



San Francisco State University

Electrical Engineering

How to Prepare Gerber File from EAGLE

Step 1: download the gerber file processor (.cam)

- Download the file “engr301-gerb.cam” from iLearn
- Save the file into the folder “(where you install EAGLE)/cam”

Step 2: prepare the CAM Processor

When you have finalized your design, the last step before sending the design to the fab house is to generate gerber files.

Gerber files are kind of a “universal language” for PCB designs. Each gerber file describes single layers of the PCB. One gerber might describe the silkscreen, while another defines where the top copper is. In all, we will generate the following 7 gerber files:

- file_name.**GTL** (top copper)
- file_name.**GTO** (top silkscreen)
- file_name.**GTS** (top soldermask)
- file_name.**GBL** (bottom copper)
- file_name.**GBO** (bottom silkscreen)
- file_name.**GBS** (bottom soldermask)
- file_name.**TXT** (drill file)

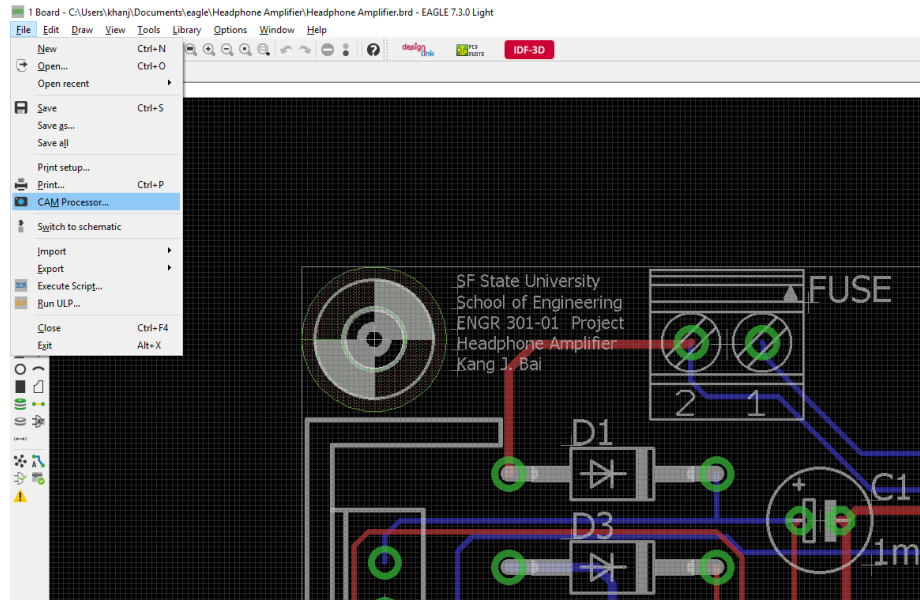


Fig. 1: EAGLE Layout Page

First, load up the CAM Processor from the *File menu* followed by *CAM Processor*.

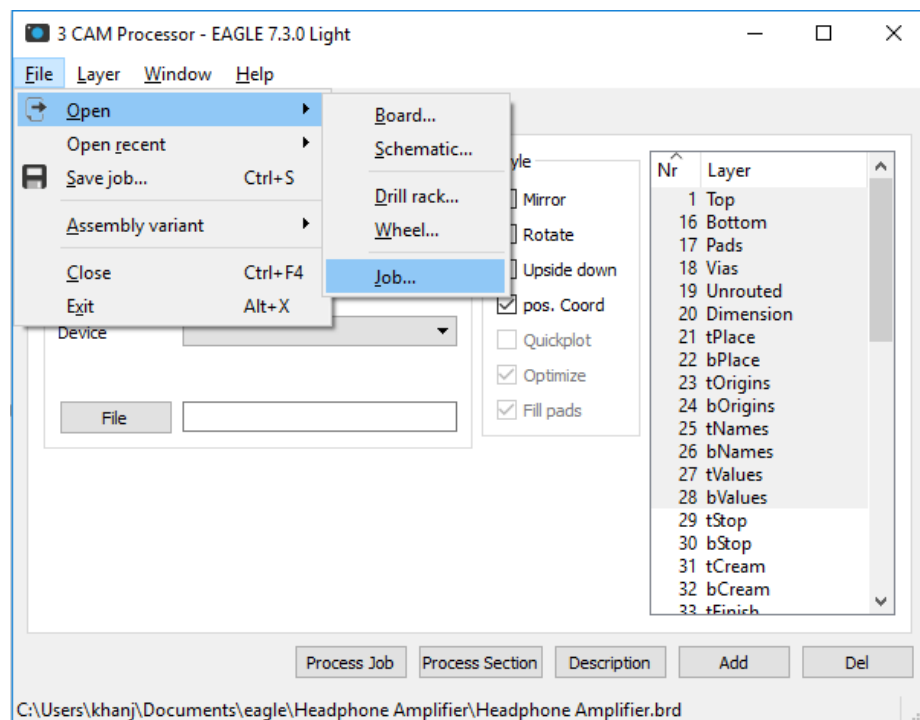


Fig. 2: CAM Processor Window

From the CAM Processor Window, go to the **File menu** followed by **Open** and **Job**. In the file browser that opened, locate the file (engr301-gerb.cam) that you just downloaded.

