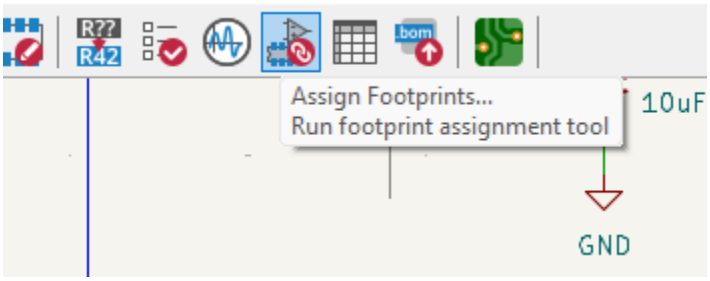
Layout Reference Guide

Before moving onto layout, make sure all your symbols have an assigned footprint.

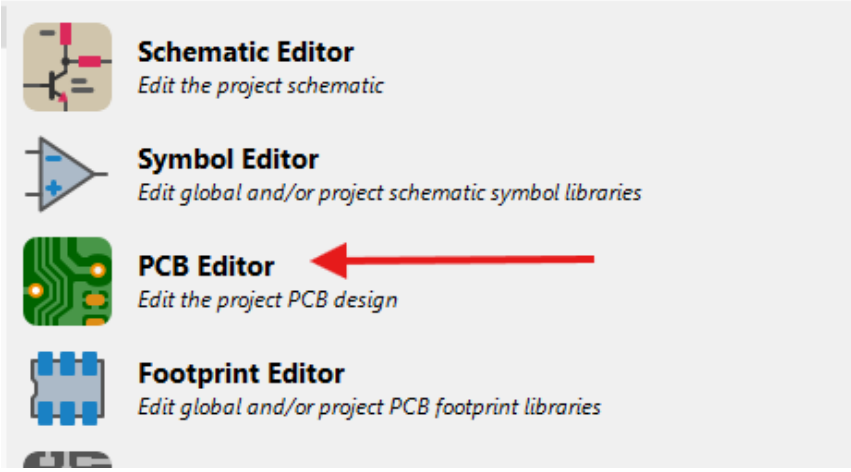


Your Assignments should look something similar to this, a symbol should have a matching footprint, check the given STM32_PCB_Workshop_Design.xlsx for finding the right footprints. **You should not need to import any symbol or footprint for this project.**

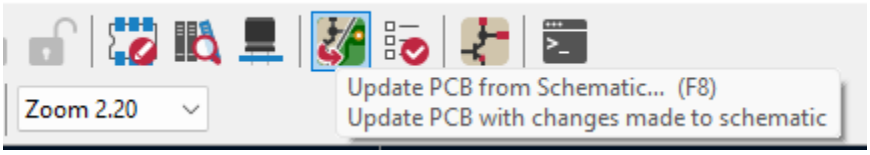
Symbol : Footprint Assignments

1	C1	100nF	Capacitor_SMD:C_1206_3216Metric_Pad1.33x1.80mm_HandSolder
2	C2	10uF	Capacitor_SMD:C_1206_3216Metric_Pad1.33x1.80mm_HandSolder
3	C3	22uF	Capacitor_SMD:C_1206_3216Metric_Pad1.33x1.80mm_HandSolder
4	C4	22uF	Capacitor_SMD:C_1206_3216Metric_Pad1.33x1.80mm_HandSolder
5	C5	100nF	Capacitor_SMD:C_1206_3216Metric_Pad1.33x1.80mm_HandSolder
6	C6	10u	Capacitor_SMD:C_1206_3216Metric_Pad1.33x1.80mm_HandSolder
7	C7	100n	Capacitor_SMD:C_1206_3216Metric_Pad1.33x1.80mm_HandSolder
8	C8	100n	Capacitor_SMD:C_1206_3216Metric_Pad1.33x1.80mm_HandSolder
9	C9	100n	Capacitor_SMD:C_1206_3216Metric_Pad1.33x1.80mm_HandSolder
10	C10	100n	Capacitor_SMD:C_1206_3216Metric_Pad1.33x1.80mm_HandSolder
11	C11	100n	Capacitor_SMD:C_1206_3216Metric_Pad1.33x1.80mm_HandSolder
12	C12	100n	Capacitor_SMD:C_1206_3216Metric_Pad1.33x1.80mm_HandSolder
13	C15	30 pF	Capacitor_SMD:C_1206_3216Metric_Pad1.33x1.80mm_HandSolder
14	C16	30 pF	Capacitor_SMD:C_1206_3216Metric_Pad1.33x1.80mm_HandSolder
15	D1	RED (FV ~1.8V)	LED_THT:LED_D5.0mm
16	D2	RED (FV ~1.8V)	LED_THT:LED_D5.0mm
17	D3	RED (FV ~1.8V)	LED_THT:LED_D5.0mm
18	D4	RED (FV ~1.8V)	LED_THT:LED_D5.0mm
19	D5	RED (FV ~1.8V)	LED_THT:LED_D5.0mm
20	D6	RED (FV ~1.8V)	LED_THT:LED_D5.0mm
21	D7	RED (FV ~1.8V)	LED_THT:LED_D5.0mm
22	D8	RED (FV ~1.8V)	LED_THT:LED_D5.0mm
23	D9	RED (FV ~1.8V)	LED_THT:LED_D5.0mm
24	D10	RED (FV ~1.8V)	LED_THT:LED_D5.0mm
25	D11	RED (FV ~1.8V)	LED_THT:LED_D5.0mm
26	D12	RED (FV ~1.8V)	LED_THT:LED_D5.0mm
27	D13	RED (FV ~1.8V)	LED_THT:LED_D5.0mm
28	D14	RED (FV ~1.8V)	LED_THT:LED_D5.0mm
29	H1	MountingHole	MountingHole:MountingHole_3.2mm_M3_Pad
30	H2	MountingHole	MountingHole:MountingHole_3.2mm_M3_Pad
31	H3	MountingHole	MountingHole:MountingHole_3.2mm_M3_Pad
32	H4	MountingHole	MountingHole:MountingHole_3.2mm_M3_Pad
33	J1	USB_B	Connector_USB:USB_B_OST_USB-B1HSxx_Horizontal
34	J2	Screw_Terminal_01x02	TerminalBlock:TerminalBlock_MaiXu_MX126-5.0-02P_1x02_P5.00mm
35	J3	Conn_01x04_Pin	Connector_PinHeader_2.54mm:PinHeader_1x04_P2.54mm_Vertical
36	L1	3.9uH	Inductor_SMD:L_Bourns_SDR0604
37	R1	1k5	Resistor_SMD:R_1206_3216Metric_Pad1.30x1.75mm_HandSolder
38	R2	10k	Resistor_SMD:R_1206_3216Metric_Pad1.30x1.75mm_HandSolder
39	R3	470	Resistor_SMD:R_1206_3216Metric_Pad1.30x1.75mm_HandSolder
40	R4	470	Resistor_SMD:R_1206_3216Metric_Pad1.30x1.75mm_HandSolder
41	R5	470	Resistor_SMD:R_1206_3216Metric_Pad1.30x1.75mm_HandSolder
42	R6	470	Resistor_SMD:R_1206_3216Metric_Pad1.30x1.75mm_HandSolder
43	R7	470	Resistor_SMD:R_1206_3216Metric_Pad1.30x1.75mm_HandSolder
44	R8	470	Resistor_SMD:R_1206_3216Metric_Pad1.30x1.75mm_HandSolder

