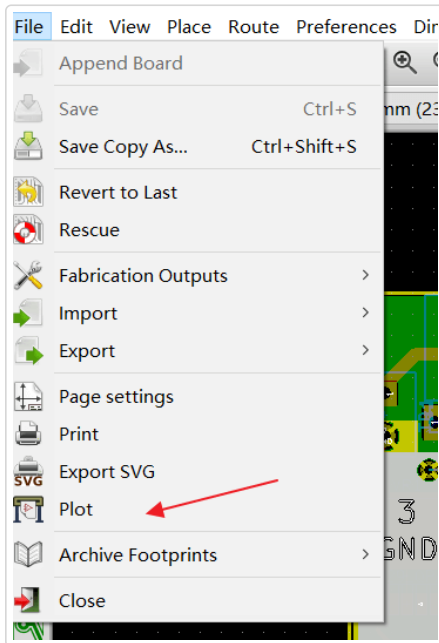


How to export Kicad PCB to gerber files

Generate Drill and Gerber Files

Select **File** -> **Plot** from the menu to open the gerber generation tool.

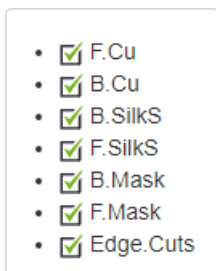


In general, there are 8x layers you need to have a PCB fabricated:

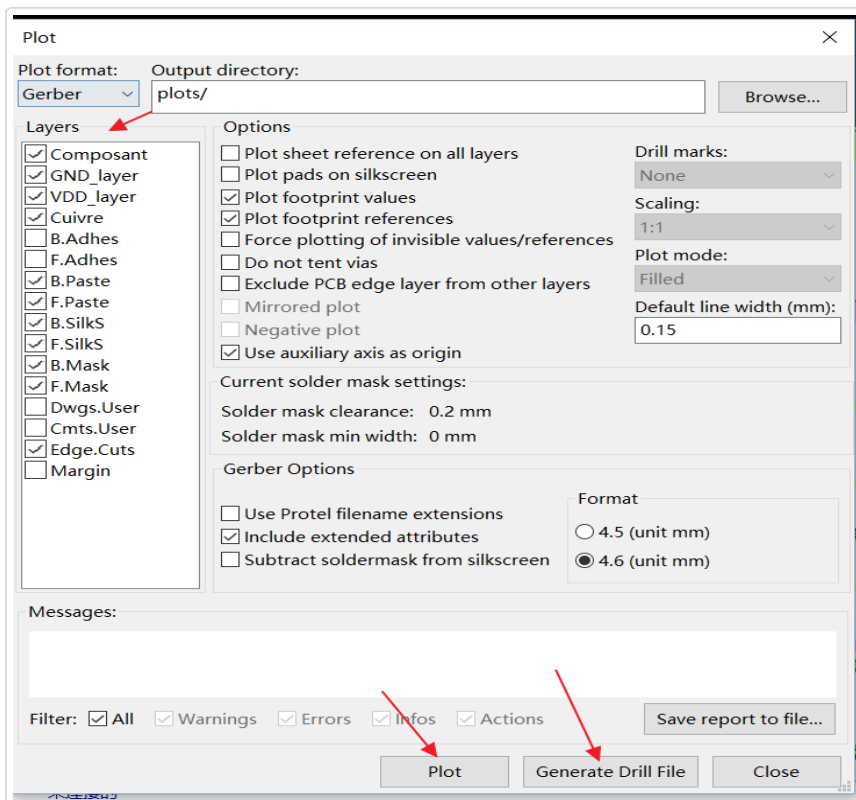
- Top Copper (F.Cu)+ Soldermask (F.Mask) + Silkscreen (F.SilkS)
- Bottom Copper (B.Cu) + Soldermask (B.Mask) + Silkscreen (B.SilkS)
- Board outline (Edge.Cuts)

