

MMC PCB Fabrication Manual

Version 1.0
2020/05/05



Table of Contents

1. Introduction	3
1.1. What is PCB?	3
1.2. What is PCB Fabrication?	3
1.3. What is gerber file?	3
2. PCB design tools	4
3. How to generate PCB gerber files for fabrication?	4
Eagle	5
Altium designer	5
Kicad	7
4. How to check and confirm PCB gerber files?	8
5. Where to send PCB gerber files for fabrication?	8
Reference	10



1. Introduction

1.1. What is PCB?

PCB (Printed circuit board) is a board that has lines and pads that connect various points together. The board includes traces that electrically connect the various connectors and components to each other. A PCB allows signals and power to be routed between physical devices. Solder is the metal that makes the electrical connections between the surface of the PCB and the electronic components. [1]

1.2. What is PCB Fabrication?

PCB fabrication is the assembly method for circuit boards used in electronic devices. The layers of the board are put together along with the specific surface pattern based on the design files that is used in electronics manufacturing.[2]

1.3. What is gerber file?

Users create their own files for manufacturing based on the design files and send them to the manufacturer. The design files for manufacturing PCB is converted to a standard format known as a Gerber file. This is an industry standard for recording the specification of a PCB.[2]



2. PCB design tools

There are many different tools for designing PCB's and you can select any of the available tools based on your previous experience or the requirements of your design. The process of designing PCB is not in the scope of this document but you can find courses or books that can help you understand the process of designing electronics circuits and PCB's.

Here are the most common PCB design tools:

- [Altium Designer](#) : Paid with Free Trial and student license.
- [Eagle](#) : Free limited version includes 2 schematic sheets, 2 signal layers, and an 80cm² (12.4in²) board area.
- [Kicad](#) : Opensource with GNU's GPL copyright agreement.
- [Fritzing](#) : Opensource version available on github and the paid application is available. Limited options suitable for basic designs.
- [Upverter](#) : Free cloud-based design tool. It requires sign up for usage.

There are other available tools that can be used to design PCBs.

3. How to generate PCB gerber files for fabrication?

Once the PCB board file is created using your favorite PCB design tool then you can generate gerber files which is used by PCB fabrication companies to fabricate your design. The CAM processing tool uses the gerber generator file to create gerber files. The gerber generator file is usually provided by PCB fabrication company of your choice or can be created based on gerber files format which can be found on the website of the PCB fabrication company.

Each PCB board file includes different top and bottom layers and gerber files provide specifications for each of these layers by combining layers together based on the format specified by gerber file generator.

Common layers:

- Top & Bottom Copper
- Top & Bottom Silkscreen
- Top & Bottom Soldermask
- Drill
- Drill Station Info
- Photoplotter Info
- Mill Layer
- Top Paste

There is no standard format for the gerber file generator and there are variations of formats used by PCB fabrication companies.



You can find the final generated gerber files of the projects involving PCB designs for common PCB fabrication companies on our website. You can use the generated gerber files available if you are using one of the fabrication companies listed below or you can create your own gerber files based on PCB design files.

- [Seeed Fusion PCB](#)
- [PCBWay](#)
- [OSH Park ~](#)
- [JLCPCB](#)

We now go over process of generating gerber files using three common PCB design tools.

Eagle

1. Open the board file of your PCB design using Eagle
2. Select File > CAM Processor
3. Click on “Load job file” dropdown menu button and select “Open CAM file”
4. Download or create the gerber generator file (.cam) used by your PCB fabrication company of your choice
5. Navigate in your system and find the directory that includes the gerber file generator file with. cam extension
6. Select the gerber file generator file and click on “Open” button
7. Double check the Output file layers on left menu bar to make sure there are no layers are missing
8. Click on “Process Job” button
9. Select the output directory (Create a new directory called “gerbers” as an example)
10. The process of creating the gerber file will start and finish once you receive a popup message indicating that “Job Proceed Successfully!”
11. Click on “OK” button
12. Navigate to the directory that includes the generated gerber files (“gerbers” directory as an example)
13. Select all the gerber/drill files in the directory that saved them and generate a compressed file in Zip format.
14. You can now submit the compressed Zip file that includes the gerber files to the PCB fabrication company of your choice.

Altium designer

1. Open the board file of your .PCB design file using Altium designer. [3]
2. Select File > Fabrication Outputs > Gerber Files
3. Select General setting in Gerber Setup window
4. Set the units in “inches” precision to “2:5” in General setting
5. Select Layers setting in Gerber Setup window
6. Next, make sure you have the clear outline in mechanical layer.



