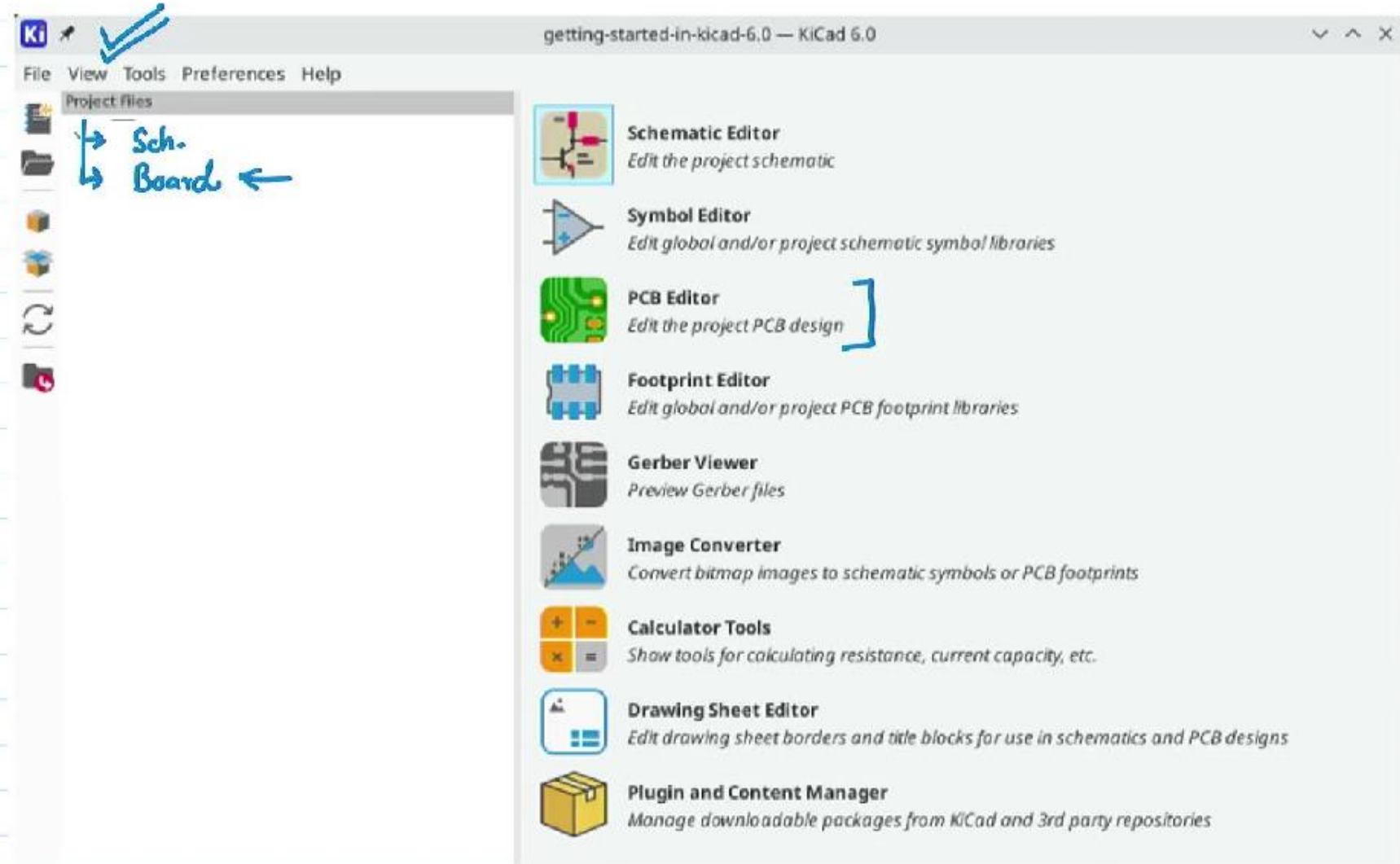


Circuit Board :

With the schematic completed, go back to Project window and open the PCB Editor.



Board Setup and Stack-up:

Before designing the board, set the page size and add information to the title block.

File → Page settings.

The screenshot shows the 'Page Settings' dialog box with two tabs: 'Paper' and 'Drawing Sheet'.

Paper Tab:

- Size:** A 8.5x11in (indicated by an arrow)
- Orientation:** Landscape (indicated by an arrow)
- Custom paper size:**
 - Height: 279.4 mm
 - Width: 431.8 mm
- Preview:** A black rectangular area representing the paper (indicated by an arrow).

Drawing Sheet Tab:

- File:** [Empty text field]
- Title Block:** A blue box highlighting the title block section.
- Issue Date:** 2021-12-18 (with navigation buttons <<< and >>> and a dropdown arrow)
- Revision:** 0 (indicated by an arrow)
- Title:** Getting Started in KiCad 6.0
- Company:** [Empty text field]
- Comment1:** [Empty text field]
- Comment2:** [Empty text field]
- Comment3:** [Empty text field]
- Comment4:** [Empty text field]
- Comment5:** [Empty text field]
- Comment6:** [Empty text field]
- Comment7:** [Empty text field]
- Comment8:** [Empty text field]
- Comment9:** [Empty text field]

At the bottom right are 'Cancel' and 'OK' buttons.

Board Setup and Stack-up

To exit full screen, press Esc

Very important settings.

What copper and dielectric layers PCB will have and their thickness.

Board Setup

Board Stackup
Board Editor Layers
Physical Stackup
Board Finish
Solder Mask/Paste
Text & Graphics
Defaults
Text Variables
Design Rules
Constraints
Pre-defined Sizes
Net Classes
Custom Rules
Violation Severity

Copper layers: 2

☐ Impedance controlled

Add Dielectric Layer... Remove Dielectric Layer...

