

[Blog](#) > [Generate gerber file from Altium](#)

Generate gerber file from Altium

 by: [Anson Bao](#) Apr 25,2017 11554 Views 2 Comments Posted in [Help Center](#)
[gerber file](#) [pcb design](#) [altium](#) [pcb software](#)

Step1: The work before generate the Gerber file

Before generate Gerber file, you need to set criteria, **but for most of PCB designers, these are not necessarily.** The following is a brief explanation of these settings, You can also ignore these setting when generate Gerber file.

1. The settings of PCB shape and size, generally are mechanic1,2 layers.

2. Before generate the Gerber file, the preprocessed of PCB file.

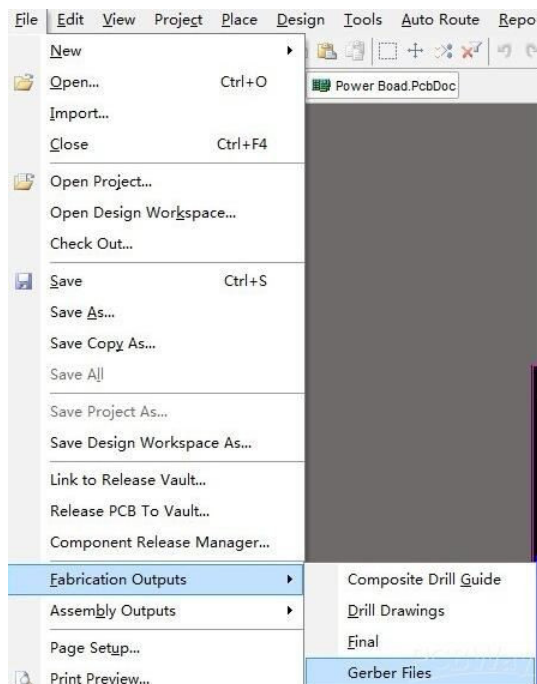
The preprocessed include:

- Add break away rails of PCB
- Add stamp hole.
- Add the machine jack.
- Add mouting hole
- Add drilling description.
- Add dimension description.
- Set the origin

Step2: Generate Gerber file

1. Open the Gerber Files.

File -> FabricationOpuputs -> GerberFiles.



2. Parameter setting.

a. "General" setting

HOT TAG

- [printed circuit board](#)
[PCB](#)
- [PCB board](#)
[PCB Design](#)
- [circuit board](#)
- [Printed Circuit Boards](#)
[PCB Layout](#)
- [PCB industry](#)
[Circuit Boards](#)
- [Printed Circuit Board \(PCB\)](#)
- [PCB production](#)
[PCB assembly](#)