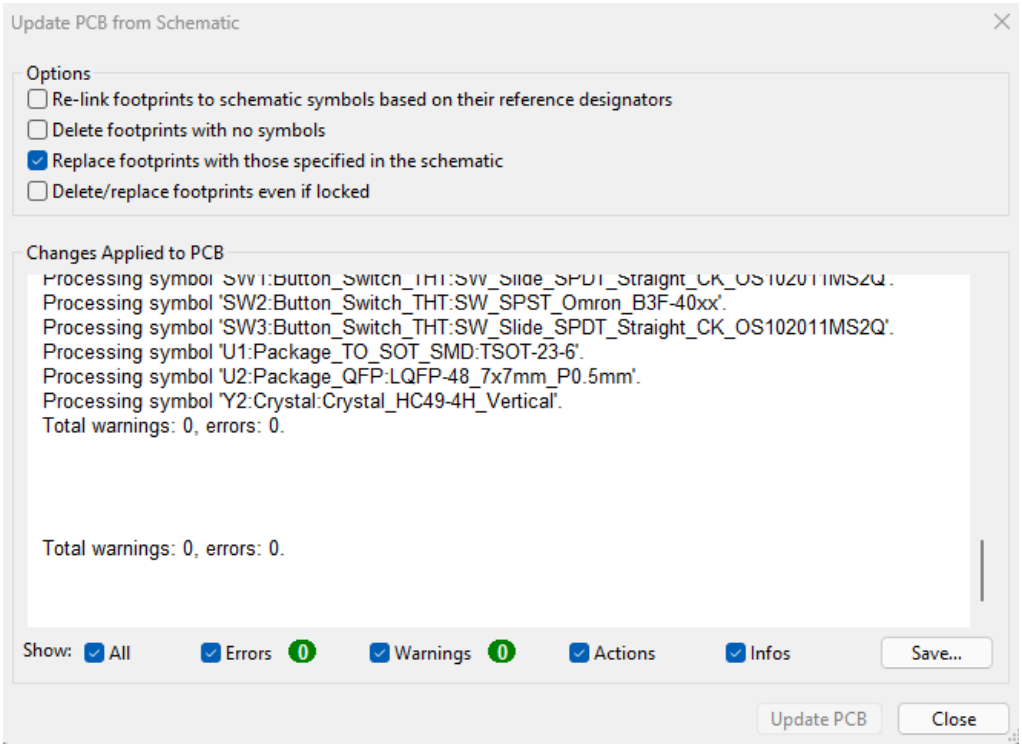
Now if you are ready for Layout, click PCB Editor in the Project window.



This should open up a new window with a blank grid space. To import your footprints click the Update PCB from Schematic window up top.

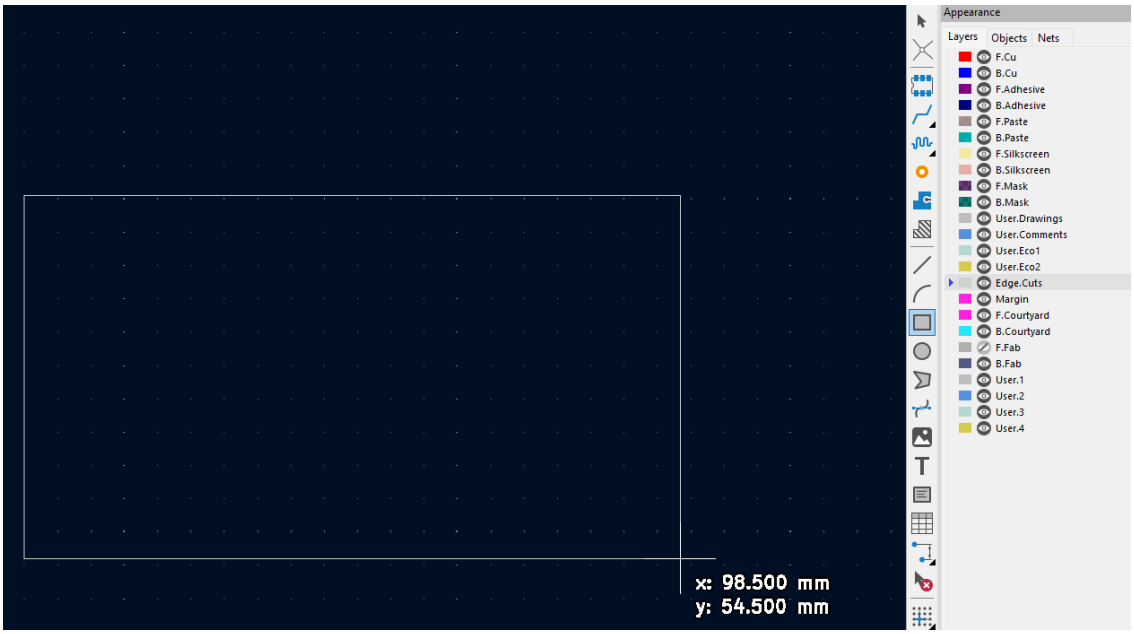


Click on Update PCB, you will see all your footprints will appear. If you have no errors, you are good to move onto layout, if there are errors, those will need to be fixed before moving on.

