

Blog > Generate Gerber file from Kicad 5.1.6

Generate Gerber file from Kicad 5.1.6

by: [PCBWay](#) Oct 06,2020 4431 Views 1 Comments Posted in [Help Center](#)

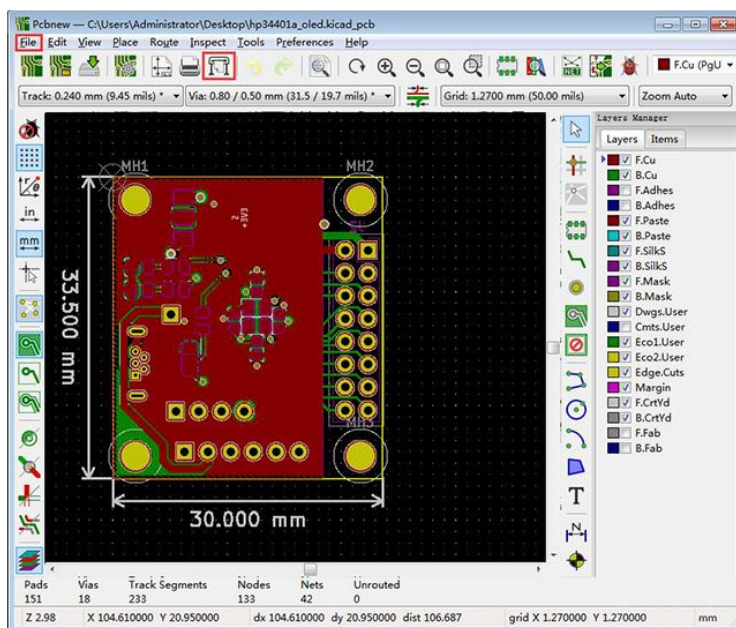
[Tutorial](#) [PCB design](#) [Gerber](#) [Kicad](#)

1. Open your .kicad_pcb file

After opening your Kicad project - .pro file, you can double click the **.kicad_pcb** file or click the **"PCBNew"** button to open your PCB editor.

2. Plot your Kicad PCB as Gerber files

Click the **"File"** menu --> **"Plot"** and choose the necessary layers shown as below (for 2 layer boards), then click the **"Plot"** button to generate the related layers.



The necessary layers for 2-layer PCB could be:

Top Layer: pcbname.GTL

Bottom Layer: pcbname.GBL

Solder Mask Top: pcbname.GTS

Solder Mask Bottom: pcbname.GBS

Silk Top: pcbname.GTO

Silk Bottom: pcbname.GBO

Drill Drawing: pcbname.TXT

Board Outline: pcbname.GML/GKO

Submit for Publication

HOT TAG

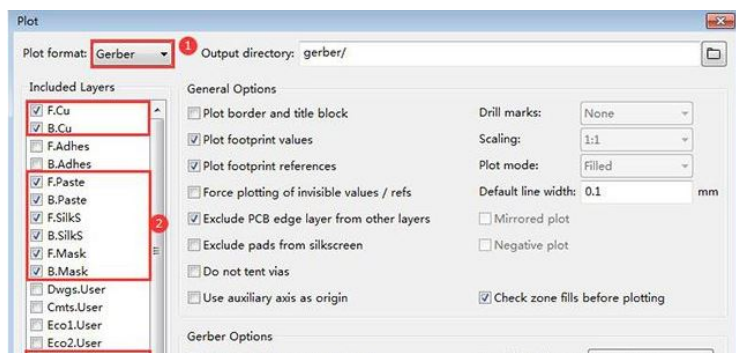
[Printed Circuit Board](#) [PCB](#)
[PCB Board](#) [PCB design](#)
[PCB layout](#) [Circuit Board](#)
[Printed Circuit Boards](#)
[PCB Design Tutorial](#) [PCB industry](#)
[Circuit](#) [Kicad](#) [circuit boards](#)

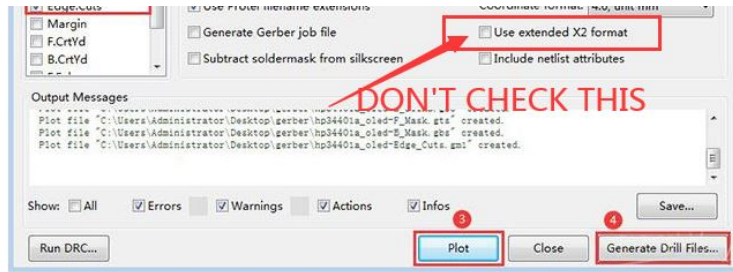
Categories

[Help Center](#)