Layer	Id	Type	Material	Thickness	Color	Epsilon R	Loss Tan
F.Silkscreen		Top Silk Screen	Not specified		Not specified		
F.Paste		Top Solder Paste					
F.Mask		Top Solder Mask	Not specified	0.01 mm	Not specified	3.3	0
F.Cu		Copper		0.035 mm			
Dielectric 1		Core	FR4	1.51 mm		4.5	0.02
B.Cu		Copper		0.035 mm			
B.Mask		Bottom Solder Mask	Not specified	0.01 mm	Not specified	3.3	0
B.Paste		Bottom Solder Paste					
B.Silkscreen		Bottom Silk Screen	Not specified		Not specified		

Board thickness from stackup: 1.6 mm

Adjust Dielectric Thickness

Export to Clipboard

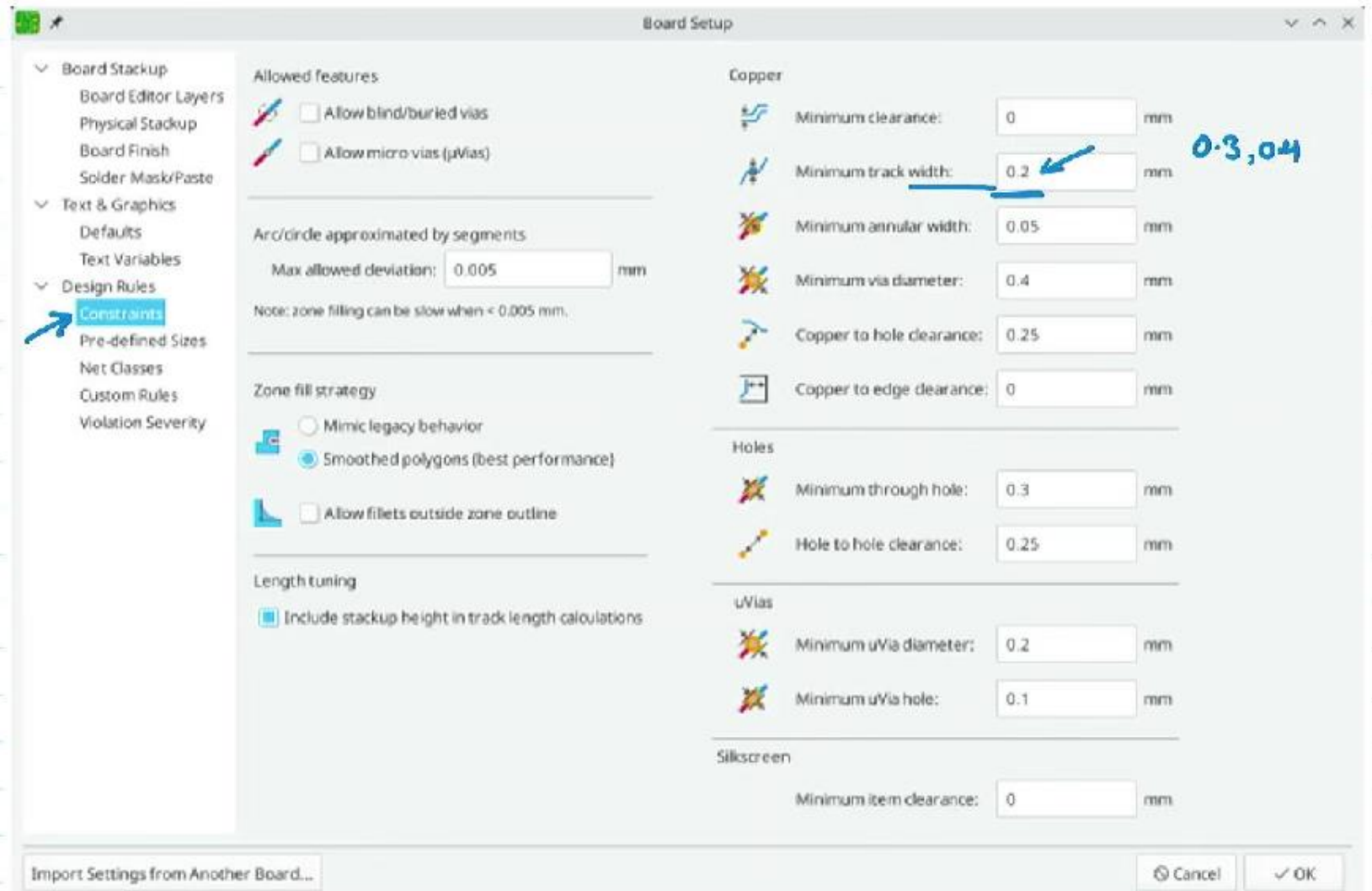
Import Settings from Another Board...

Cancel OK

Design Rules:

Design Rules → Constraints.