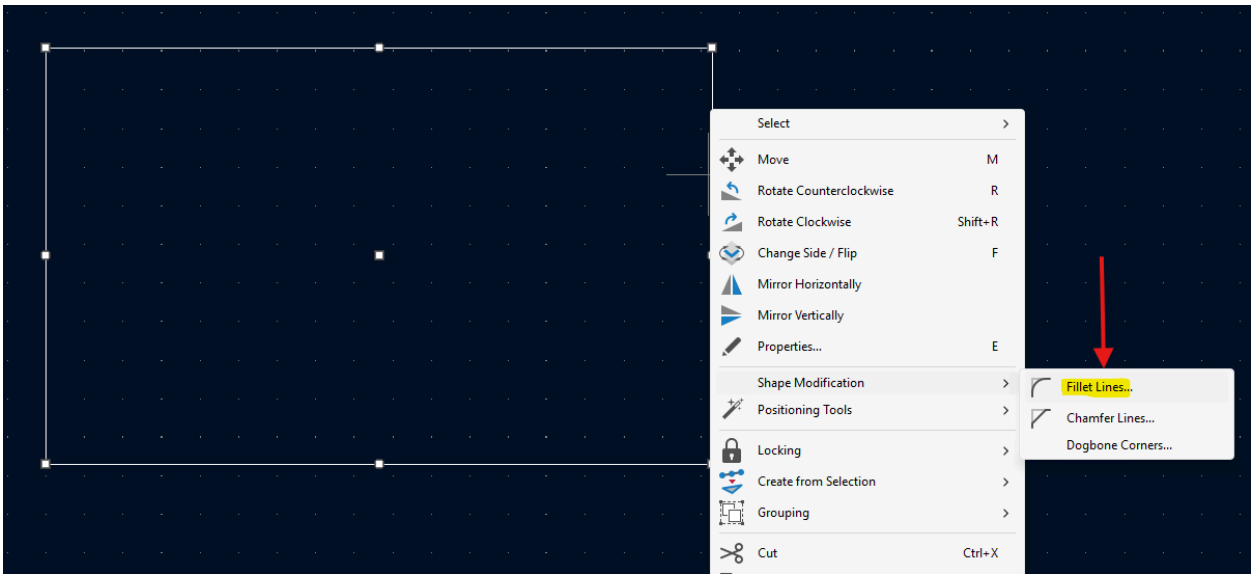


Time for Layout!

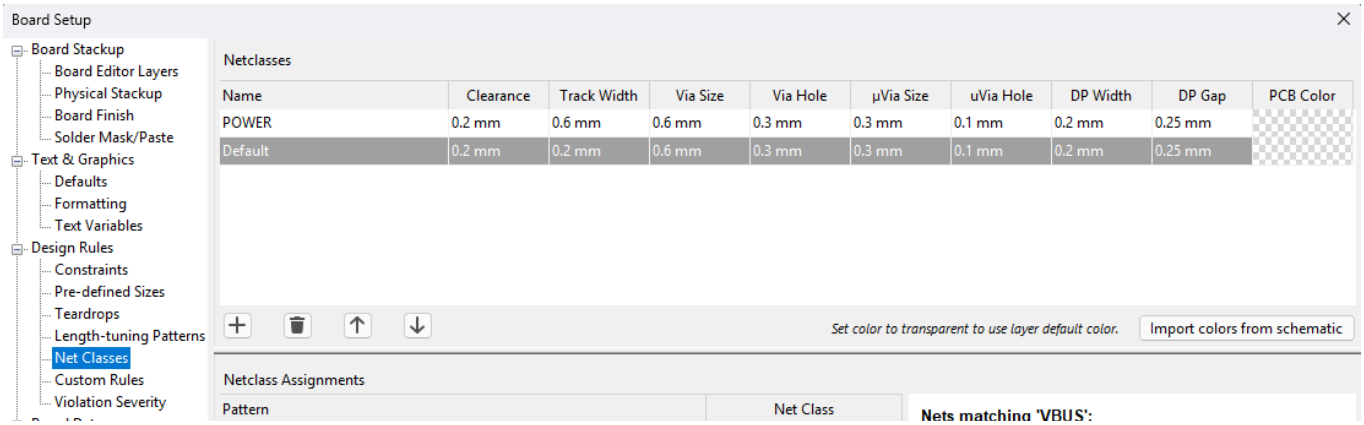
Start off with your Edgecuts and this will be defining the board size, for our purpose make sure your board is less than 100mm x 100mm.

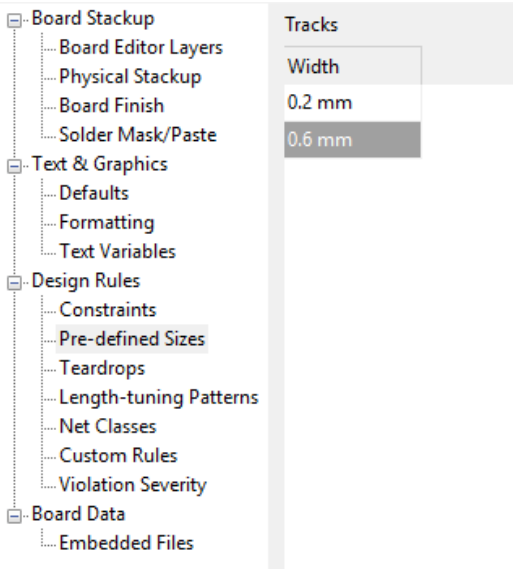


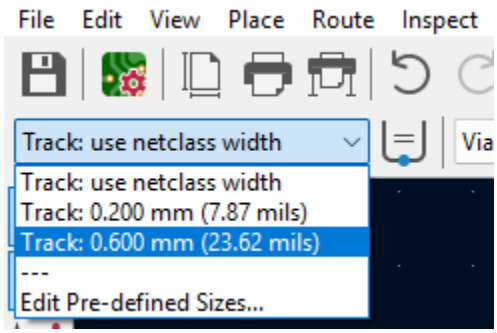
This step is optional, but good practice is to round your corners by selecting Fillet Lines, 4mm should be a sufficient size.