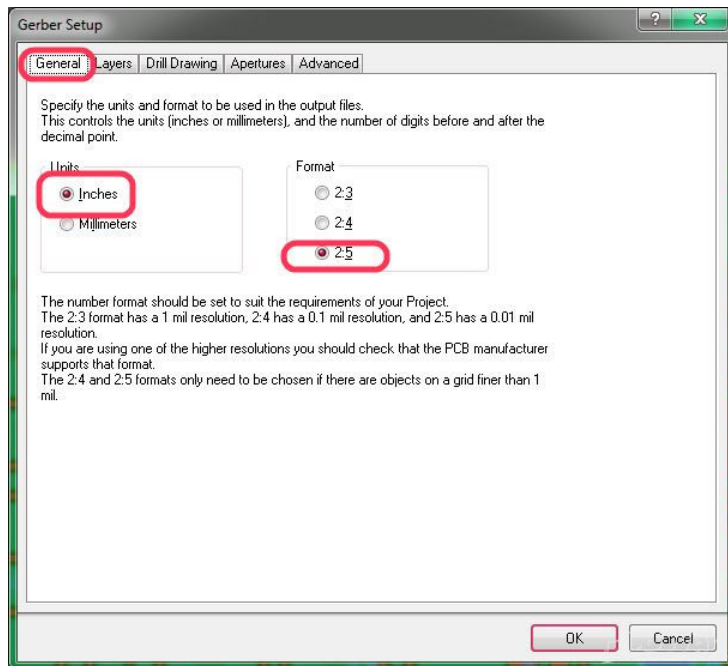
Categories

IF YOU MISSED IT

- 1 [How to generate Gerber from Sprint Layout 6.0](#)
- 2 [How to Read & Understand a Circuit Diagram](#)
- 3 [4 important tips for PCBWay 24-Hour Express Service](#)
- 4 [Necessary file for PCB Layout](#)
- 5 [Check your order's fabrication & processing status online](#)
- 6 [Gerber File Extention from Different Software](#)
- 7 [Generate Gerber file from Kicad](#)
- 8 [How to deal with the theoretical conflicts on wiring ?](#)
- 9 [How to avoid high frequency interference?](#)

Set Units to "Inches"

And Format to "2:5"



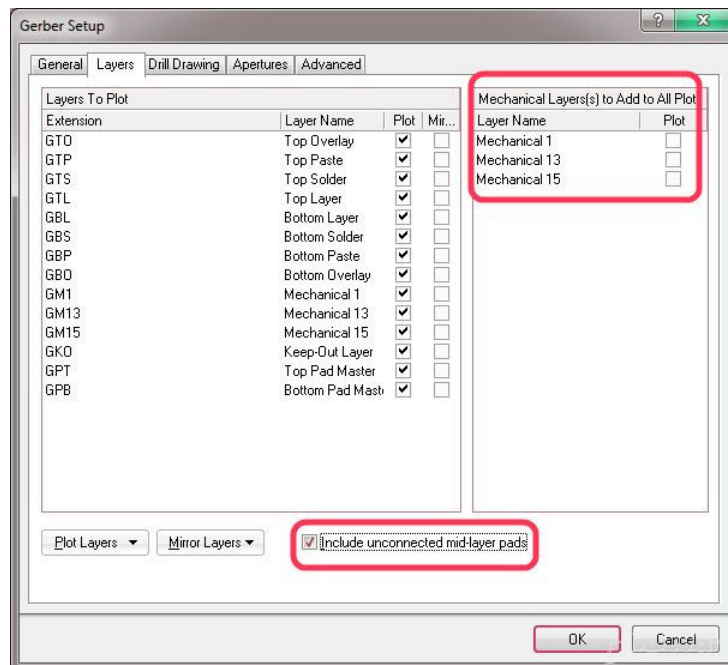
b "Layers" setting

Select "Used On" in Plot Layers

Select "All Off" in Mirror Layers

Check "include unconnected mid-layer pads"

Uncheck all in Mechanical Layers to Add to All plot

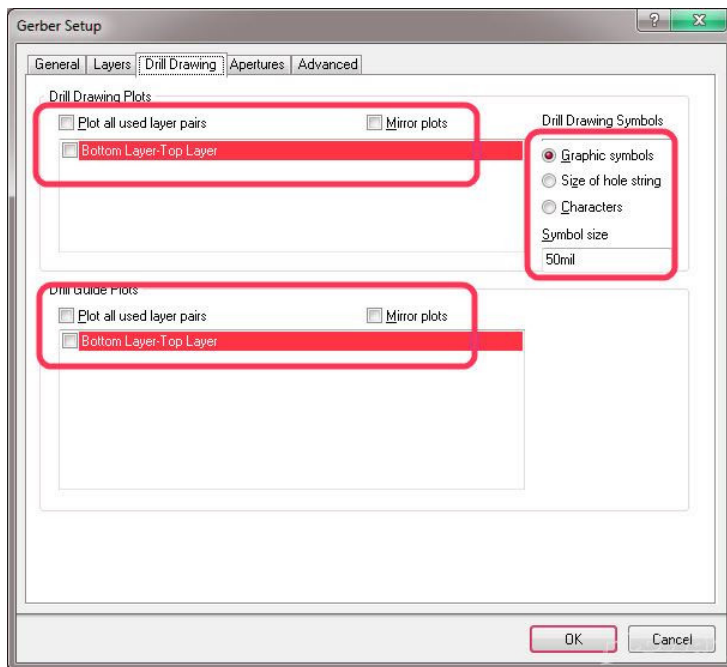


c. "Drill Drawing" setting

Uncheck all boxes.