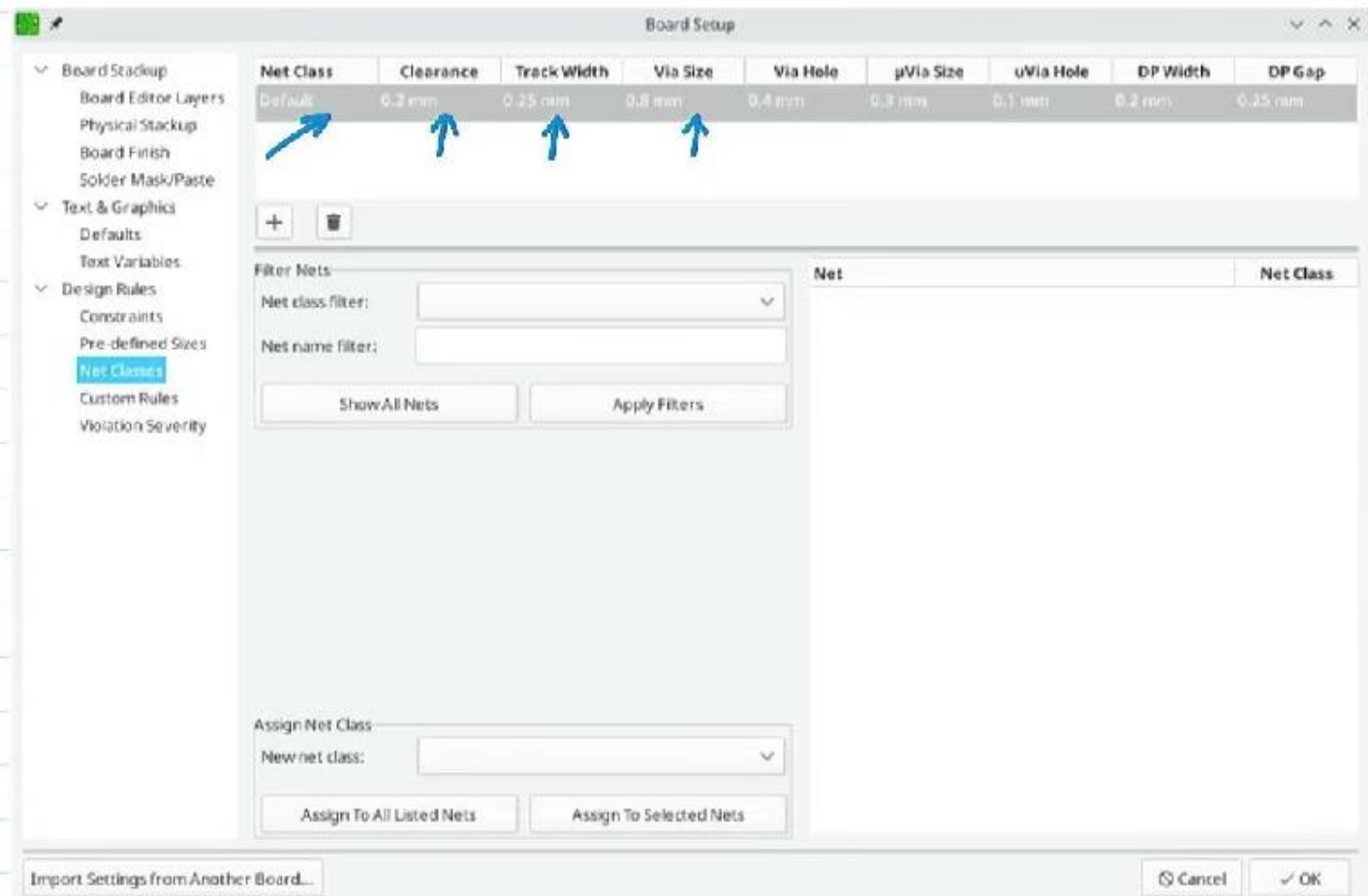
The settings on this page specify the overriding design rules for everything in the board design.



Design rules → Net classes.

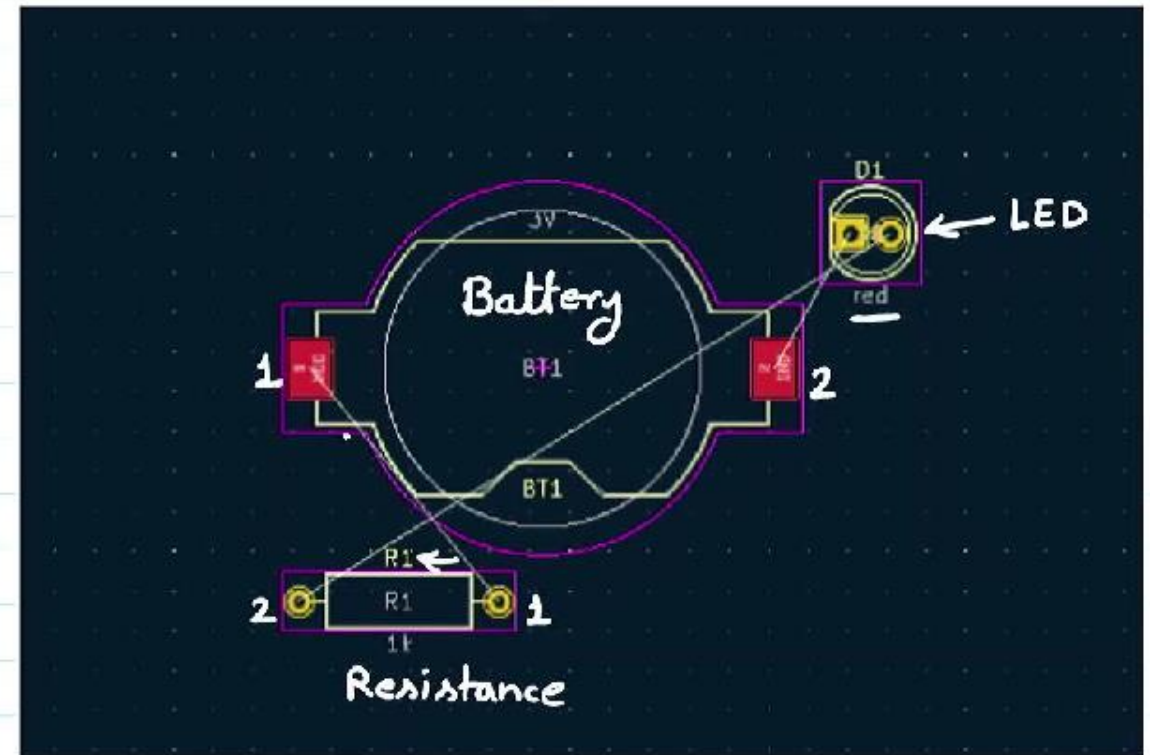
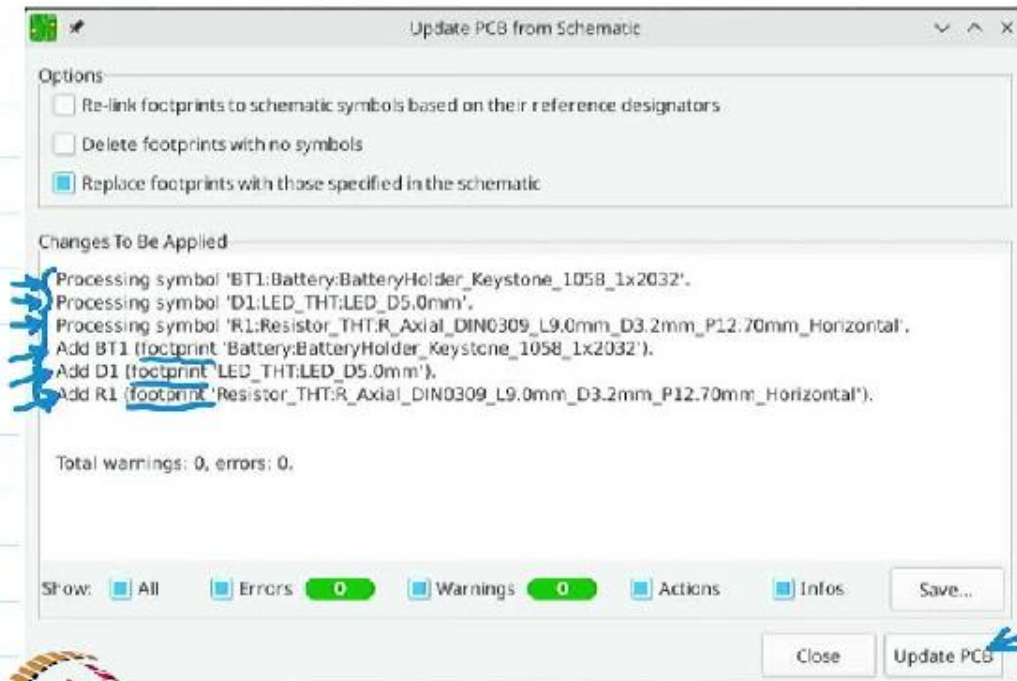
→ The net class is a set of design rules associated with a specific group of nets.