One last thing before we start connecting all of our components is to either define your **netclasses** or make **Pre-defined Sizes**. The point of this is to set what our trace widths need to be for certain connections, for most signal lines, having the default value of 0.2mm will be sufficient. However for power sections, or areas with a high amount of current flowing through we want thicker traces.





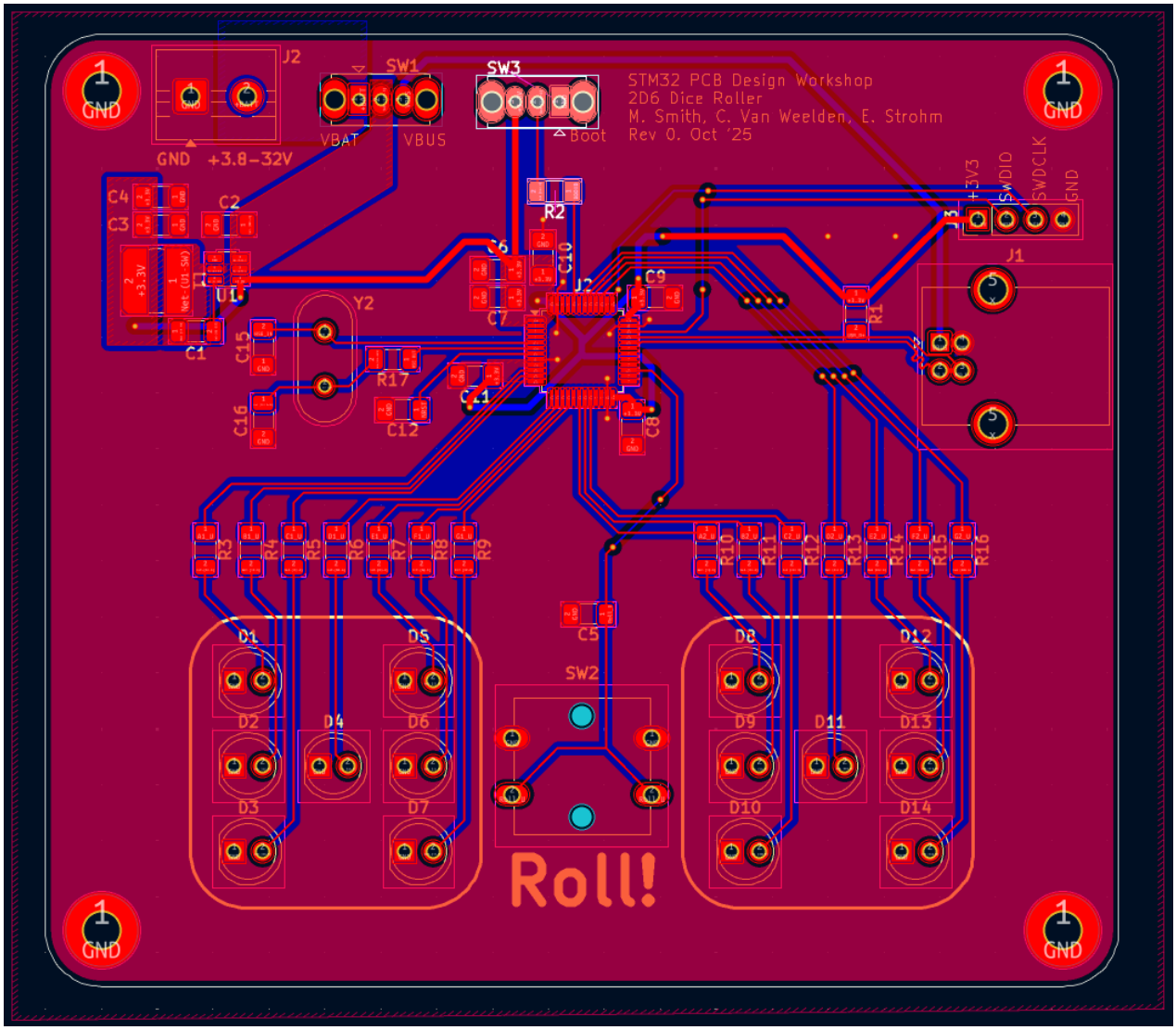


If you are interested in knowing the exact value to choose for trace widths, DigiKey has a calculator reference: [PCB Trace Width Conversion Calculator | DigiKey]