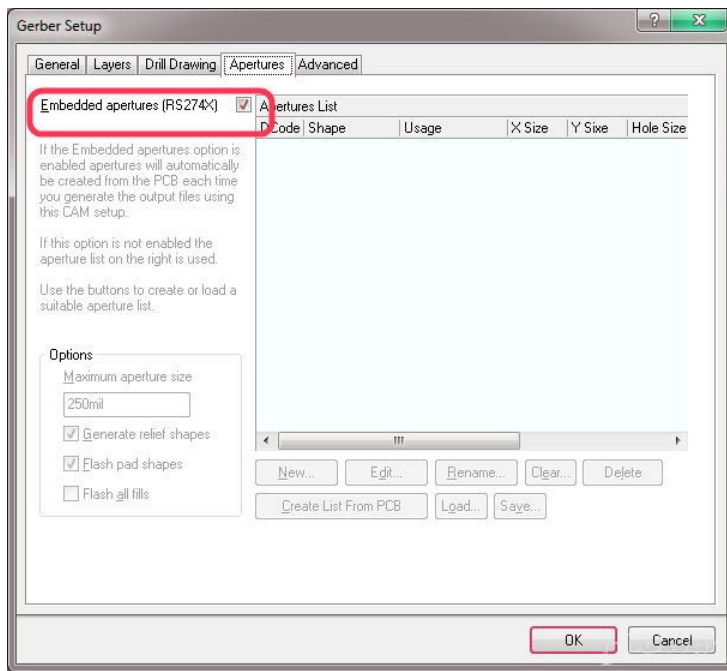
Select "Graphic Symbols" in Drill Drawing Symbols





d. "Apertures" setting

Select RS274X



e. "Advanced" setting

Select "Separate file per layer" for Batch Mode

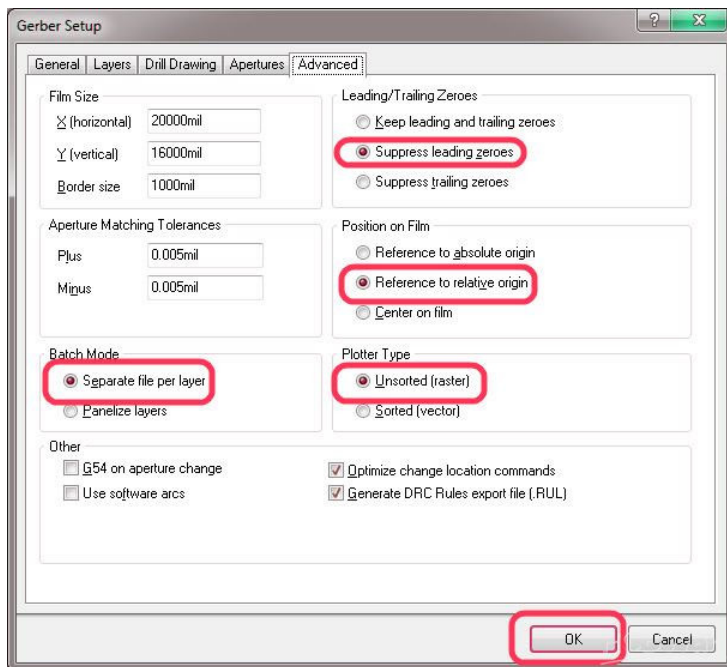
Select "Suppress leading zeroes" in Leading/Trailing Zeroes

Select "Reference to relative origin" in Position on Film

Select "Unsorted" in Plotter Type

Click "OK"





Gerber Files are automatically loaded in the Altium cam viewer. This tool allows you to verify that all layers have been generated correctly and that they are all in positive mode.