→ This page lists the design rules for each net class in the design and allows assigning nets to each net class.



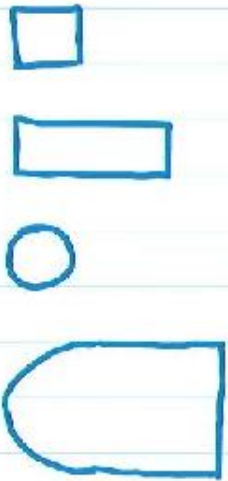
Import Changes from Schematic :

Tools → Update PCB from schematic.

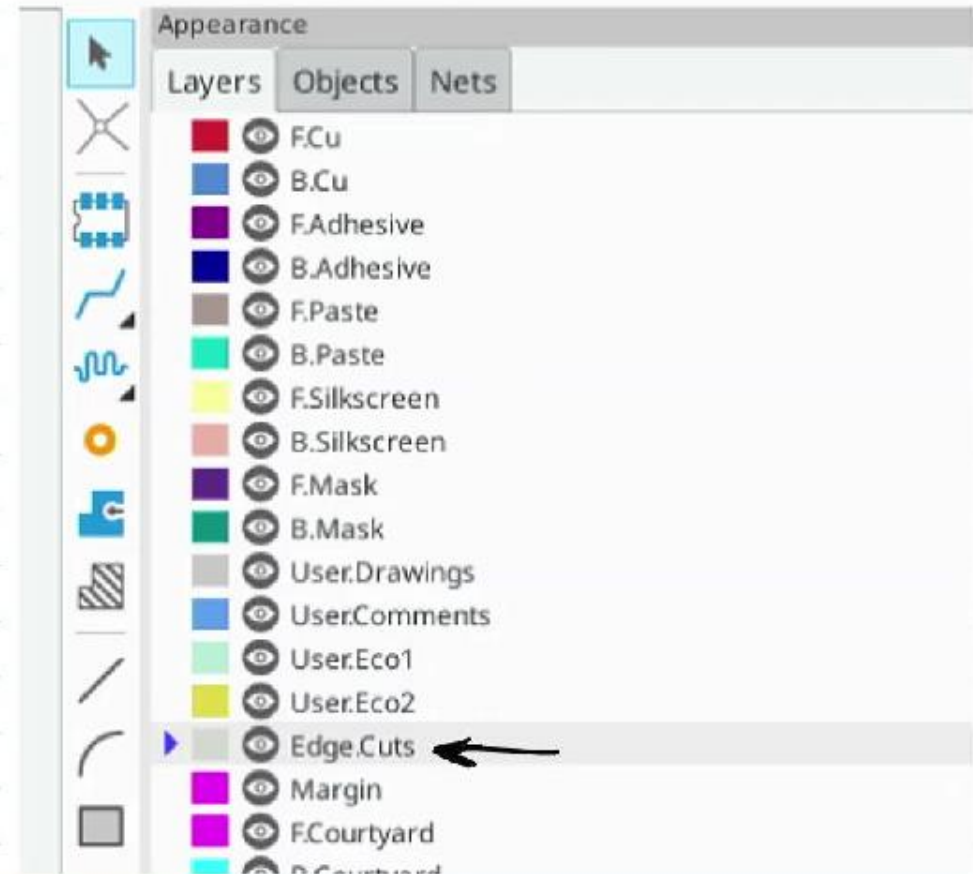


Drawing a board outline :

Click on Edge.cuts in the layers tab and draw the PCB boundry.