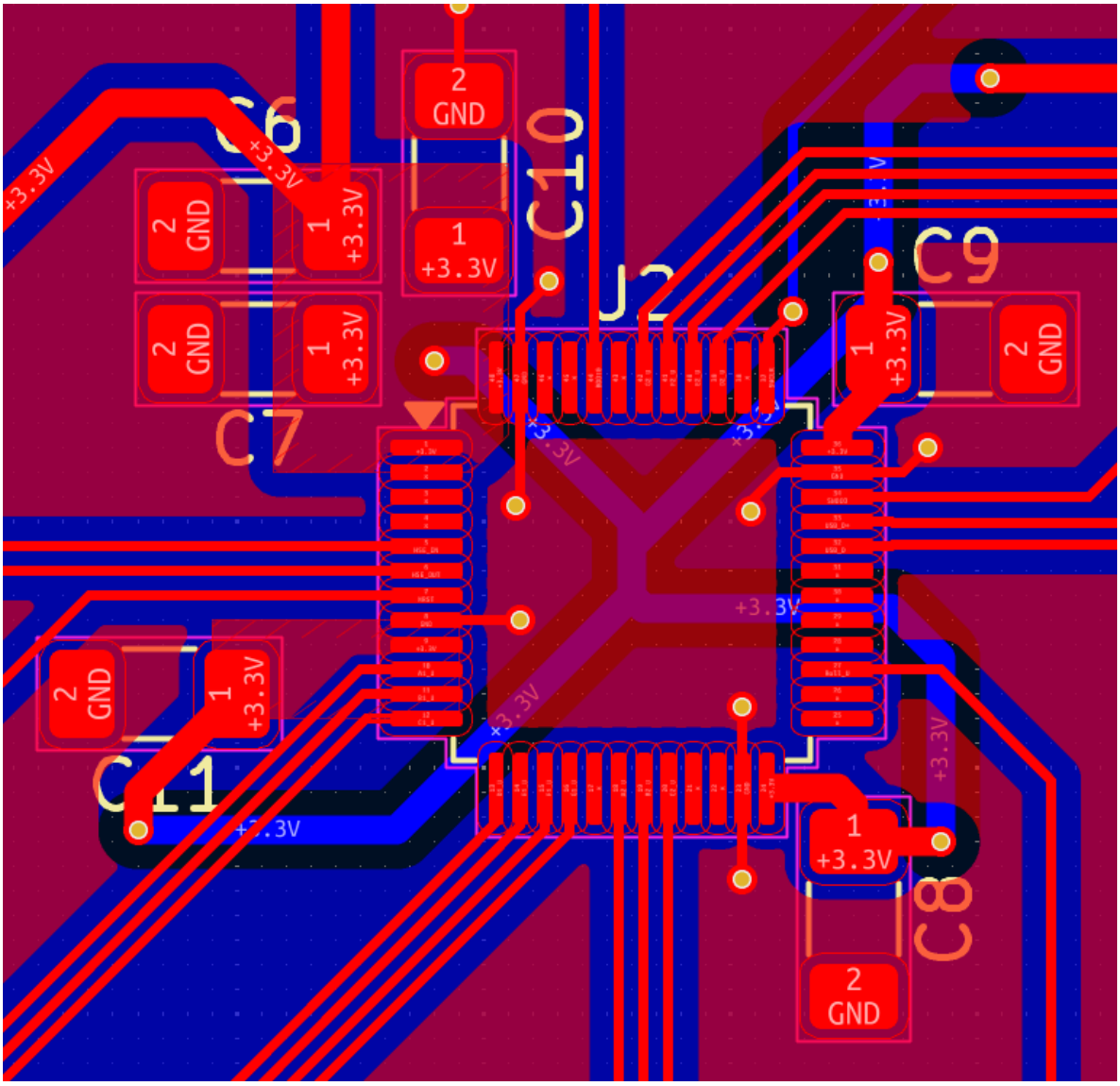
General rule of thumb is go as thick as you can for known high current traces.

Reference Layout

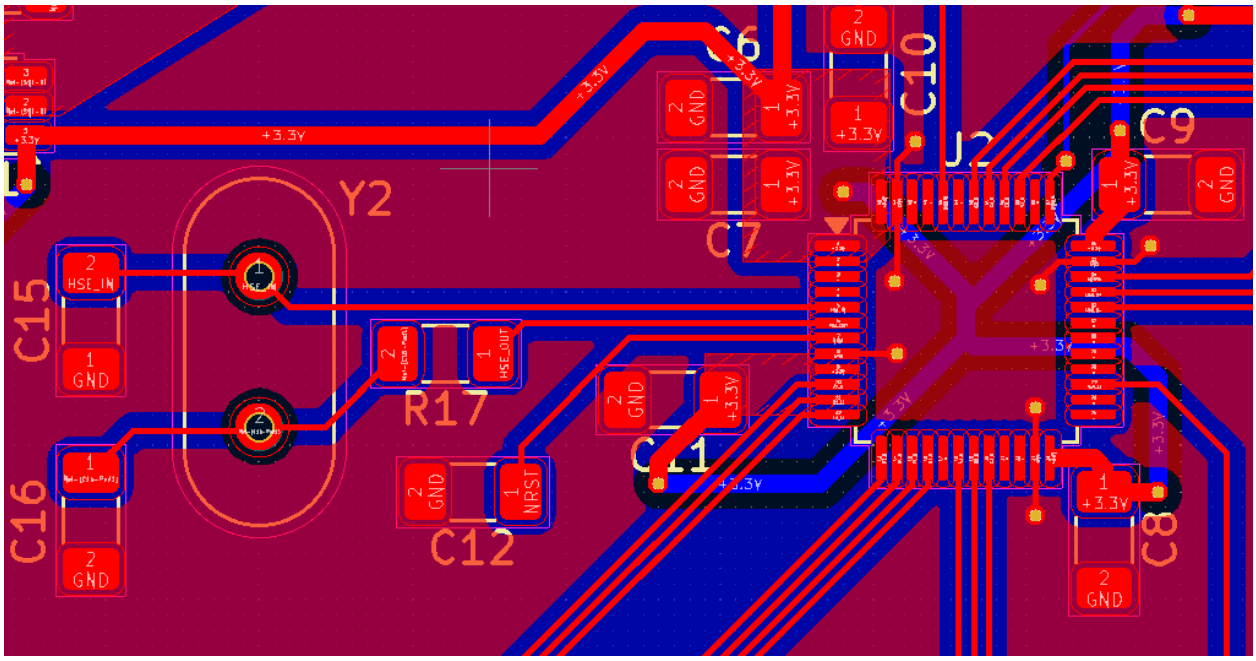
At this point, you have the freedom of laying out the components however you would like. There are some general guidelines in layout we recommend to prevent signal integrity issues, performance, and easier chance of soldering components(keep in mind you will end up soldering these components yourself, make your job as easy as possible)

