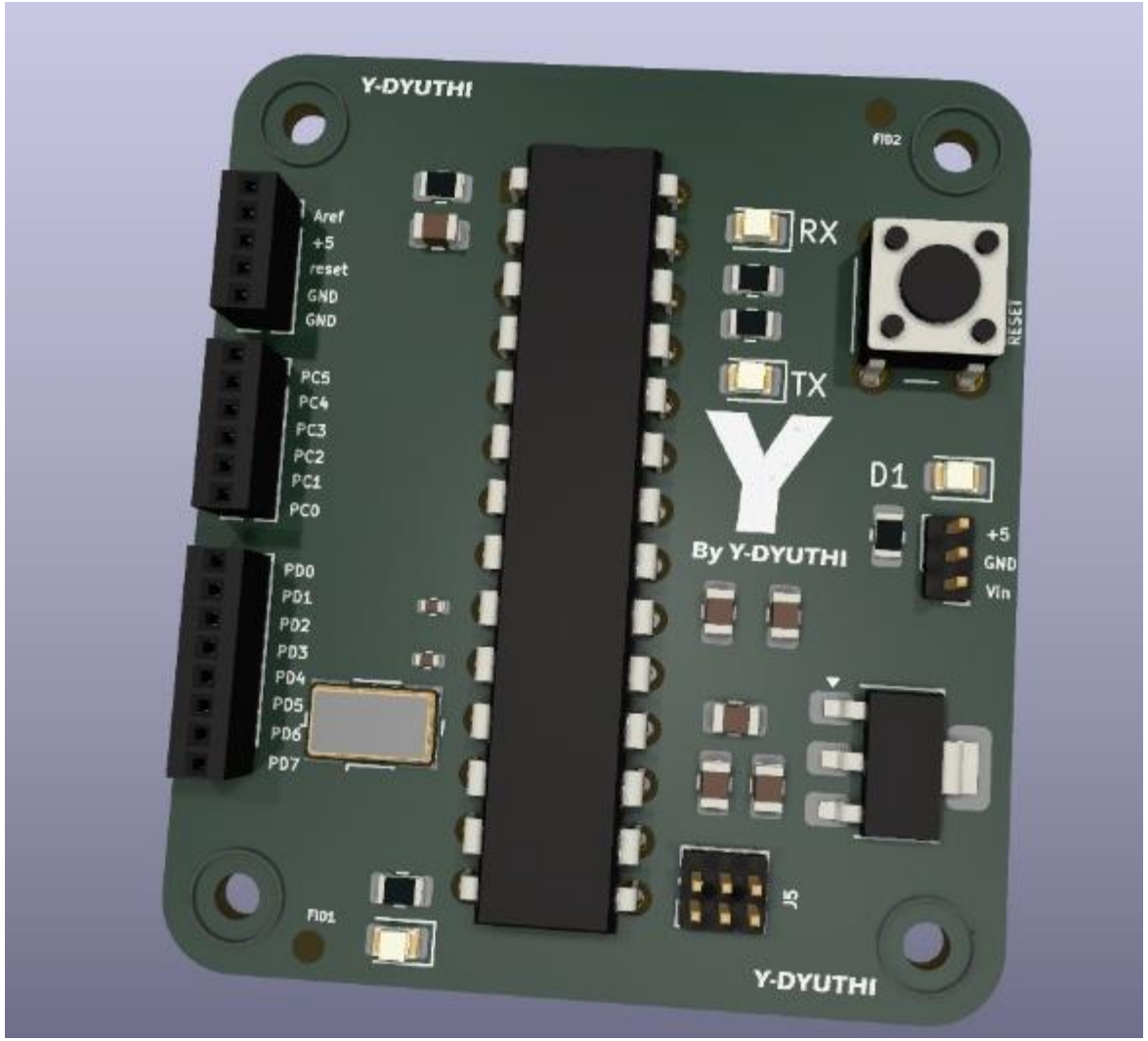
Note that there will be error while running the DRC . Most common mistakes are unconnected traces, and zone getting isolated (you can fix it by connecting the zone to the nearest zone or by using via to connect to the bottom plane). Other exceptions the DRC will generate are mostly violation of Design constraints.

You can edit the design constraints to your specification to remove these errors.

Go to File – Board setup



Voila !!


Y-DYUTHI
Aref
+5
reset
GND
GND
PC5
PC4
PC3
PC2
PC1
PC0
PD0
PD1
PD2
PD3
PD4
PD5
PD6
PD7
RX
TX
By Y-DYUTHI
D1
+5
GND
Vin
RESET
J5
Y-DYUTHI