Kicad Copper Pour / Filled Zone

Copper Pour or also called as filled zones are the area of copper that are not etched away after copper etching process. Normally, it is electrically connected to 'Ground Plane' and sometimes it may also be a copper island where it is not tied to any electrical signal.

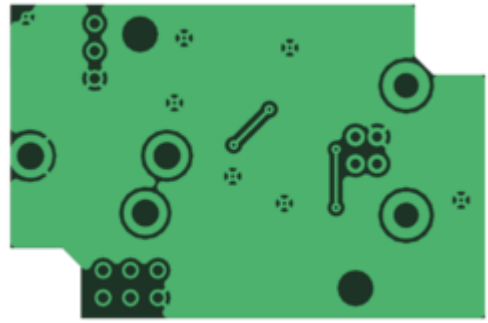


Figure 1 Copper Pour Around Track

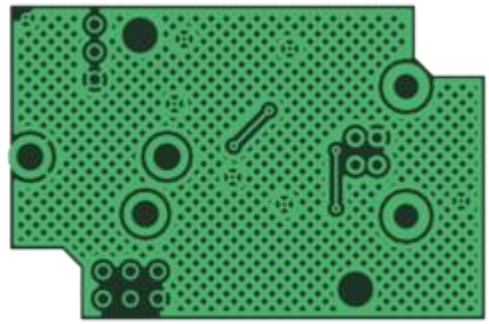


Figure 2Copper Pour with Hatched Style

A proper use of copper pour will;

1. Reduce electrical noise from surrounding
2. Reduce signal crosstalk for high frequency signal
3. Acts as thermal heatsink
4. Prevent PCB board from warping during manufacturing by providing mechanical support

How to Add Copper Pour

For this example, a board from Radar Doppler module is used.

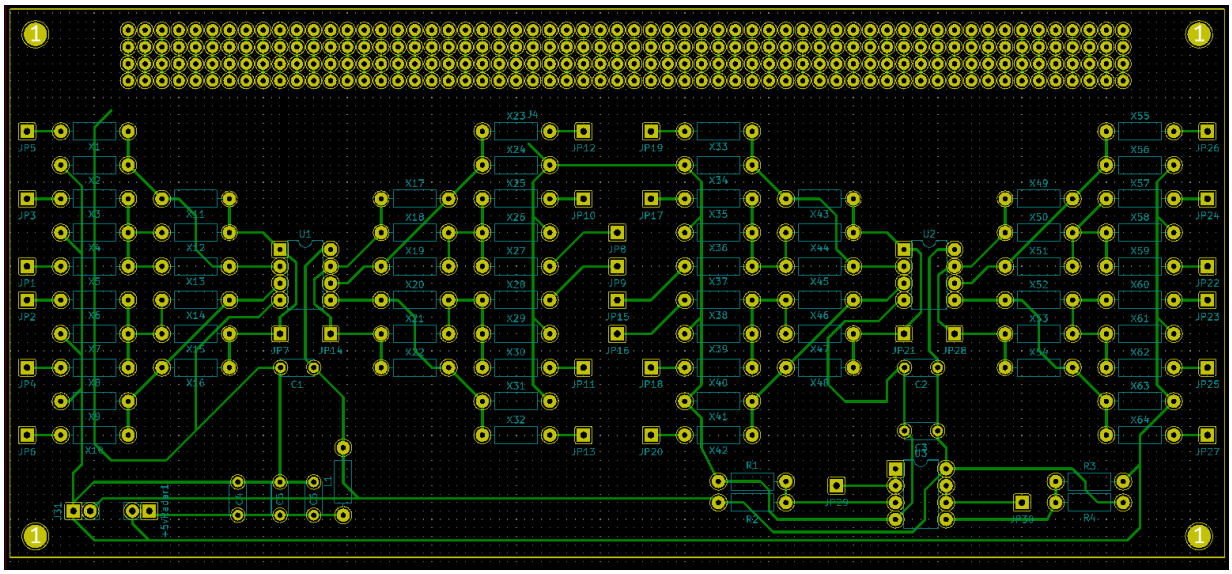


Figure 3 Radar Doppler Template Without Copper Pour

In Pcbnew, the icon for the Copper Pour function is located at the right hand side.