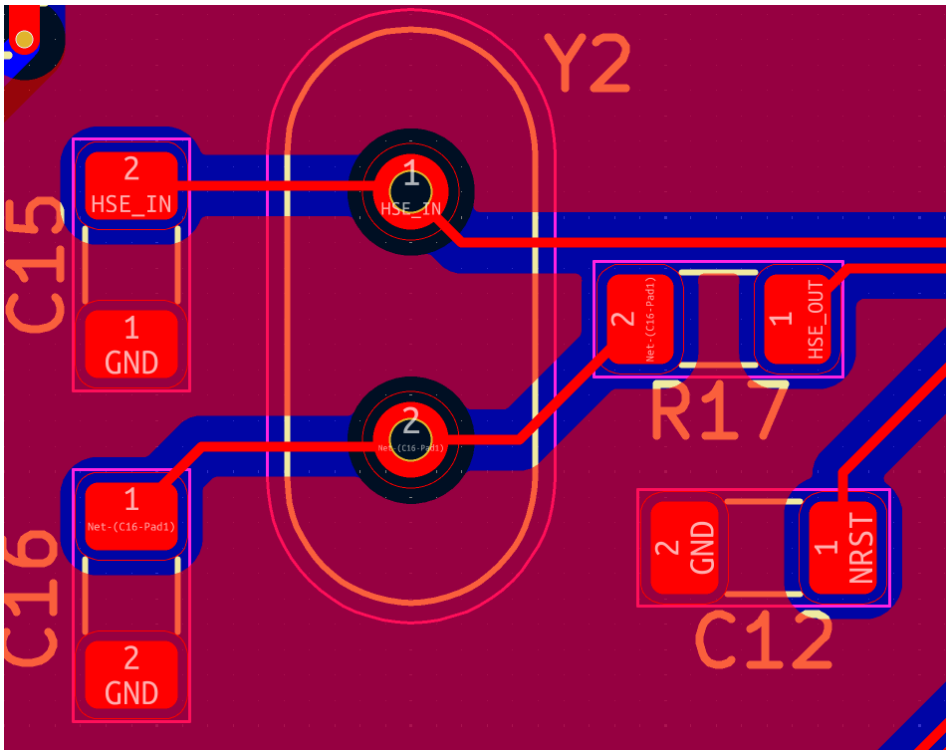


If unsure where to start placing components. Here is our reference design to get started.

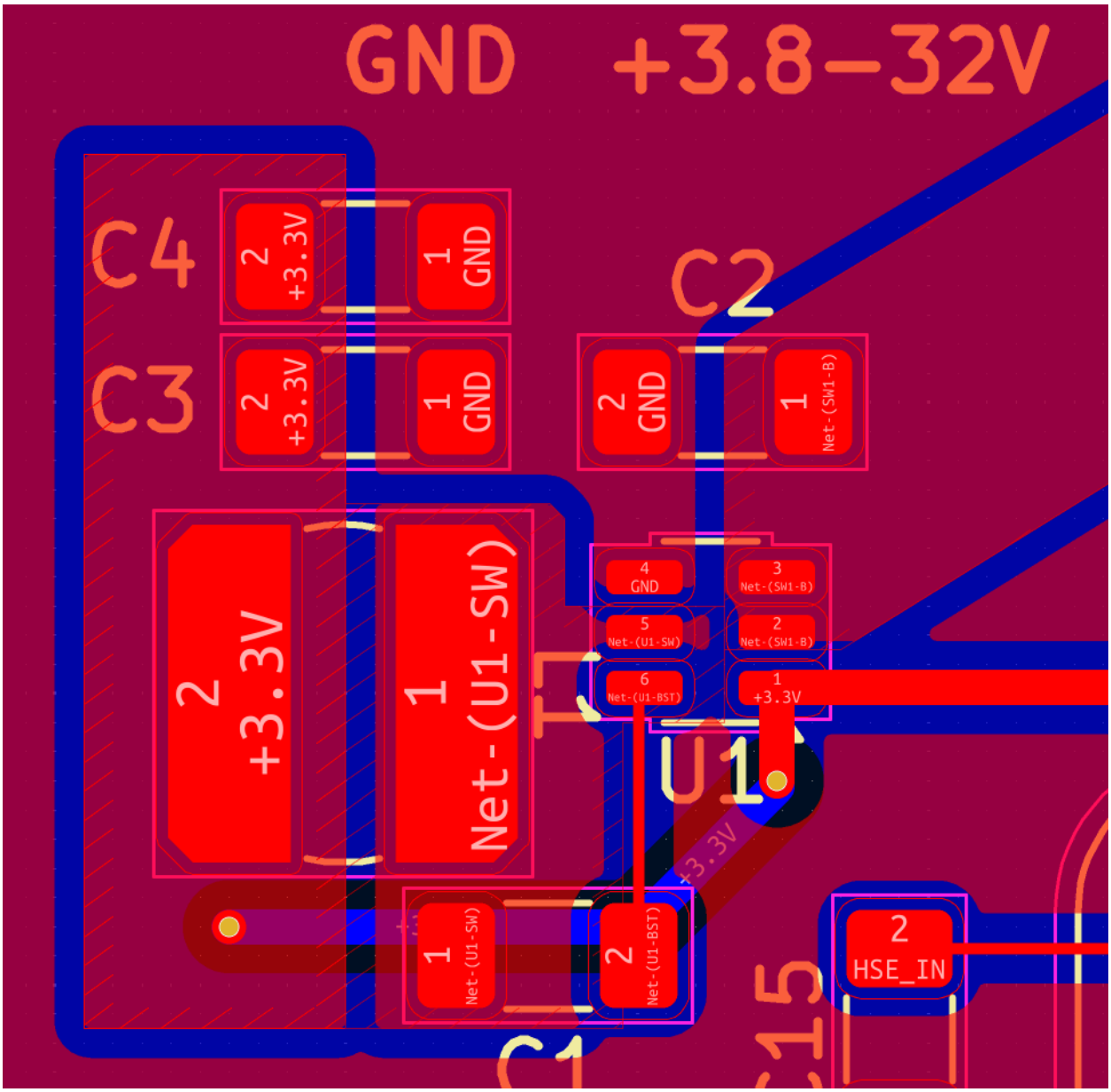


In most designs, you want to start off with your MCU(Micro Controller Unit) in our case is the STM32 board. This will usually be in the **center** of the board for options in routing our signal lines. Make sure your decoupling caps (C6,C7,C8,C9,C10,C11) are close to the +3.3V lines of your MCU.