IF YOU MISSED IT

- [1 PCBWay adjustment on EU VAT](#)
- [2 Basic instructions for using KiCad](#)
- [3 Generate Position File in Kicad](#)
- [4 3 important tips to PCB Typesetting Engineering](#)
- [5 How to generate Gerber from Sprint Layout 6.0](#)
- [6 How to Read & Understand a Circuit Diagram](#)
- [7 4 important tips for PCBWay 24-Hour Express Service](#)
- [8 Generate Gerber file from Kicad 5.1.6](#)
- [9 Necessary file for PCB Layout](#)

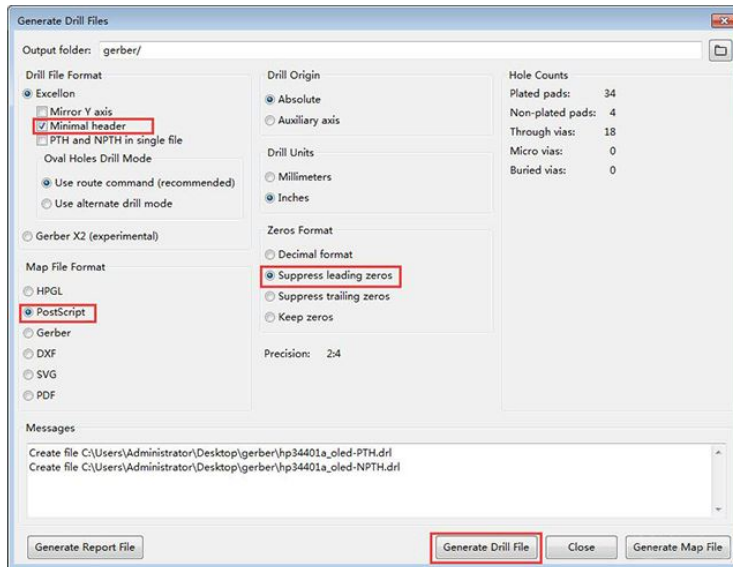




Note: In order to facilitate our access to your files, please do NOT check the "Include extended attributes" before Plot.

3. Generate the drill file

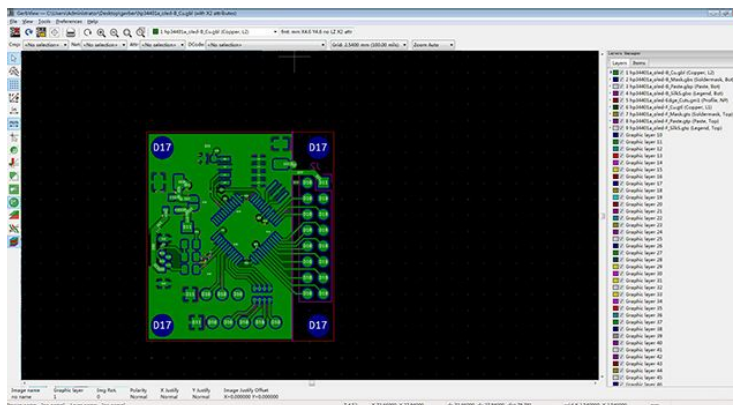
Before closing the plot window, you need also generate the drill for manufacturing. Select **"Suppress leading zeros"** and **"Minimal header"** and click **"Drill File"** button as following shown.



4. Check the Gerber files in GerbView

Now you have finished the job. But you should always check whether your Gerber files are working or not. Open the **"GerbView"** and check what your board looks like before sending it to manufacturer.

Now you can see your board like this.



5. Compress all the files in a single .zip file

The final step is to Compress all the files in a single .zip file, then you can fill out the form about your PCB parameters (size, quantity , layers , thickness , etc) on our **"PCB Instant quote"** page and upload your .zip (Gerber) file to PCBWay online system, our engineers will check it again and feedback to you if any problems happen before it can be fabricated. Here we go!

See **"Generate Position File in Kicad"**

https://www.pcbway.com/blog/help_center/Generate_Position_File_in_Kicad.html

Leave a Comment (1)

Share to:

Previous: 32-channel EEG ADC with Cortex-M7 ----FreeE...

Next: Running Rust and FreeRTOS on the PADI IoT Stamp

Related articles

Designing A Flexible PCB

Делаем светодиодные часы SxemWatch-v1 на ATmega3...

3 Tips for RF PCB Layout

Generate Gerber file from Kicad 5.1.6

Comments (1)

[Log in to post comments.](#)

Upload a photo: **Browse** You can only upload 1 files in total. Each file cannot exceed 2MB. Supports JPG, JPEG, GIF, PNG, BMP

Post Comment



sayanee

Reply

Thanks! Useful to know the KiCad options for generating the Gerbers.

May 31, 2021 0 Comments



THE PCBWAY THAT RECOGNIZE THE TALENT AND EFFORT OF THE BEST ELECTRONIC DESIGNERS IN THE WORLD.