

A guide to SeeedStudio's Fusion PCB Service

Among the many online services for PCB manufacturing, I was impressed by the [SeeedStudio](#)'s offering: they aggregate many orders to offer professional service at a very competitive price.

I tried out the service ordering 10 PCBs, maximum size 5x5cm, with a price of 9.90\$ + 3.52\$ for shipping (to Italy) – less than 10 euro. SeeedStudio doesn't accept Eagle .brd files, so I thought to write this **short guide** with some hints to use Fusion PCB at its best.

Connect to SeeedStudio website to place an **order** adding Fusion PCB service with the features desired (size, color...). Completed the *checkout*, write down the order number.

Download from Fusion PCB [webpage](#), the **Eagle Design Rule** file.

The downloaded archive, in *Fusion eagle* subfolder, contains two files:

- Fusion_eagle_rule_v1.1.dru
- Seed_Gerber_Generater_v0r95_DrillAlign.cam

Copy the first in *dru* subfolder of the main Eagle installation folder, the second in *cam* subfolder:



Open your Eagle project and focus on **Board** layout.

Write in silk layer (top or bottom one) the **order number**: this will speed up the finding of our PCBs (as I wrote SeeedStudio combines many order in a single production *batch*).

To test PCB compliance to Fusion PCB rules, click on **Drc** button in the right *toolbar*, then click on **Load...** and choose the file **Fusion_eagle_rule_v1.1.dru**:



Click on **Check**, if everything is ok, **DRC Errors** window won't show any errors; otherwise you have to modify your PCB following the error messages.

To export the project, from **File** menu click on **CAM Processor**.

In CAM Processor window, click on **File -> Open -> Job...** and choose the **Seed_Gerber_Generator_v0r95_DrillAlign.cam** file:



Click on **Process Job**: Eagle will write some new files in our project's folder.

Before sending those files to Screenshot, I strongly suggest to **check their content** with a *Gerber* editor/viewer: in my experience sometimes I found errors in the exported files (for example if you use proportional fonts, they may change size).

A great *opensource* software is [Gerby](#), which has been *ported* also on Windows platforms.

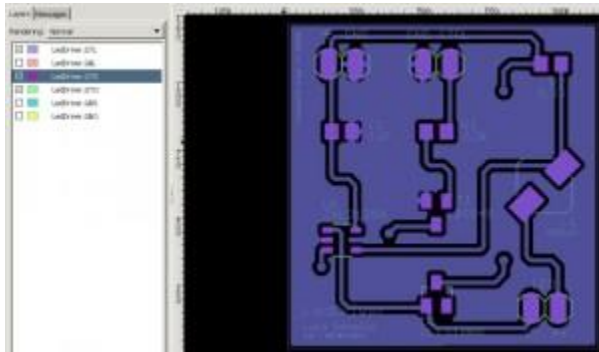
To open your project, click on **File -> Open layer(s)...**

Choose (pressing CTRL key, you can select more than a file at the same time) the 7 files you have to send to Screenshot:

- <pcbname>.GTL (top layer)
- <pcbname>.GBL (bottom layer)
- <pcbname>.GTS (solder stop mask top)
- <pcbname>.GBS (solder stop mask bottom)
- <pcbname>.GTO (silk top)
- <pcbname>.GBO (silk bottom)
- <pcbname>.TXT (drill sizes and positions)

Nome	Dimensione	Data di modifica
eagle.epf	16,8 kB	09/10/2011
LedDriver.zip	16,2 kB	05/09/2011
LedDriver.GTL	26,8 kB	05/09/2011
LedDriver.gpl	1,4 kB	05/09/2011
LedDriver.GBL	13,1 kB	05/09/2011
LedDriver.TXT	203 byte	05/09/2011
LedDriver.GML	226 byte	05/09/2011
LedDriver.GTS	12,2 kB	05/09/2011
LedDriver.GTO	20,7 kB	05/09/2011
LedDriver.GBS	490 byte	05/09/2011
LedDriver.GTP	914 byte	05/09/2011
LedDriver.GBO	226 byte	05/09/2011

Hiding/Showing each *layer* you can check if export was successful:



If everything looks good, prepare an archive with the files listed above.

After having placed the order, send an **email** to pcb@seedstudio.com with the order number as subject and the archive as an attachment... after a couple of days you should receive a order confirmation and you could check order status in the customer page of SeedStudio website:



And after some weeks, here are the PCBs...