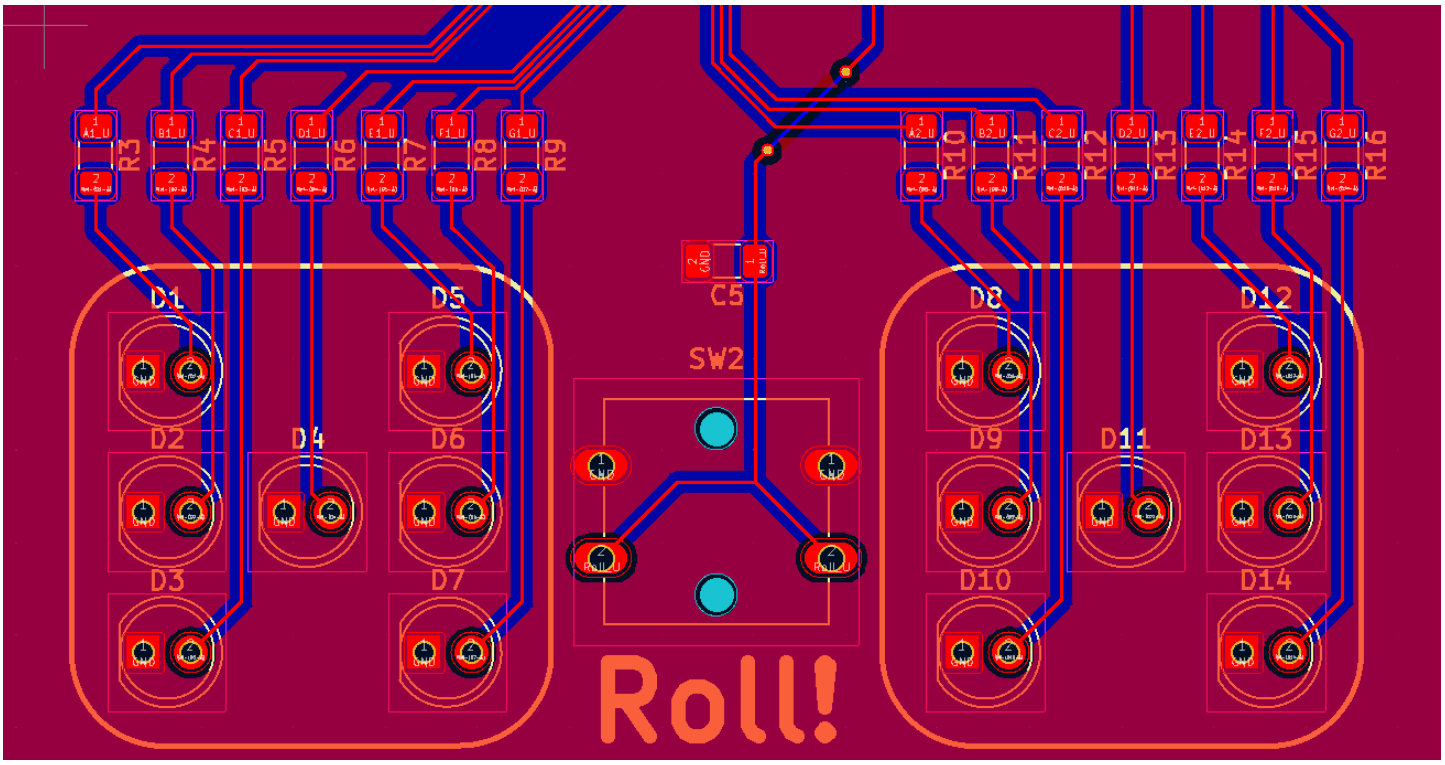




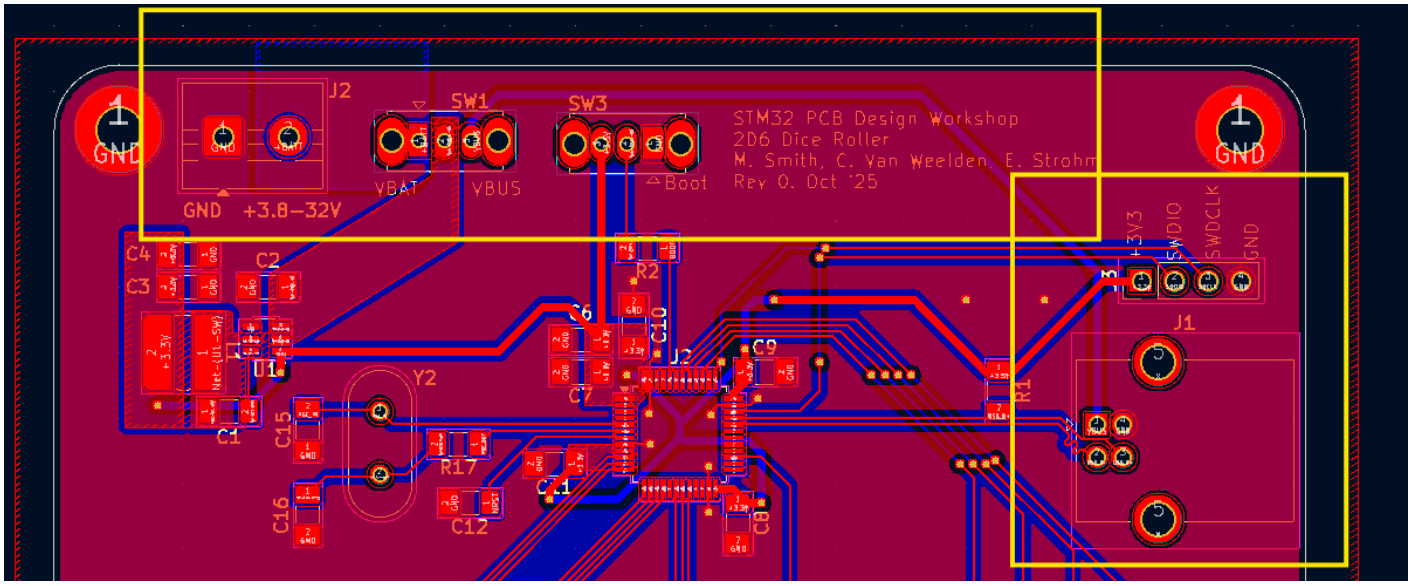
After setting up the Decoupling Caps, we also want our Oscillator close to our MCU, this is a high speed signal at 16MHz in our case, and want to set this circuitry as close to the MCU as we can to prevent clock issues.



