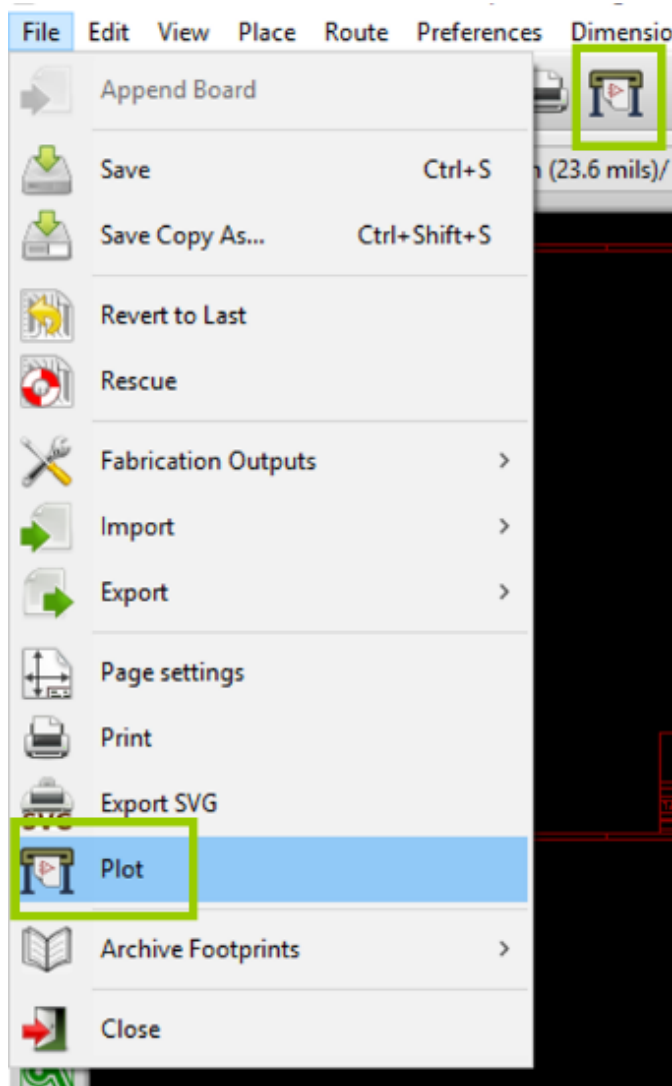




How to generate Gerber and drill files from KiCad

← How to generate PCB Gerber files?

In KiCad, open your PCB design file and go to *File -> Plot*, or find the plot icon on the main interface.



This will bring up the plot window. Select an output directory for your Gerber Files and select the following layers and options.

New and returning users may [sign in](#)

How to generate PCB Gerber files?

What is Gerber file?

How to generate the Gerber (manufacturing) files?

