



How to generate gerber from KiCAD?

← Fusion technical support

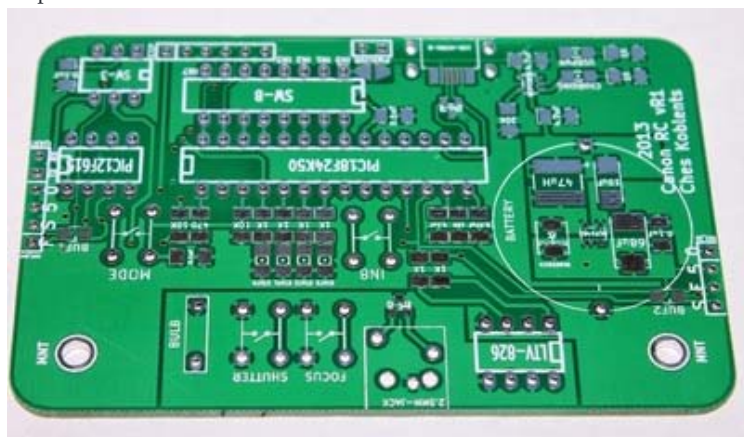
Thanks so much for our Friend Ches, this article was written by him in 2003. The content is very detail even a green hands can generate a Gerber file according to it. Thanks again for Ches's sharing.

Original article: <http://koblenets.com/Ches/Original-Work/46-KiCAD---Seedstudio-Fusion--Getting-Started/>

KICAD + SEEDSTUDIO FUSION: GETTING STARTED

Seedstudio Fusion is a batch PCB service for 2- and 4-layer boards. With shipping, the cheapest option is ten boards, 5x5 cm (2x2 inch) maximum size, for \$15. The turn-around time is just about a month, which includes about a week to build and about three to ship. The quality is high enough, and the DRC rule limitations are acceptable, for hobbyist work.

Their instructions are not entirely clear for KiCAD. Here's how I got started, and received boards precisely as I expected.



Let's get the basics out of the way.

1. Learn KiCAD
2. Create a schematic
3. Create a board layout
4. Pass DRC with Seedstudio Fusion's rules
5. Learn how to generate gerbers and a drill file

At this point, the real challenge is to figure out how to generate the proper gerbers. Why is this an issue? Well, if we look at Seedstudio's requirements (as of writing this article):

Gerber file requirements:

The following layers are needed:

Top Layer: pcbname.GTL

Bottom Layer: pcbname.GBL

New and returning users may **sign in**

Fusion technical support

What is Gerber file?

What are the PCB panelization rules?

How to export the drill files correctly?

Can I use another software beside Eagle to generate the Gerber files?

What is the minimum milling slot?

What is the correct way to draw the silkscreen on the bottom side?

If my PCB does not require Solder Mask, what should I do?

If my PCB do not require Drilling Holes, what should i do ?

Can I panelize the PCB?

Which layer should I use in EAGLE for milling?

What is the minimum dimension of the boards/PCB for milling?

Is PCB panelization only for the same PCBs, or I can Panelize different PCBs?

Do I have to use the Fusion PCB Eagle Design Rule and Eagle CAM file for my Panel?

If I have the v-cut, what is the minimum length between circuit wire and the v-cut line?

What is the depth of the v- cut?

What is the minimum/maximum dimension of the boards/PCB for v-cut?

What is squeegee side/line?

If my PCB's do not have the silkscreen, how can I proceed with my PCBA order?

Exporting manufacturing files from Altium Designer

PCB Production No.

Export PCB gerber from Cam

How to generate gerber from KiCAD?

How to export the correct drilling layer from eagle

What is the Minimum inner trace width for 4 layers?

What is the minimum trace width?

What are the dimensions for Dielectric Separation thickness?

What is the Board Thickness Tolerance?