Next we can set up the Switching Regulator, again, this layout is based on this recommended datasheet layout from AP63203WU. Something to note is that this Switching regulator is switching around 1.1MHZ and should be put further away from the crystal oscillator if you can or other high speed circuitry.

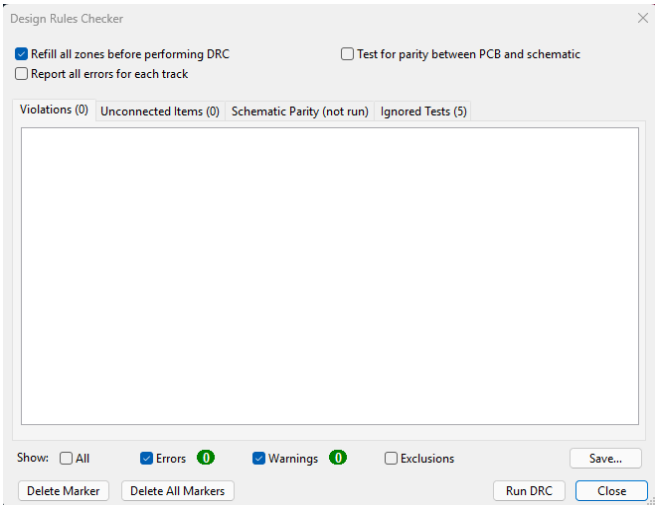


For the LED's try to format them in this way to represent our 2D6 Dice. This will make programming your board a lot easier if you follow our convention.



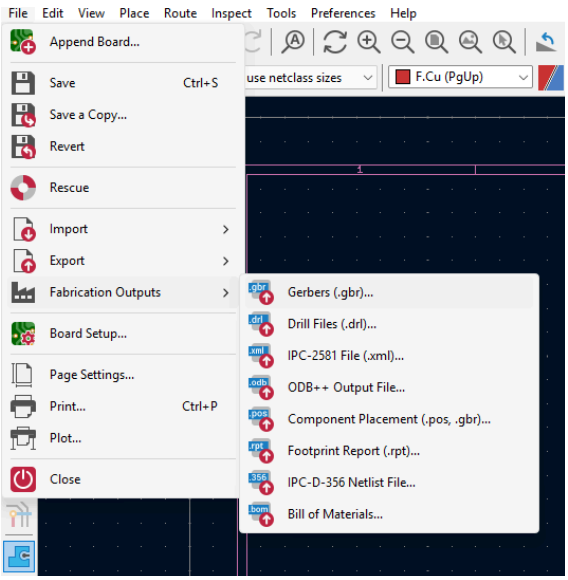
Lastly try to place your connectors at the edge of the board, for the USB we want to avoid the Data Lines being to far away from the MCU, and adding too many bends.

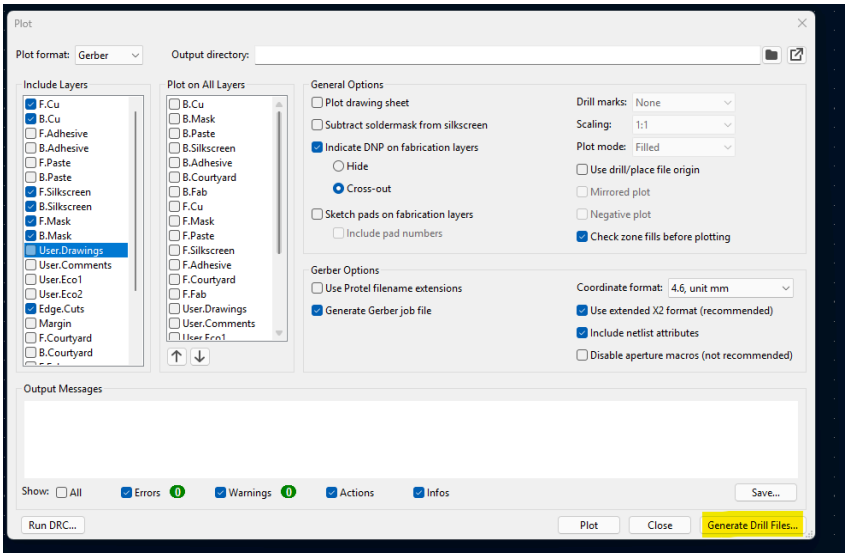
Once you have your components laid out and finished routing, you want to run DRC before moving onto generating gerber files. Make sure you fix any errors you see.



Gerber File & Submission:

Now that your board has passed ERC and DRC without any issues, we can move onto fabricating your board! Click on File -> Fabrication Outputs -> Gerbers. This will take you to a window, I would recommend following the layout shown in the figure below and Generate your Drill Files and Gerbers in the Same Folder





For your board to be fabricated We need a Zip file from you and should include the following:

- Your KiCAD Project with your Schematic(kicad_sch) and Layout(kicad_pcb)
- Your Gerber and Drill Files
- Title your Zip Folder your First and Last Name