How to generate Gerber files in Altium Designer

How to generate Gerber files from CircuitMaker

How to generate Gerber and Drill files from Eagle

How to generate Gerber and drill files from KiCad

How to generate Gerber files from DipTrace

How to generate Gerber files from Proteus

How to generate Gerber files from DesignSpark

How to convert drilling layer from Gerber to Excellon format in CAM350

How to generate Gerber files from PCB

Contact Support

Give feedback

Knowledge Base

Assurance 2

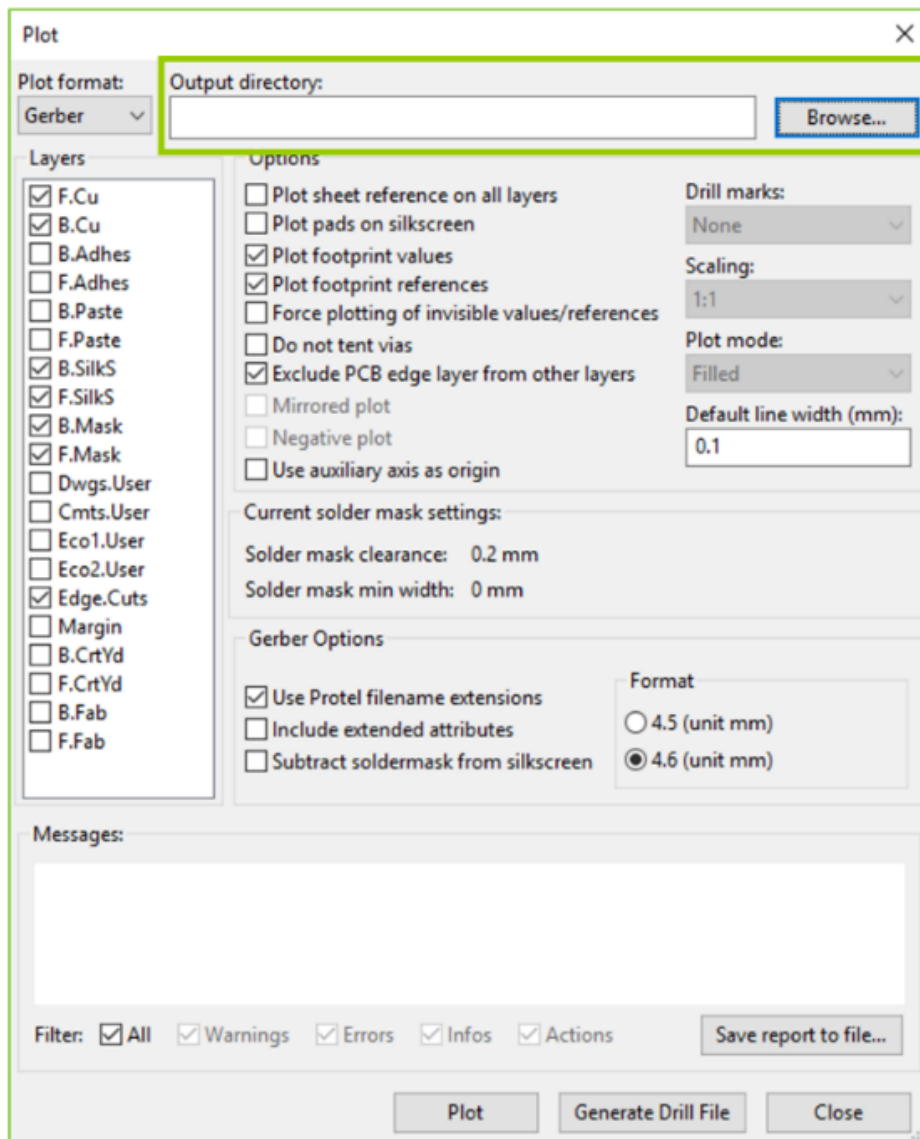
General 2

Fusion Specification 9

Fusion Service FAQ 14

Fusion Technical FAQ 20

Have a problem with PCB design? 7



Please pay attention to the required layers, there should be 7 layers in total for typical two layer boards. If your boards are single or multi-layered, please select the necessary layers. Please note that all mechanical elements such as cut-outs and v-cuts should be drawn in the Edge.Cuts layer.

If you need the paste layers for stencil orders, please check *B.Paste* and *F.Paste* as required.

Also, please ensure that you **do not** select the *Include extended attributes* option under *Gerber Options*. This will effectively convert the format of your Gerber files to X2, which not all our equipment supports.