What are the maximum and

Solder Mask Top: pcbname.GTS

Solder Mask Bottom: pcbname.GBS

Silk Top: pcbname.GTO Powered By UserVoice · Product Management Platform

Silk Bottom: pcbname.GBO

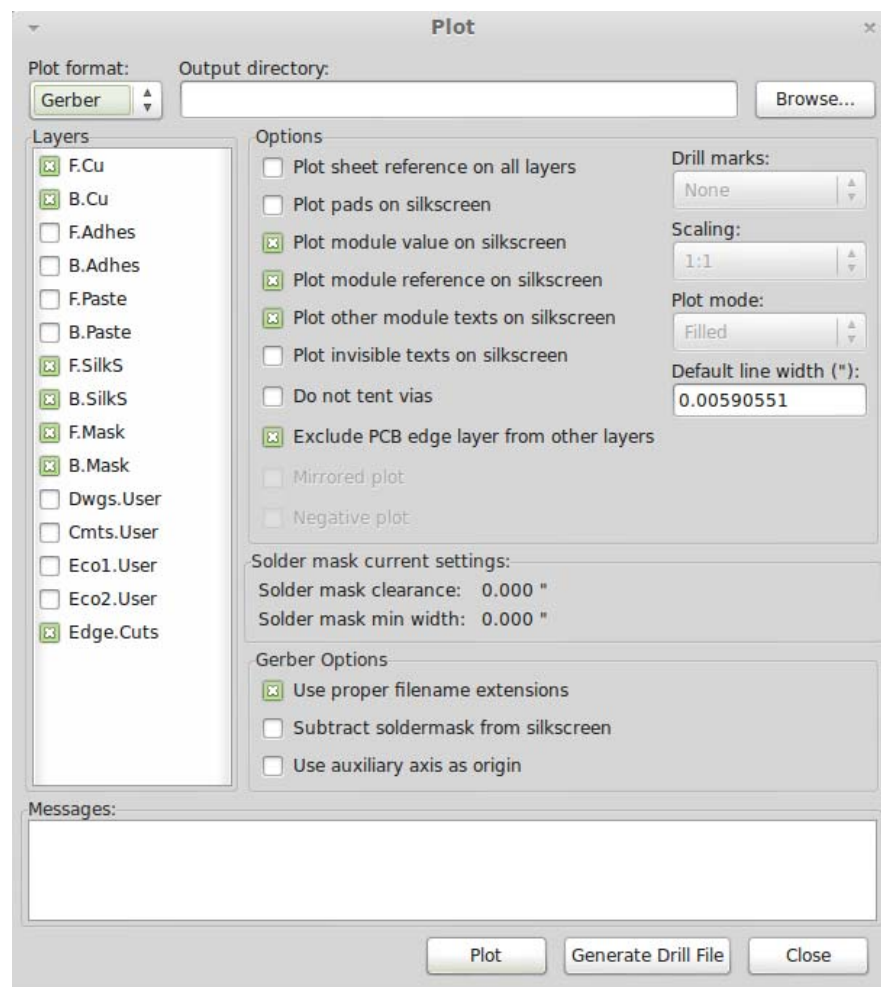
Drill Drawing: pcbname.TXT

Board Outline?pcbname.GML/GKO

Note:

The Gerber file must be RS-274x format.

Now when you open up your **Plot** dialog, you should check the **B. Silks** (back/bottom silkscreen), **F. Silks** (front/top silkscreen), and **Edge.Cuts**(outline) gerber options. Your screen should look something like this:



And in the drill file screen, you should check **Minimal Header** box, and select **Suppress Leading Zeroes** from the 'Zeroes Format' selection, leading to a 2:4 precision. It should look like this:

minimum board dimensions?

What is the minimum spacing for traces, vias, or pads?

What is the minimum silkscreen width?

What are the available Board Thicknesses?

What is the Minimum silkscreen text size ?

What is the minimum distance between pads and silkscreen?

What is the inner layer copper thickness?

What is the range for the drilling hole diameter?

What is the size of the annular ring?

What is the machining diameter tolerance?

What are the specifications for the solder mask?

What is the dimensions for Outer Layer Copper Thickness (example 4 layers)

What is the minimum circuit to edged distance?

What is the minimum distance between inner line/copper and drilling hole

What is the minimum milling slot?

What is the slot tolerance?

What is the Connection Width to panelize the board?

What is the minimum board dimension of V-CUT

What is the material of solder mask?

How to Use Open Parts Library

Could PCB manufacture blind/bury via?

What is the circuit width/spacing for copper thickness

When i choose PCB of 1 layer , can manufacture as 2oz of copper thickness?

How to output drill file correctly ?

Is the board outline could only be rectangle ?

What is the layer?

What is half-hole?

What is the V-cut rules?

If i have 2 different design boards , how should i choose "Panelized PCB"?

What are the circuit board specification ? Like Board Dimension ,Board Thickness or something.

Do you have any real PCB pictures of slots ?

What is the dielectric constant for

Now click the **Drill File** button, close the drill dialog, hit the **Plot** button, and close that dialog as well.

Check your gerbers in gerbviewer (gerbv) to make sure everything is right. In the example below, I made sure that the vias looked right, the drills looked right, the stop-soldermask looked right on the top and bottom, the silkscreen looked right on the top and bottom, the copper looked right on the top and bottom, and that the dimensions looked right. A lot of checking! (And I still missed one small thing, which thankfully turned out to be a gerbv bug and not my mistake.)

Here's what mine looked like for a recent project (front and back, respectively):

the fusion PCB?

What is the maximum temperature of PCB ?

My board size is 6.3cm*5cm, which size should i choose ?

What is the diameter of BGA design ?

What type of material do you user for PCB ?

Predefined EAGLE Layers

How can i do if i do not required to make plated through holes?