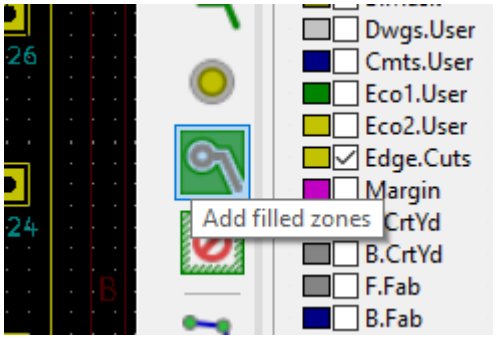


Figure 4 Copper Filled Icon in Pcbnew

Once the icon is clicked, a crosshair will appear and click at where you wanted the copper pour to be generated. A new windows will pop out in order for your to define the settings.

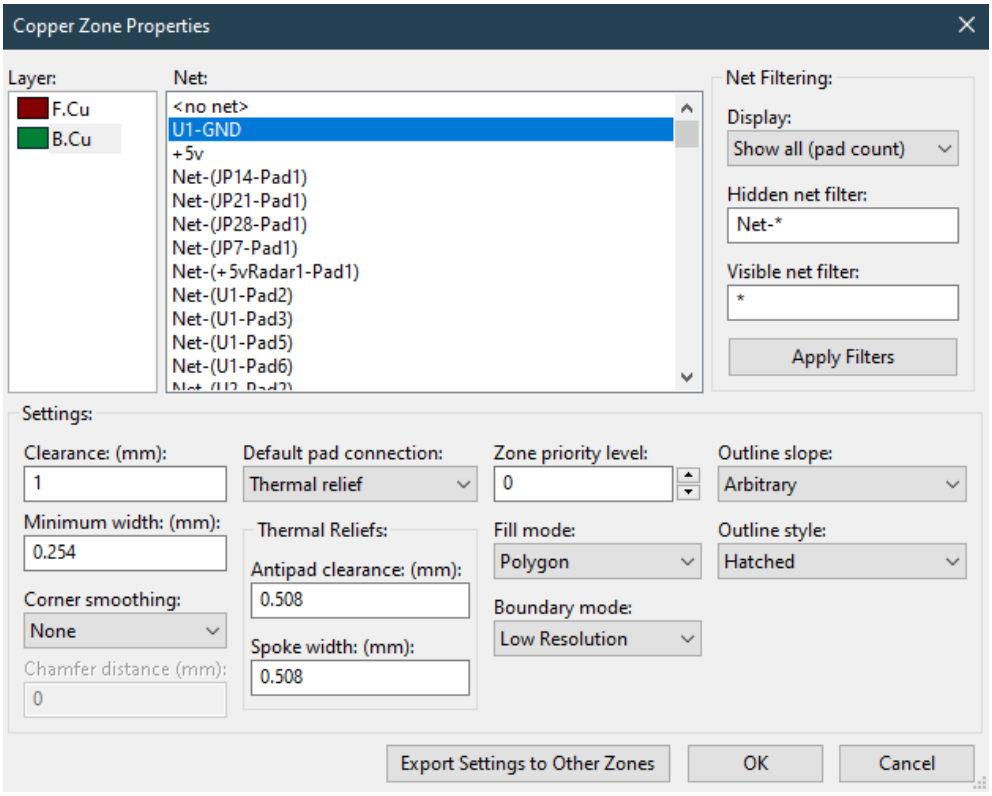
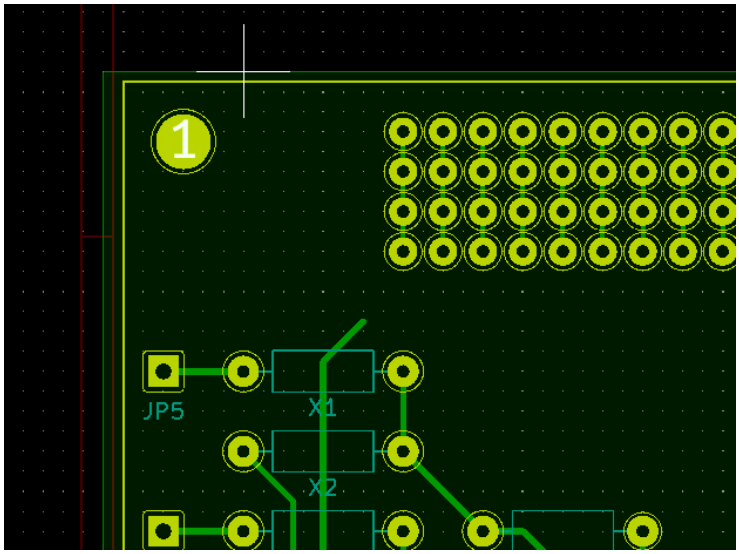


