



HOME	PROJECTS	BLOG	PUBLISH NEW PROJECT
------	----------	------	---------------------

How to generate Gerber from Eagle

Using EAGLE: Board Layout

755072

Refer friends and Earn \$25

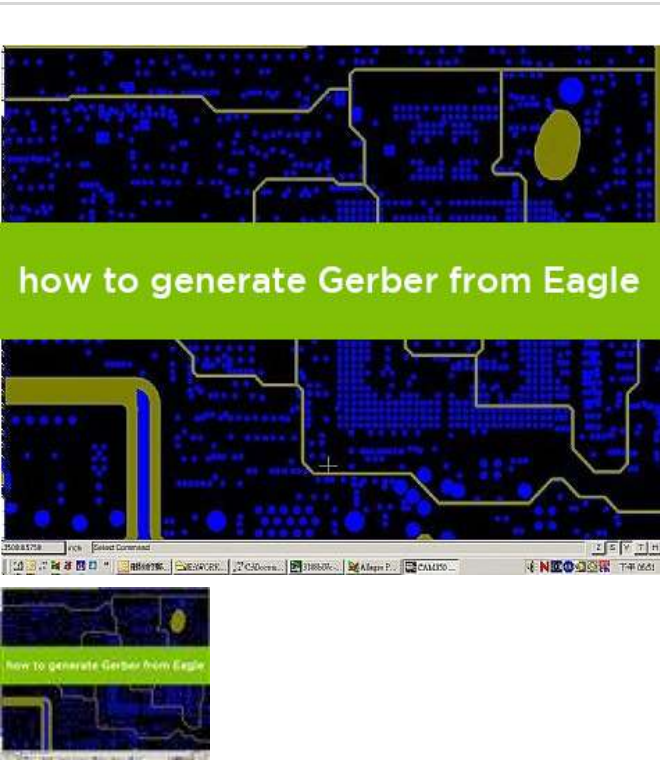
Total:1 Steps | By P***En |
In SOFTWARE

pcbway.com

Step 1: Using EAGLE: Board Layout

Generating Gerbers

When you’ve finalized your design, the last step before sending it off to the fab house is to generate gerber files. Gerber files are kind of a “universal language” for PCB designs. EAGLE is far from the only PCB CAD software out there, and its design files are nothing like those of Orcad or Altium. Fab houses can’t possibly support every piece of software out there, so we send them the gerber files instead.



PCB Instant Quote

Dimensions:

Length

 X

Width

 mm

Quantity:

Choose Num (pcs)

Layers:

2 Layers

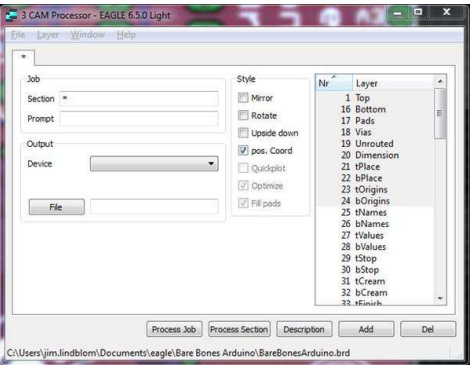
Thickness:

Gerber files – note the plurality – each describe single layers of the PCB. One gerber might describe the silkscreen, while another defines where the top copper is. In all, we’ll generate seven gerber files to send to the fab house.

CAM Processor

Before we get too much further, you’ll need to download another definition file: SparkFun’s CAM file.

Then, load up the CAM processor by clicking the CAM icon – – which will open up this window:



From here, go to the **File** menu, then go **Open > Job....** In the file browser that opens, select the **sfe-gerb274x.cam** file that you just downloaded. Now the CAM processor window should have a series of tabs: “Top Copper”, “Bottom Copper”, “Top Silkscreen”, etc. Each of these

1.6mm

Quote Now

Project By



P***En
UNITED STAT...

Follow

Collection

SOFTWARE

COMPLETED PROJECT

Gerber Eagle

Team

This project was created on 5/5/2015 5:42:47 PM

Project Copyright for members,
Report contact: service(at)pcbway.com

tabs define how to create one of the gerber files. Now all you have to do is click ***Process Job***. If you haven't saved recently, it'll prompt you to.

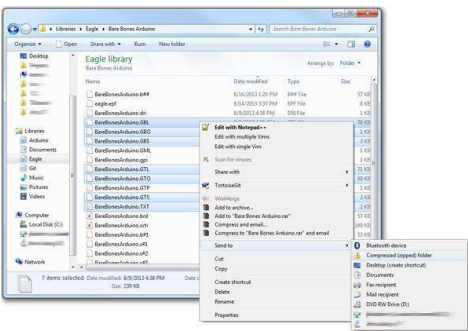
The gerber generation process should be pretty quick. Once it's run its course, have a look in your project directory, which should have loads of new files in it. In addition to the board (BRD) and schematic (SCH) files, there should now be a .dri, .GBL, .GBO, .GBS, .GML, .gpi, .GTO, .GTP, .GTS, and a .TXT. Meet the Gerbers!

Gerber File	Extension
Bottom Copper	GBL
Bottom Silkscreen	GBO
Bottom Soldermask	GBS
Top Copper	GTL
Top Silkscreen	GTO
Top Soldermask	GTS
Drill File	TXT
Drill Station Info File	dri
Photoplotter Info File	gpi

Mill Layer	GML
Top Paste	GTP

Delivering the Gerbers

The process of sending gerber files varies by fab house. Most will ask you to send them a zipped folder of select files. Which gerber files? Check with your fab house again (e.g. Advanced Circuits and OSH Park’s guidelines), but usually you want to send them GTL, GBL, GTS, GBS, GTO, GBO and the TXT files. The GTP file isn’t necessary for the PCB fabrication, but (if your design had SMD parts) it can be used to create a stencil.



So zip those gerbers up. Play the waiting game. And get ready

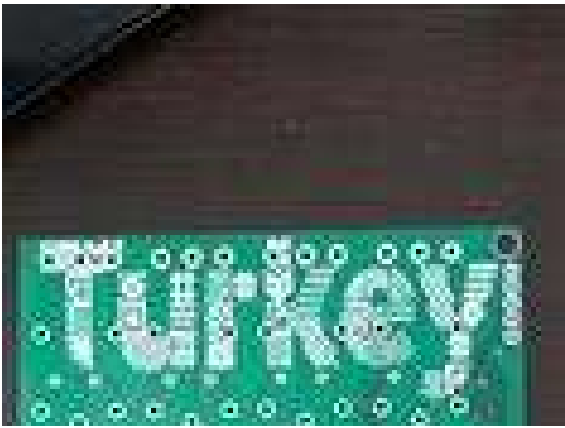
to assemble your very own PCB!

DISCUSSIONS

Sign in or become a member to
leave your comment

comment

RELATED PROFILES



132 0 0



Completed project



234 0 0



C***es Completed project



240 0 0



h***ar Completed project

**THE PCBWAYARDS THAT RECOGNIZE THE TALENT AND EFFORT OF THE BEST ELECTRON DESIGNERS
IN THE WORLD.**

© COPYRIGHT 2014

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