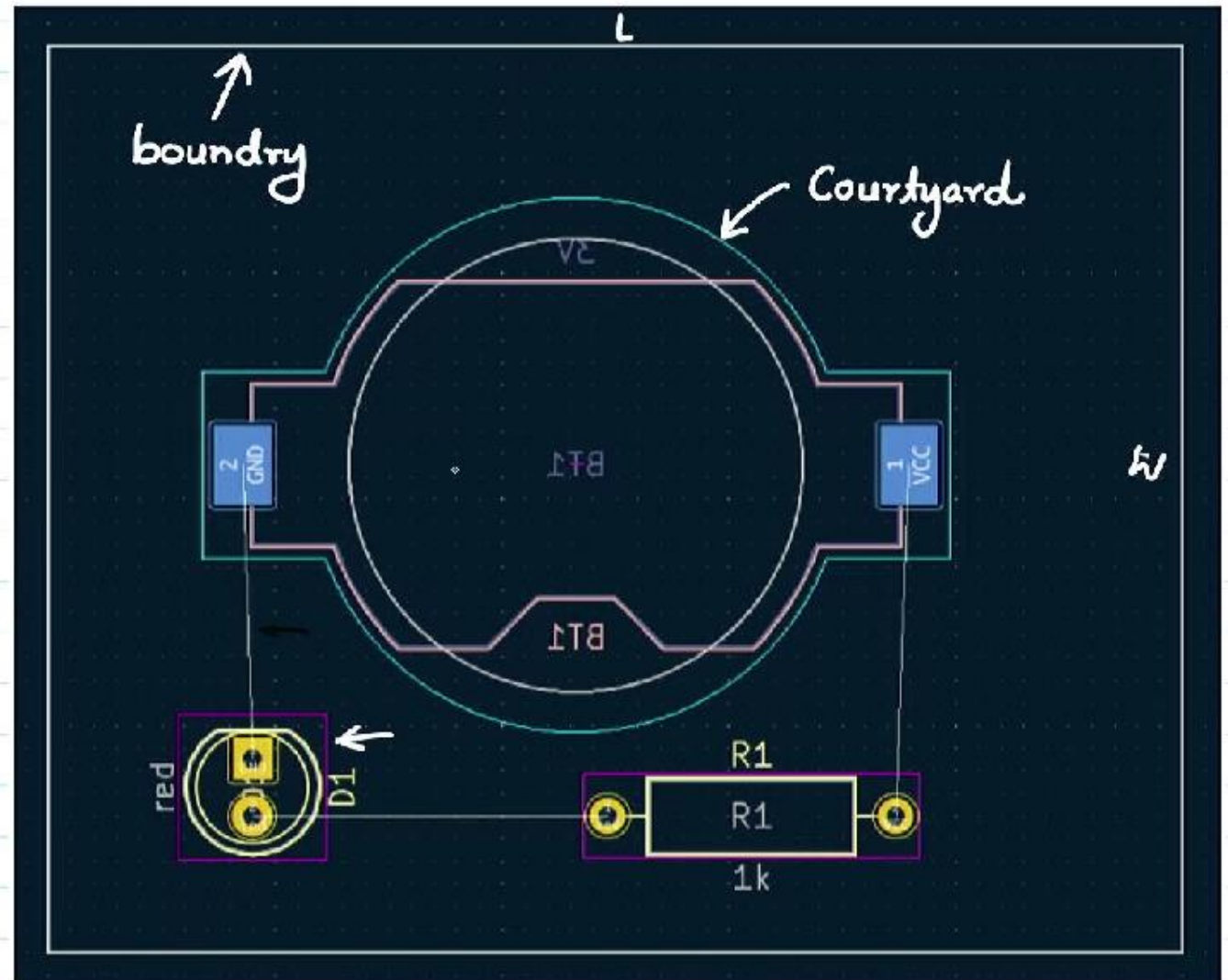


many types of shapes can be designed.



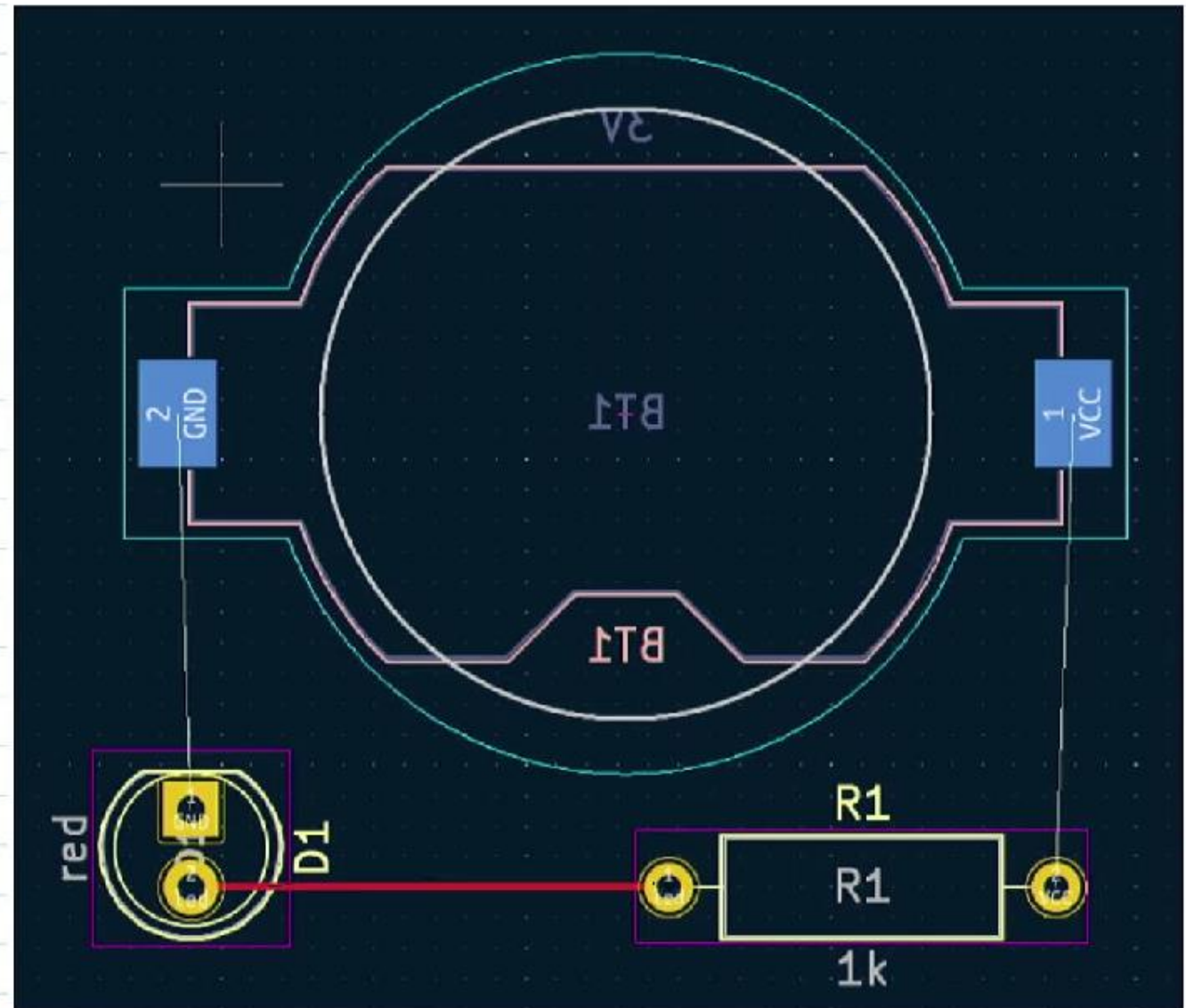
Place Footprints:

- Almost all components have a "Courtyard".
- These courtyards should not intersect.



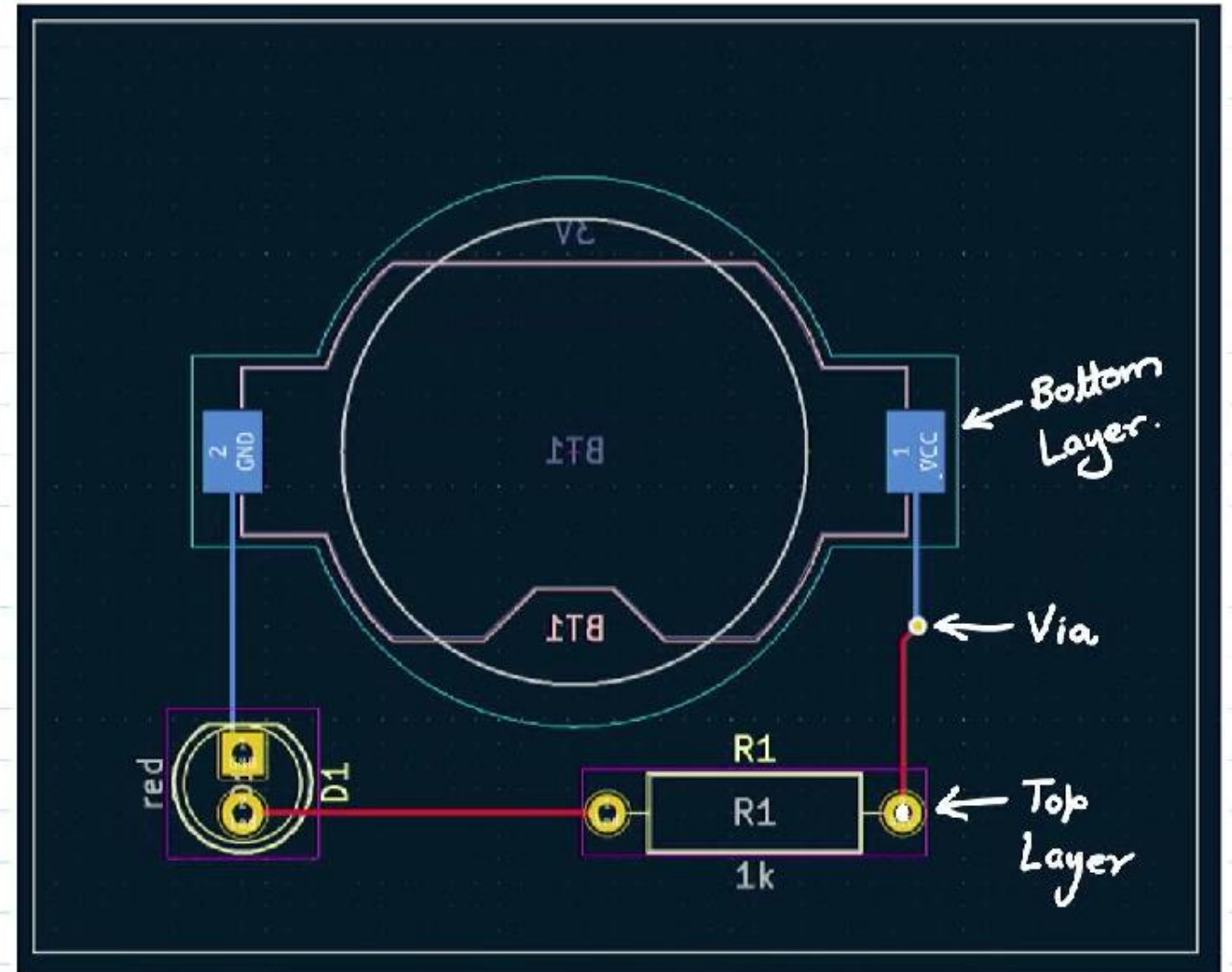
Routing Tracks :

- With the components in place ,
we connect the pads with Copper trace.
- Change the active layer to F.Cu
in the layer tab.