Figure 5 Filled Zone Settings

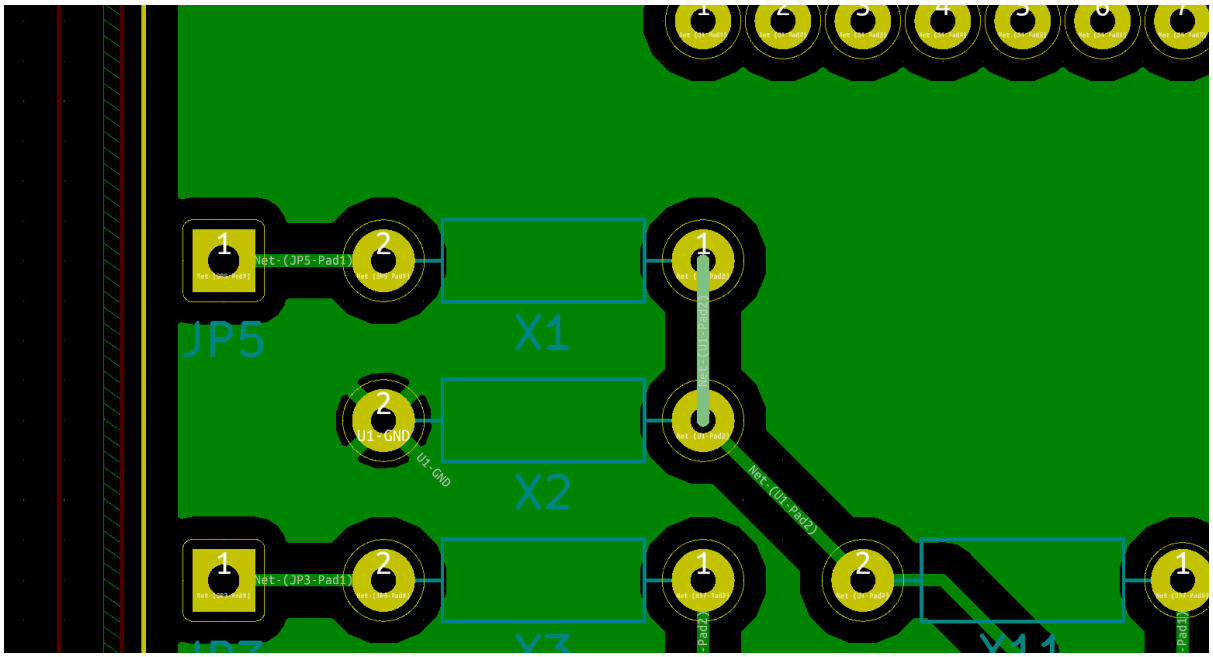
Choose where the layer of the copper will be generated, and choose the net. For this example, the copper pour will be tied to the ground plane U1-GND. You can leave the rest of the settings as it is as this will explain further later on. Click OK.



Start drawing where you want the copper pour to be generated. Once finished, double click on at the start zone.

The outline for the copper pour should be generated together with the copper pour. If not, Press ‘B’ to build it. Make sure the ‘ Show Filled Ares in Zones” is activated. The icon is located at the left hand side on the menu.