7. Check the layers in “Plot” column under “Layers To Plot” section for creating gerber files according to fabrication company instructions.
 - Check following layers: GTO, GTP, GTS, GTL, GBL , GBS , GBP , GBO , GKO ,GPT , GPB , GM(Number)
 - There will be no middle layer if you PCB board includes only 2-layers. All our design files include 2 layers which means you need to uncheck all middle layers (G1,G2,G3....)
 - You can use the “Plot Layer” option and select the “Used On” to automatically check the used layers in your design.
8. Select “All Off” in “Mirror Layers” or uncheck every layer in “Mirror Layers” column
9. Uncheck all the layers in the plot column of “Mechanical Layers to Add to All Gerber Plots”
10. Check “Embedded apertures (RS274X)” under “Aperture” setting
11. Click on the “OK” button to generate gerber files.
12. Gerber files are automatically loaded in the Altium cam viewer. You can use “CAMtastic” view to verify that all layers have been generated correctly.
13. The gerber files should now be generated and saved in directory of your project.
14. You can also export gerber files to a new directory using following steps:
 - Select File > Export > Gerber...
 - Click on “OK”
 - Select the path to the directory that you would like to save all the gerber files. (“gerbers” directory as example)
 - Click on “OK” to export all the gerber files
15. Next, Generate the Drilling layer in Excellon format
16. Select File > Fabrication Outputs > NC Drill Files
17. Set the units in “inches” precision to “2:5” in NC Drill Setup window
18. Click on the “OK” button to generate drill files
19. The drill file should now be generated and saved as .TXT file in directory of your project.
20. You can also export file to a new directory using following steps:
 - Select File > Export > Save Drill...
 - Click on “OK”
 - Select the path to the directory that you would like to save the drill file. (“gerbers” directory as example)
 - Click on “OK” to export the drill file
21. Select the “Projects” view and highlight the “CAMtastic” cam file
22. Click on “Save As” under “File” and save the “CAMtastic” cam file.
23. Click on “Save”.
24. Select all the gerber/drill files in the directory that saved them and generate a compressed file in Zip format.



25. You can now submit the compressed Zip file that includes the gerber files to the PCB fabrication company of your choice.

Kicad

1. Open the board file of your design file using Kicad. [4]
2. Select File > Plot
3. Check the layers in "Plot" window for creating gerber files according to fabrication company instructions.
 - Check following layers: Top Copper (F.Cu), Soldermask (F.Mask), Silkscreen (F.SilkS), Bottom Copper (B.Cu), Soldermask (B.Mask), Silkscreen (B.SilkS), Board outline (Edge.Cuts)
4. Set the "Output directory" to save the generated gerber files. You can also use "Browse..." button to set the path to the directory.
5. Uncheck "Plot sheet reference on all layers" option.
6. Uncheck "Plot footprint values" option.
7. Check "Exclude PCB edge layer from other layers" option.
8. Check "Use Protel filename extensions" option.
9. Click on "Plot" button to generate and save gerber files in the specified output directory to save gerber files.
10. Next, configure the settings to generate drill file
11. Click on "Generate Drill File" button.
12. Set the "Output directory" to save the generated drill file. You can also use "Browse..." button to set the path to the directory. Use the same output directory as you selected to save the gerber files.
13. Check "Inches" as drill units.
14. Check "Decimal Format" as zeros format.
15. Check "PostScript" as drill map file format.
16. Check the "Merge PTH and NPTH holes into one file" option
17. Click on "Drill File" or press "Enter" to generate the drill file.
18. Click on "Close" to close "Drill Files Generation" Window.
19. Click on "Close" again to close "Plot" Window.
20. Verify the generated gerber files using a gerber viewer tool to make sure that all layers have been generated correctly.
21. Select all the gerber/drill files in the directory that saved them and generate a compressed file in Zip format.
22. You can now submit the compressed Zip file that includes the gerber files to the PCB fabrication company of your choice.



4. How to check and confirm PCB gerber files?

It is recommended to double check or triple check the final generated gerber files of your design to make sure there are no issues with process of gerber files generation. The PCB fabrication companies mostly provide online gerber file viewer tools that you can use to confirm your design once you upload your gerber files compressed as a Zip file. However, you can use other gerber file viewer to check your design files before submitting the gerber files.

The following gerber viewer tools are recommended:

- [Online Gerber Viewer](#)
- [gerbv](#)

Select all the generated gerber file layers in the “gerbers” directory and view them to make sure all the layers are aligned.

5. Where to send PCB gerber files for fabrication?

The process to send the gerber files for fabrication may differ from company to company but there are general steps you can follow to send your gerber files for fabrication.

1. Select the fabrication company (Here are our recommended fabrication companies)
 - Seeed Fusion PCB
 - PCBWay
 - OSH Park ~
 - JLCPCB
2. Create an account on PCB fabrication website or use temporary login if it is available
3. Select the fabrication service
4. Upload the compressed Zip file that includes your gerber files
5. Select the available options based on your design requirements. The standard options for most of our projects:
 - PCB Layers: 2 Layers
 - PCB dimensions should update based on your uploaded gerber files, but you can manually entry the dimensions as well
 - PCB quantity can vary based on your requirements (Minimum 5)
 - PCB Color can be changed if the fabrication company offers the option. The default option has the fastest turnaround time.
 - Surface Finish: HASL
 - Copper Weight, Minimum Drill Hole Size, Trace Width , Minimum Solder Mask Dam doesn't require to be changed and the default value works
6. Check the uploaded gerber files using online gerber viewer tool if it is offered by the fabrication company and make sure there are no issues.



7. Add the uploaded design to the order
8. Add another design if you are doing multiple orders
9. Proceed to the checkout and enter your billing and shipment information
10. Select the shipping method
11. Submit your order for production
12. The technician will contact you for additional information or corrected files if there are issues with your design files.
13. You will receive an email when your design goes to production
14. You will receive another email once your PCBs are produced and shipped

This includes the instructions for fabrication of PCB designs based on generated gerber files.



Reference

1. <https://learn.sparkfun.com/tutorials/pcb-basics/all>
2. <https://www.hi5electronics.co.uk/what-is-pcb-fabrication/>
3. <https://support.jlcpcb.com/article/42-how-to-export-altium-pcb-to-gerber-files>
4. <https://support.jlcpcb.com/article/44-how-to-export-kicad-pcb-to-gerber-files>