Routing tracks :

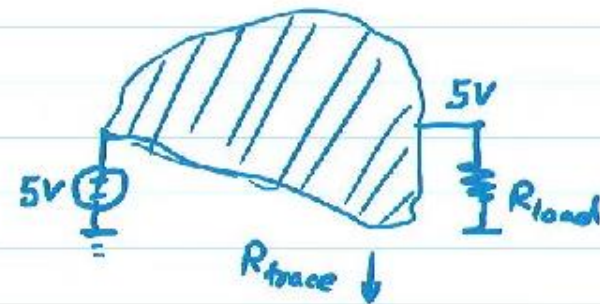
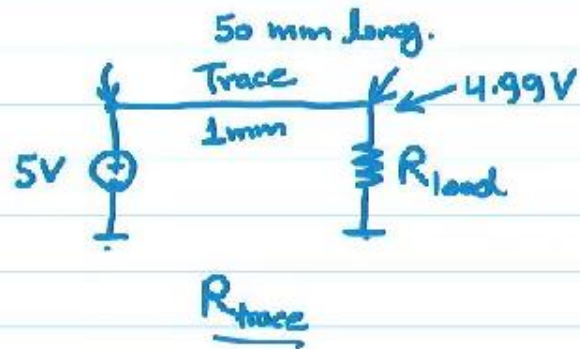
Notice that the active layer automatically changed from F.Cu to B.Cu.



Placing Copper Zones :

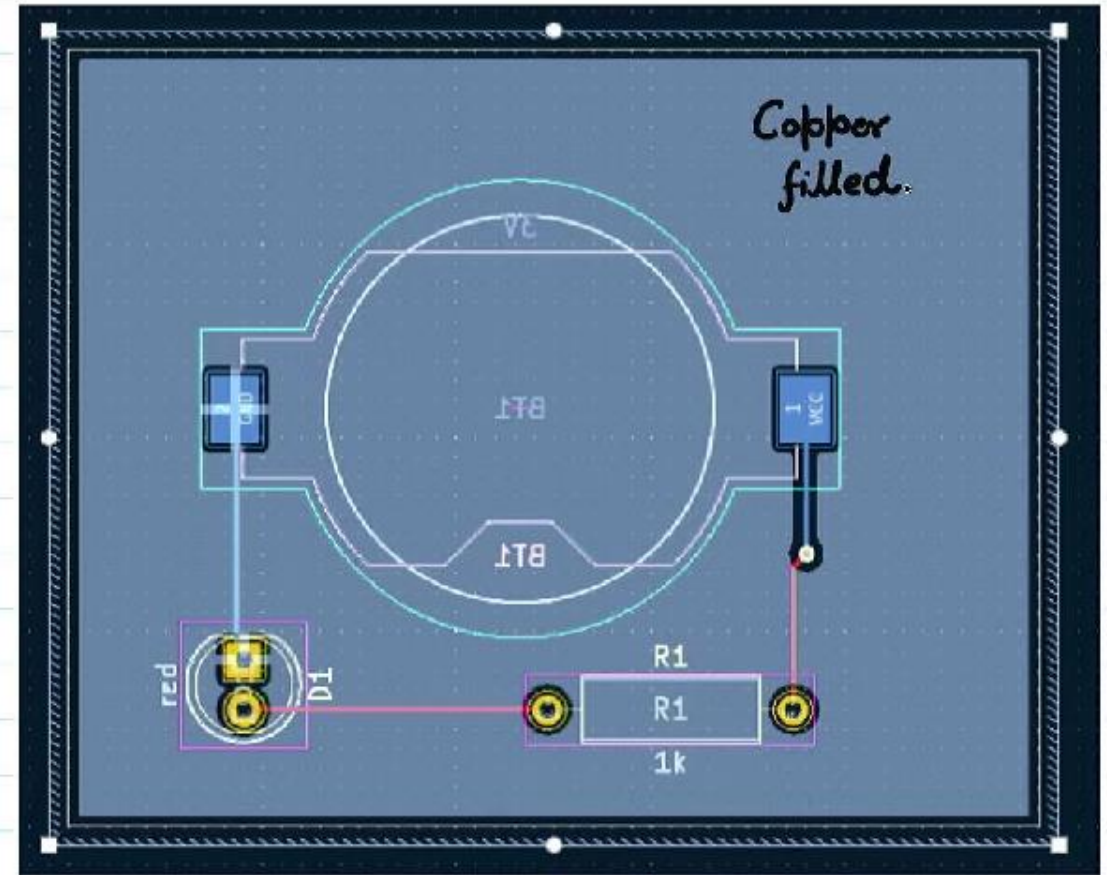
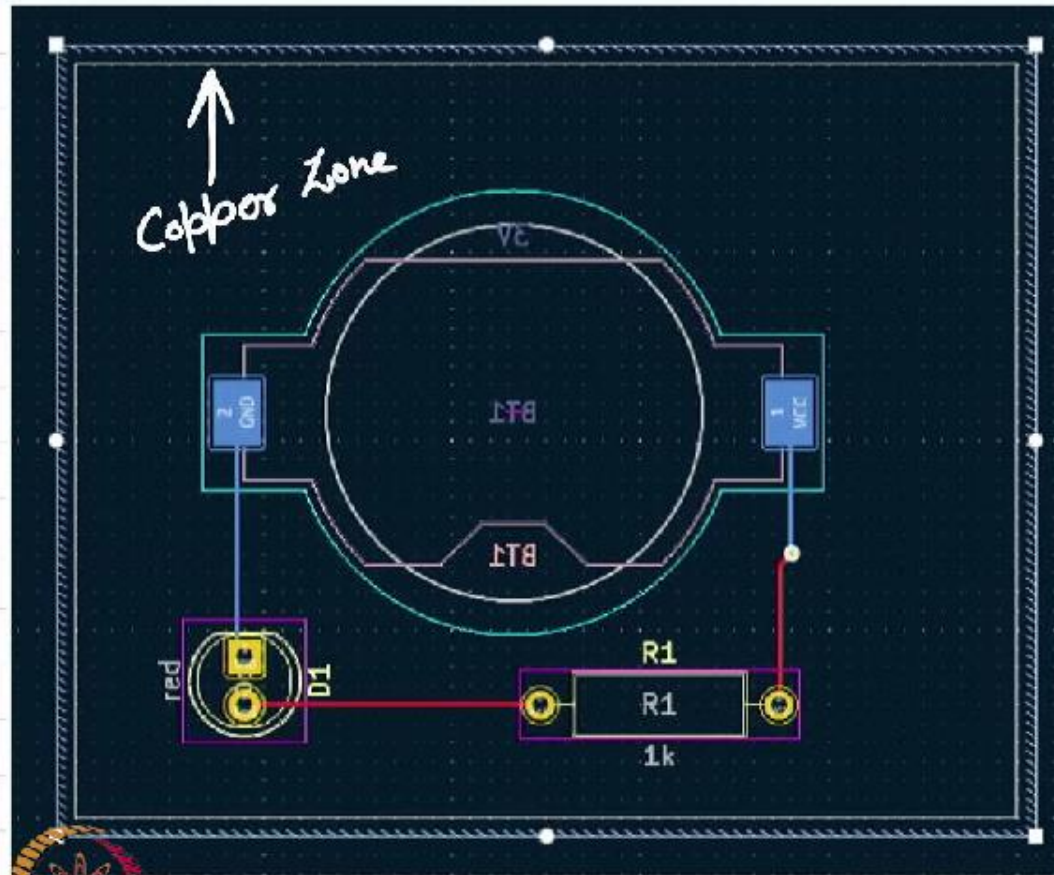
Copper zones are often used for ground and power connections because they provide lower impedance path compared to traces.