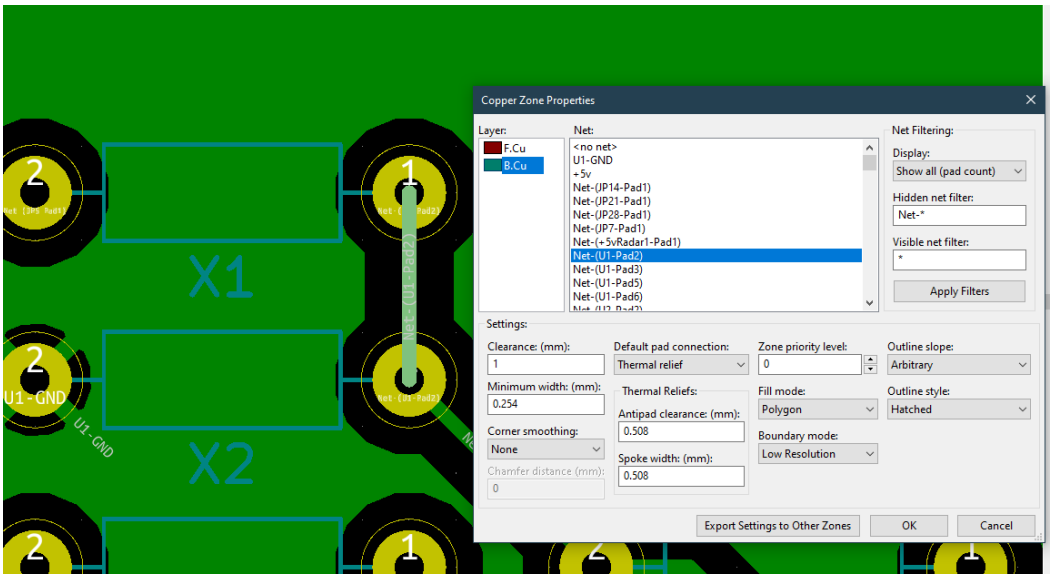
The copper pour should be generated as below. Notice that the copper is not connected to other signal path as it is not connected to it based on the net.



Copper Pour Settings

Zone Priority Level

If you have multiple zoned stacks together, the Zone Priority settings will help you in doing so. Image below shows that the copper pour is currently in Zone Priority Level 0.



Now, a new copper pour will be generated inside the previous copper pour area. The steps are the same as previous except only 2 settings are changed ;

1. Net – a new net is chose based on the track. **(Notice on the image above, the highlighted track , U1-PAD2)**
2. Zone Priority Level – Changed to Level 1

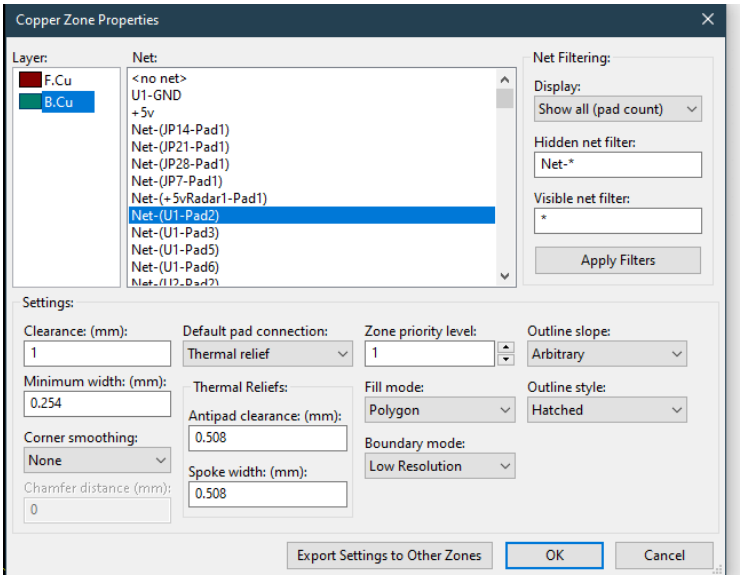
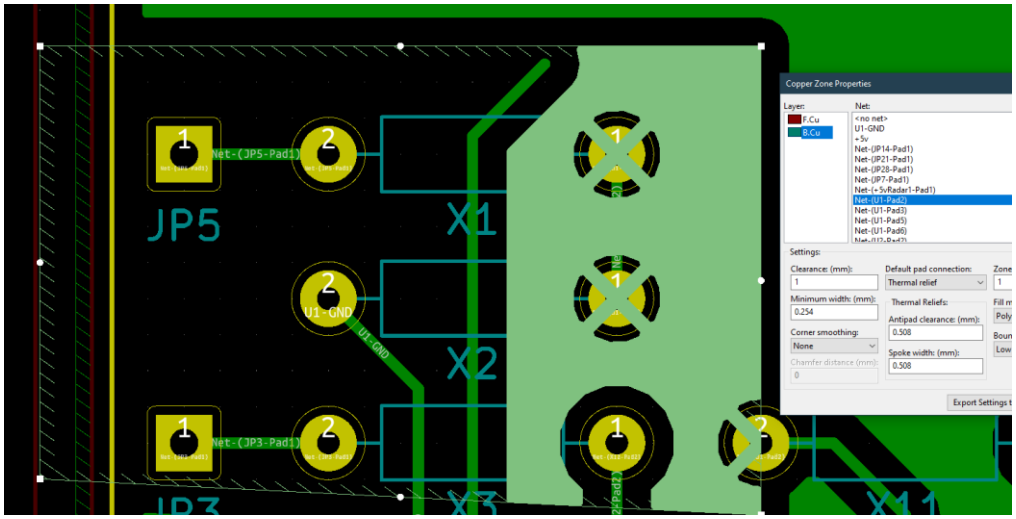