How to generate PCB Gerber files? 11

Have a problem with Fusion orders? 12

[Have a shipment problem with Fusion orders?](#) 8

Bazaar Ordering 8

Products' FAQ 90

Shipping 5

Tech Support 4

よくある質問（Fusion向け） 33

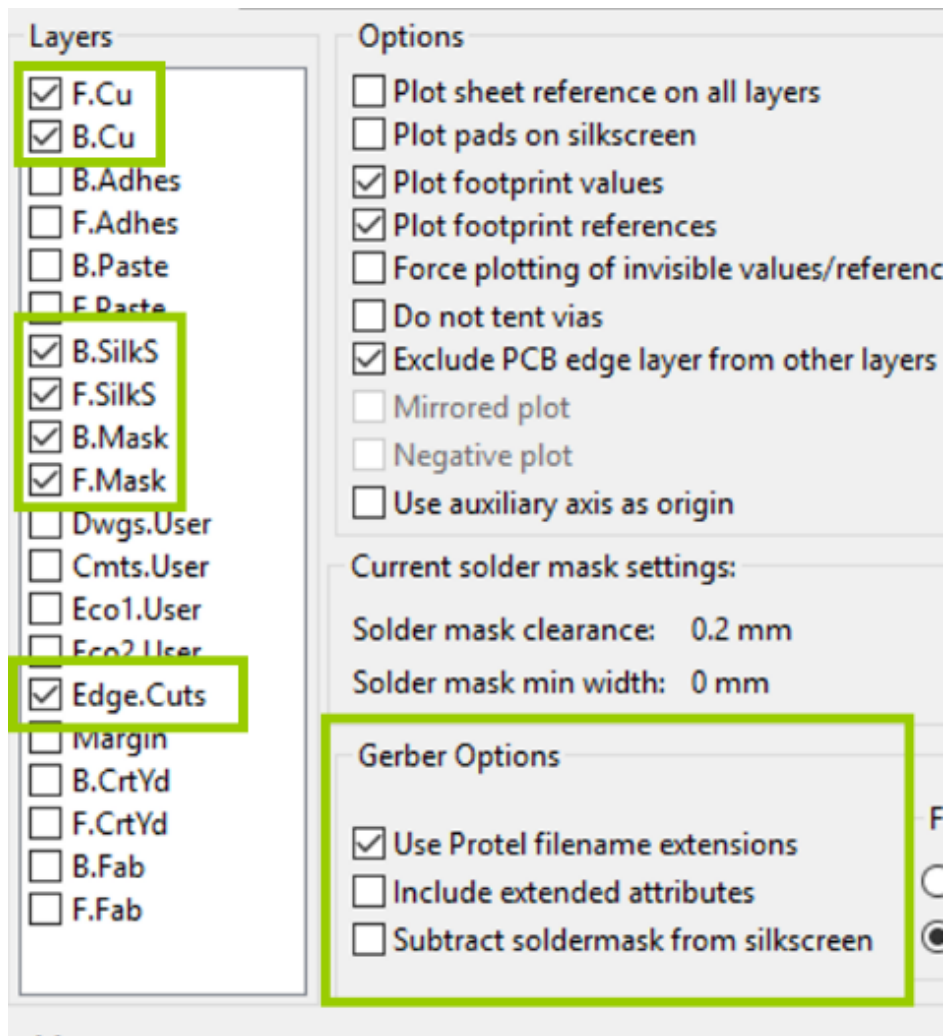
常见问题 2

技術ガイド（Fusion向け） 27

標準規格書（Fusion向け） 8

All articles

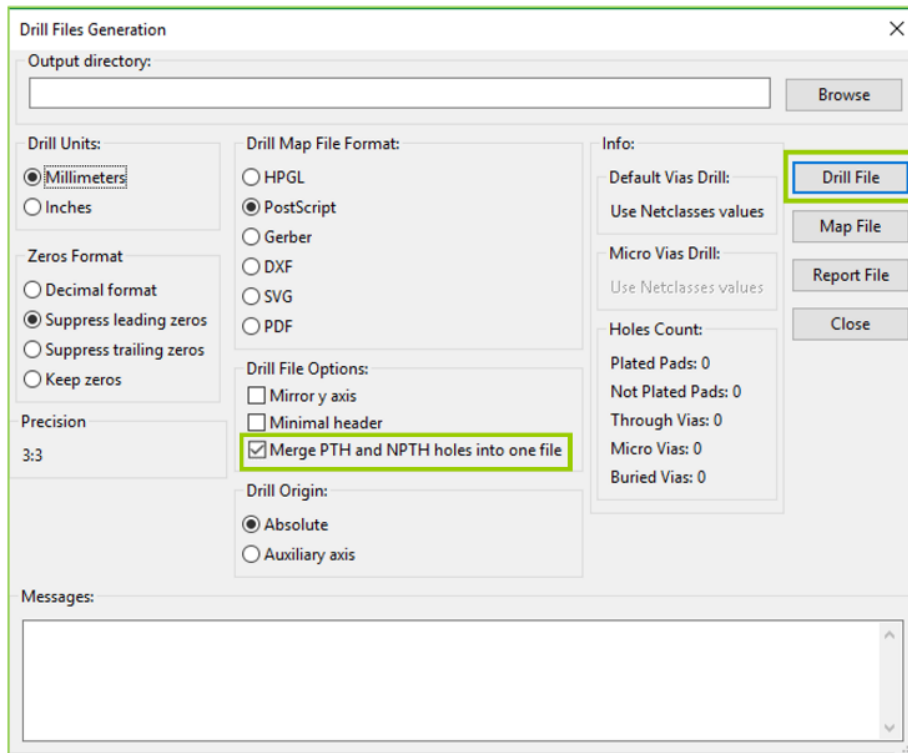
seedstudio



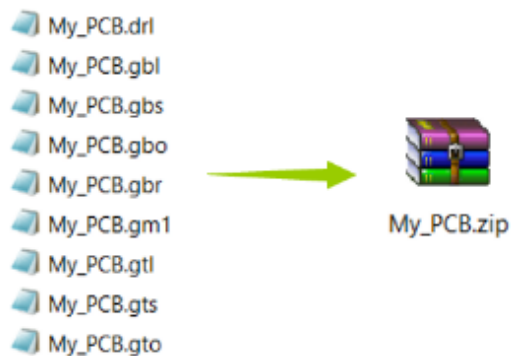
Click *Plot* and confirm that all the Gerber files were generated successfully.