"Add a filled zone" Button



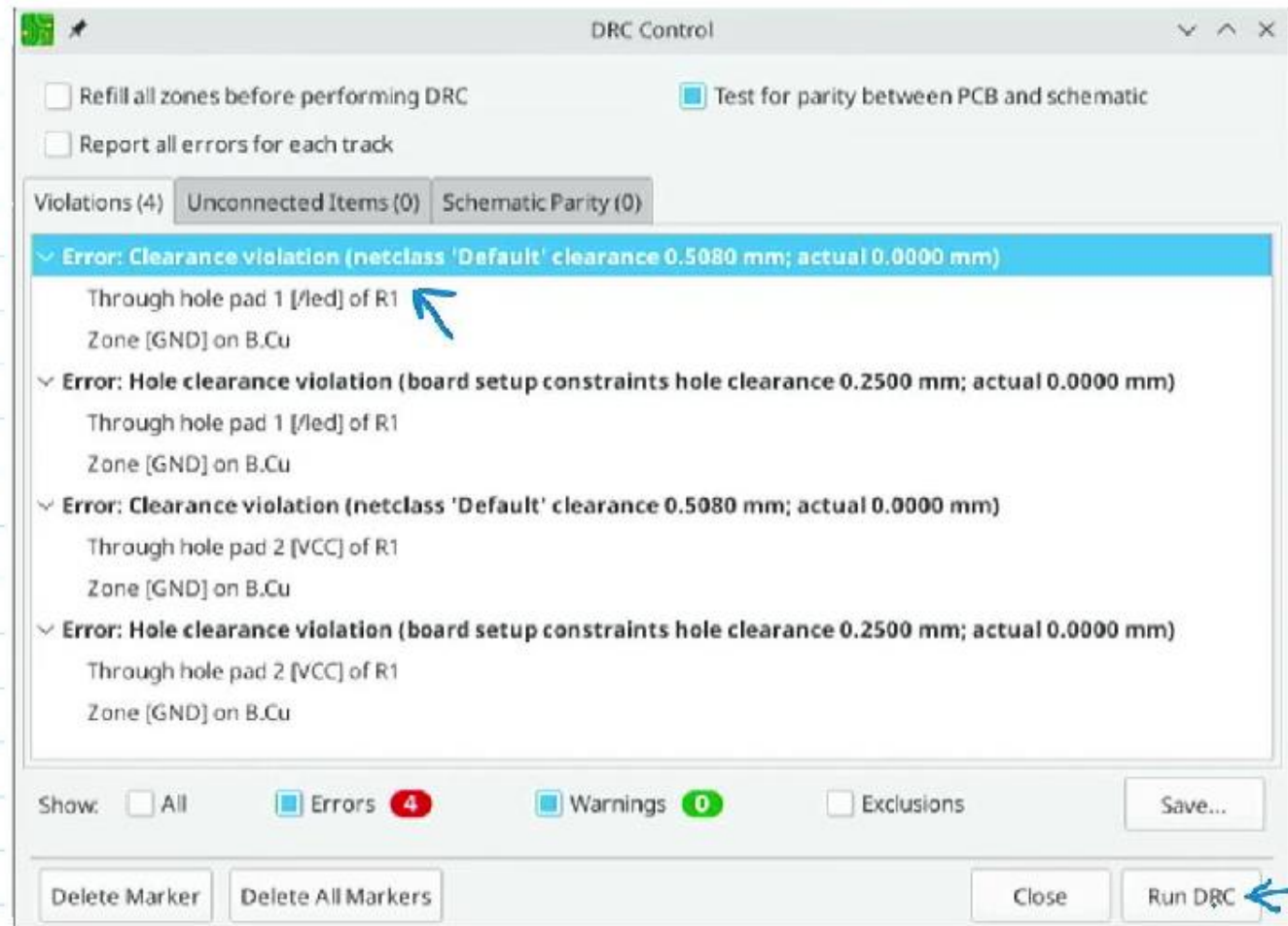
Placing the Copper Zone :

Edik → Fill all zones.



Design Rule Check (DRC):

Inspect → Design Rule Checker



Fabrication Output :

Generate drill file .