Figure 6 Zone Priority Level 1



A new copper zone is generated inside the previous zone. Notice how it is connected to the net chose before.

Thermal Spoke Width

When using copper zone, a thermal spoke will be generated on the copper pad of the component. This features is to avoid heatsinking effect during soldering in order to avoid cold joint that could leads to poor connection. (Imagine soldering a very thick wire, the copper will dissipate heat and reduce in temperature at the area and thus lead to cold soldering)

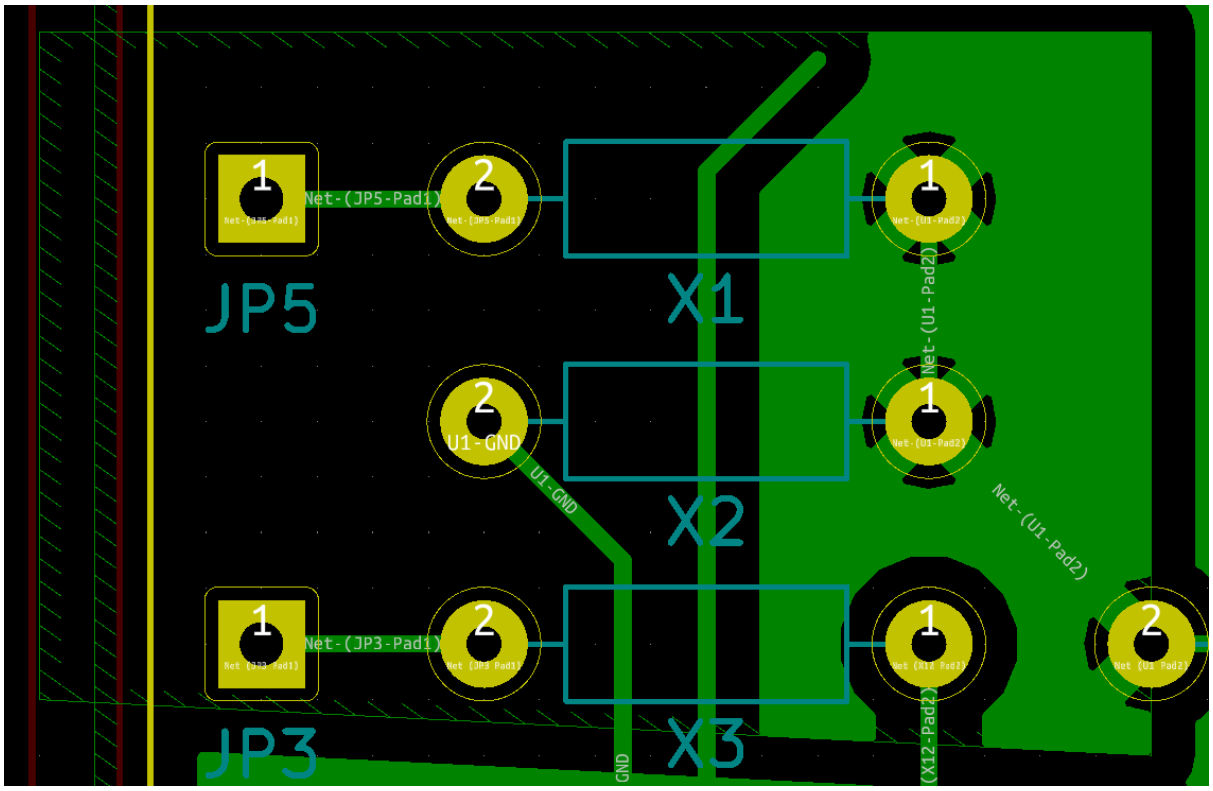


Figure 7 Thermal Spoke Width with 0.5mm thickness

Clearance

This features will help to define a new clearance between nets and copper pour zone.

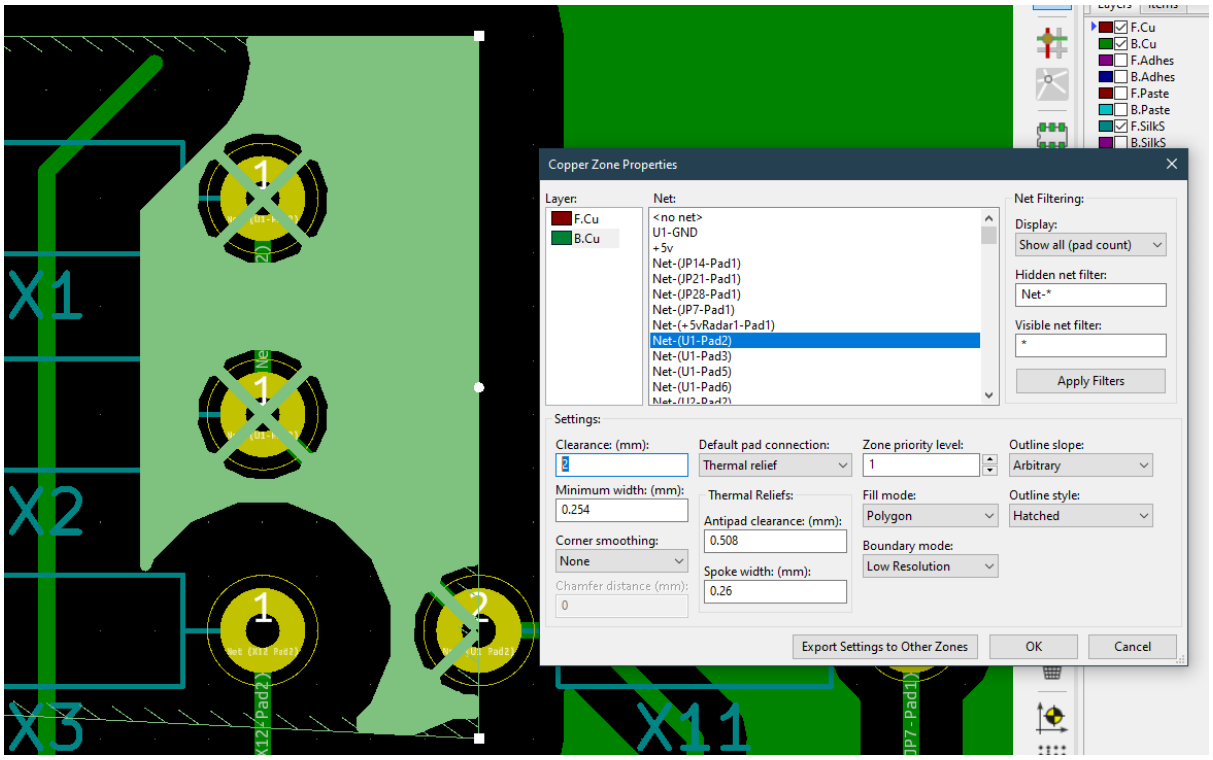


Figure 8 Copper Pour With 1mm Clearance