3. Generating NC Drill File

a. In the PCB view, and select : File / Fabrication Outputs / NC Drill Files

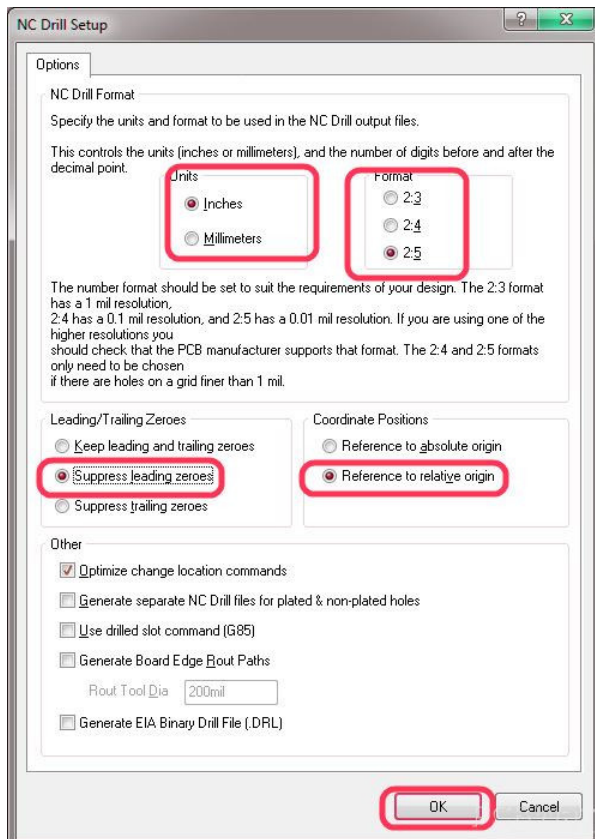
Select "Inches" in Units

Select "2:5" in Format

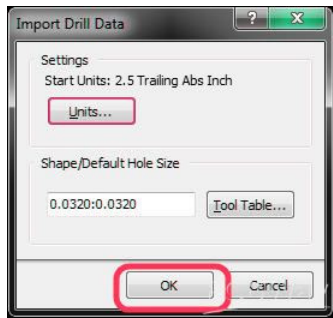
Select "Suppress leading zeroes" in Leading/Trailing Zeros

Select "Reference to relative origin" in Coordinate Positions

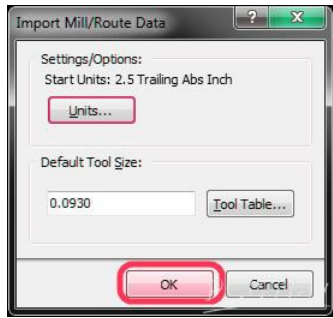
Click "OK"



b. Import Drill Data : Click "OK"




c.Import Mill/Route Data : Click “OK”



4. Package Gerber File

Compress all the files in a single .zip file.All Gerber and NC Drill files generated by Altium Designer are automatically saved in the Project output folder.

Leave a Comment (2)

Share to:   

Previous: How to generate Gerber files from Eagle

Next: Holiday Notice - International Labors' Day

Related articles

- The Purpose of PCB Design
- Problems Encountered in PCB Design and Their Solutions
- New PCB design partner introduction - ValiTech
- New PCB design partner intruduction -3D PHOTONiX

Comments (2)


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


 t***bug

good

May 03,2017

0 Comments

Reply


 N****nks

Thanks for this!


Feb 23,2018

0 Comments

Reply



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

 © COPYRIGHT 2019

[ABOUT US](#) | [CONTACT US](#) |