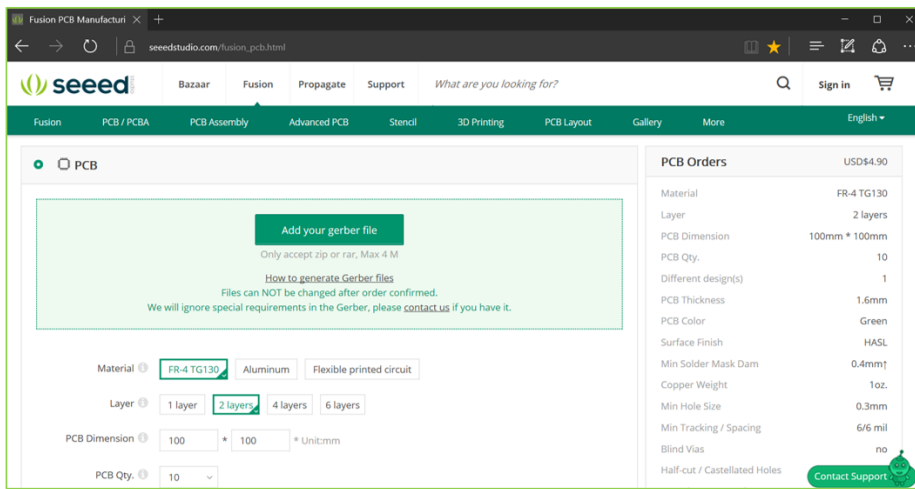
To generate the drill files, select the *Generate Drill File* button and this will open a new window. Click *Drill File* on the right and this will export the drill file in Excellon format in your chosen directory.

We suggest you select the option to *Merge PTH and NPTH holes into one file*, otherwise KiCad will separate plated and non-plated holes into two drill files, and if one is missing, there will be holes missing on your boards.



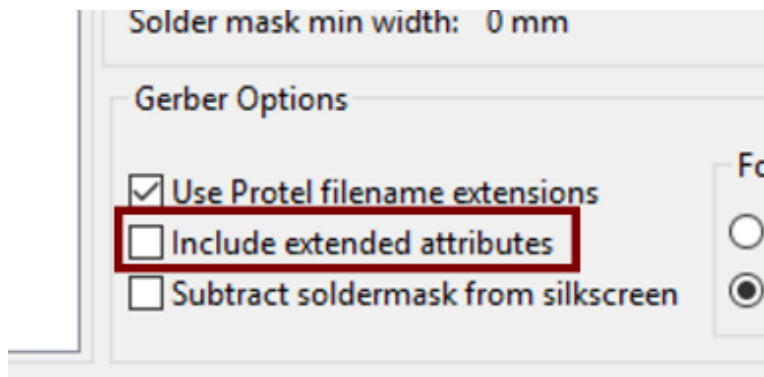
Please check that you have all 8 (or 9) files then zip them into a .zip or .rar archive file. You can use this to place the order on the [Seeed Fusion PCB order page](#). We recommend that you give the files a quick check using the online Gerber Viewer before confirming the order.



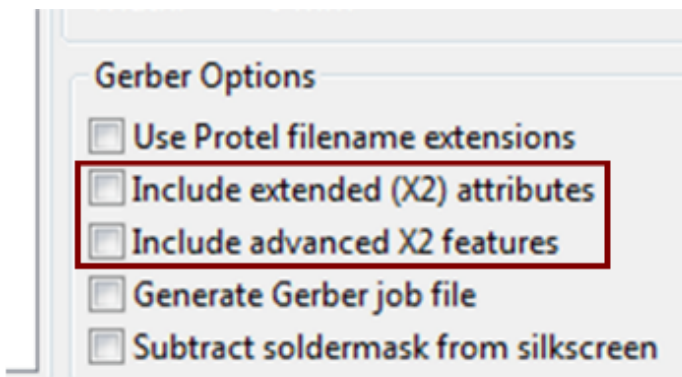


Common Mistakes and Problems:

- **Incorrect Format:** We prefer the Gerber files to be in RS-274x format over X2 since all our equipment are compatible with RS-274x. However, in standard versions of KiCad, it is not obvious whether the files are being exported in X2 or RS-274x format. In the *Plot* window, by selecting *Include extended attributes*, this converts the files into X2 format and cannot be imported into our standard CAM software. Please ensure you **do not** select this option.



In some versions of KiCad, this is mentioned but please ensure that **all** X2 options are unchecked.



- **Missing mechanical holes:** Since KiCad has the option to export the plated and non-plated drill holes separately, it is easy to forget to include them both in the archive. If the PTH drill file is missing, this is easy to catch. But if the NPTH drill file for the mechanical holes is missing, it can be easy to overlook. To avoid this issue we suggest selecting the *Merge PTH and NPTH holes into one file* option.

Last Modified 2017/11/26

seeedstudio · Feedback & Ideas for seeed · Terms of Service & Privacy Policy

Powered By UserVoice · Product Management Platform