Drill file

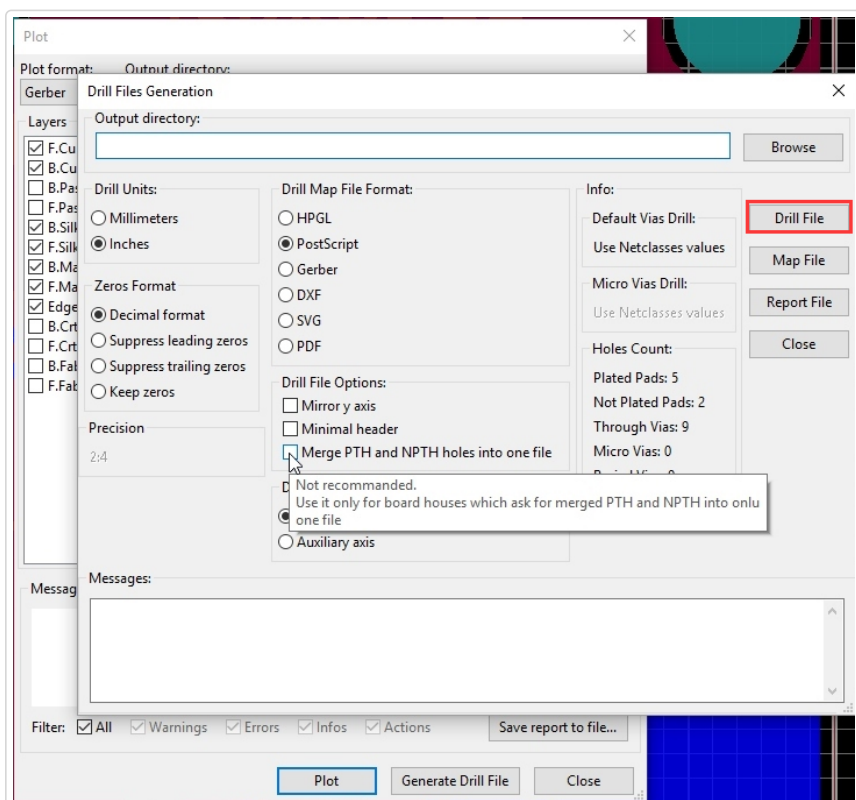
In the Plot window with the Plot format set for Gerber, be sure these Layers are checked:



If you don't know which layers, please check all layers, JLCPCB will help you to use the right layers. Click '**Plot**' to generate the gerber files for the layers.

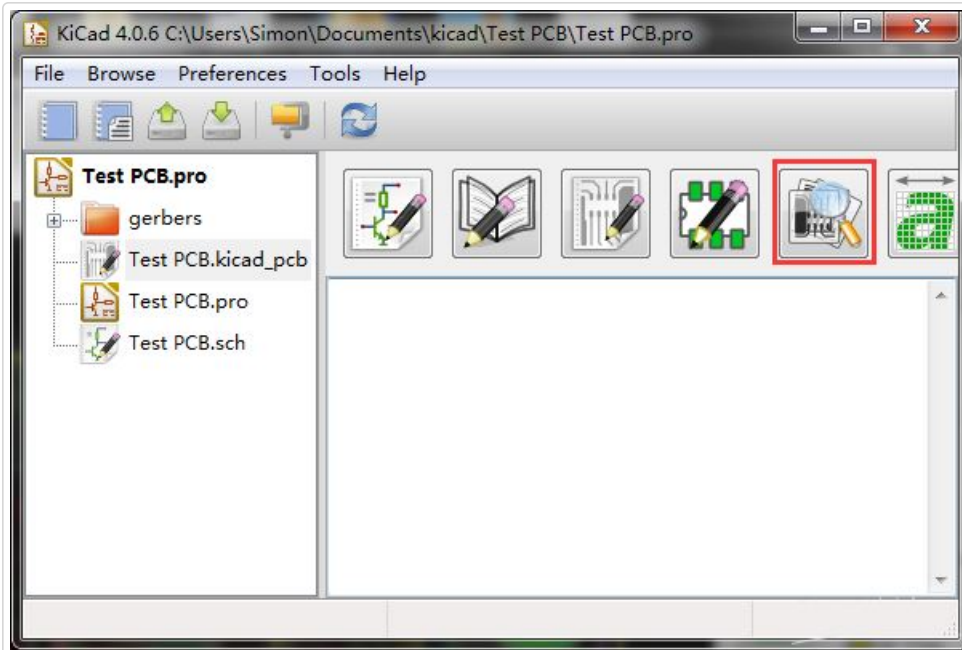


Don't forget to Generate the Drill Files. Click on '**Generate Drill File**' button. You can use the defaults here as well. More on the PTH vs. PTH check box in a minute. For now just click '**Drill File**' or press '**enter**' to generate the drill file. Use the same output folder as for the gerbers, which should be the default.



Check the Gerber files in GerbView

Next, click '**Close**' to exit the Drill and Plot windows. All of the files should have appeared in your gerbers folder. KiCAD comes with a gerber viewer called GerbView, you can open the "GerbView" and check what your board looks like before sending it to manufacturer.



For more details, please see this video tutorial (<https://www.youtube.com/watch?v=4PnY2IUQ2Tg>) of generating Gerber files for manufacturing in KiCad.

If everything looks OK, select all of the files, zip them up, and upload the zip file to JLCPCB order page (<https://jlcpcb.com/order/pcb>).

✉ *Still need help? Contact Us (/contact)*

Last updated on November 22, 2017

RELATED ARTICLES

📄 [How do I place an order? \(/article/21-how-do-i-place-an-order\)](/article/21-how-do-i-place-an-order)