Contact support

Give feedback

Knowledge Base

Assurance 2

Fusion Ordering FAQ 16

Fusion Specification 7

Fusion technical support 65

General 2

Ordering 8

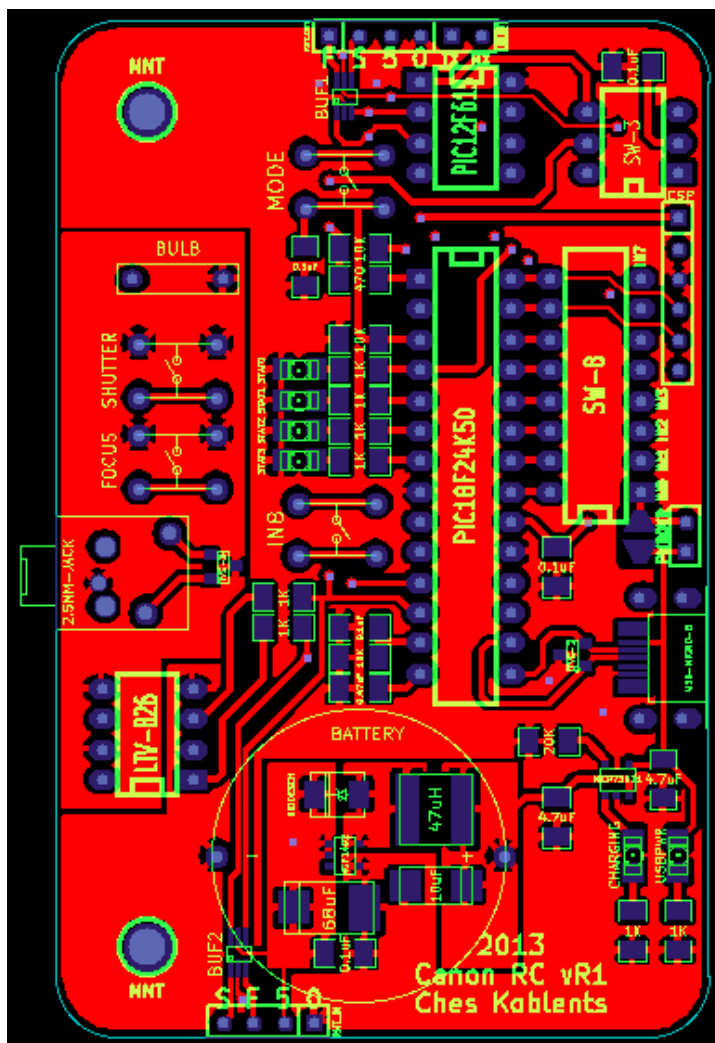
Products' FAQ 80

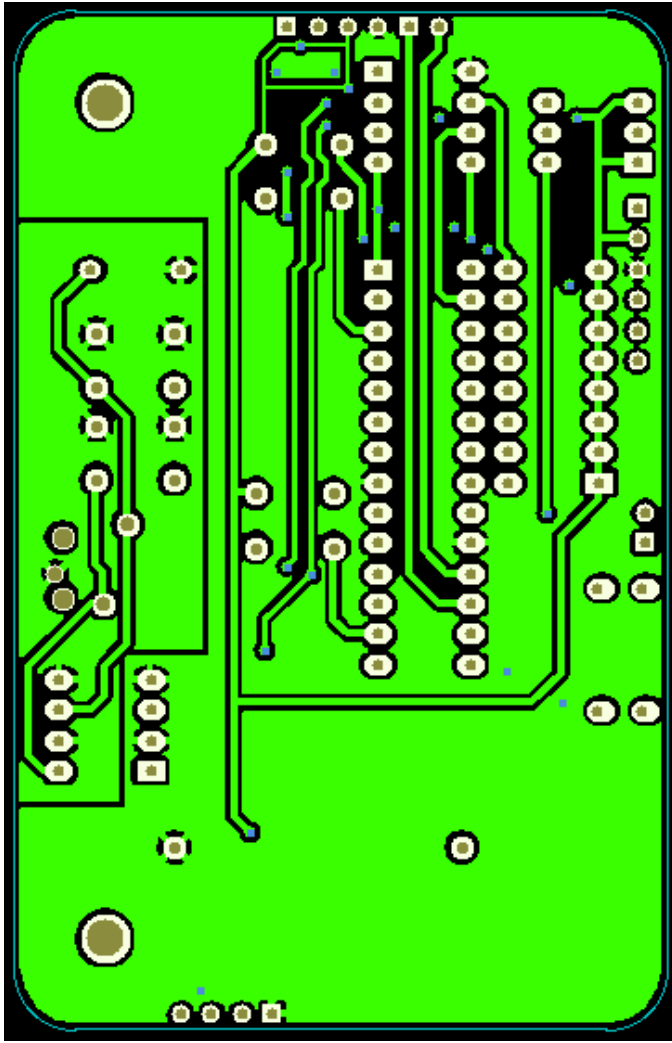
Shipping 9

Tech Support 7

All articles

seedstudio





These gerbers were generated from a layout that looked like this (signal planes hidden):

Disclaimers: I'm not responsible if any of this advice was bad, and you end up wasting money. It worked for me. But please do tell me so I can update this article. Also, as time goes on, this article may change, and Seedstudio Fusion's rules may change as well. Make sure to look at the date to the right before assuming this advice is still valid!

01.09.2016 08:13

This article was helpful

· [Flag this article as inaccurate...](#)