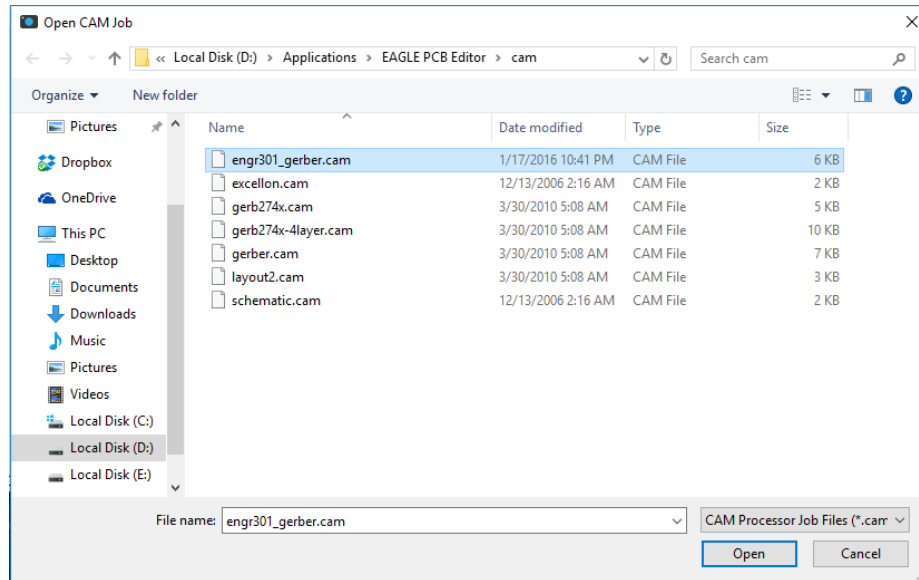


Fig. 3: Open the CAM Job

Step 3: convert the layout into gerber files

Now the CAM processor window should have a series of tabs: “Top Copper”, “Bottom Copper”, “Top Silkscreen”, etc. Each of these tabs define how to create one of the gerber files. Now all you have to do is click **Process Job**.

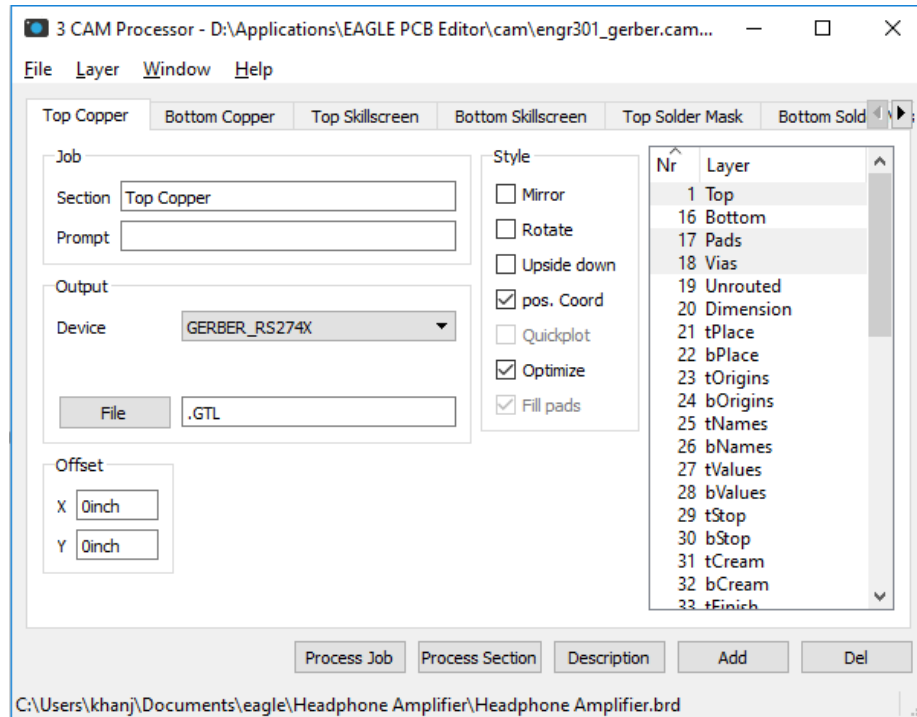


Fig. 4: Process Job for Gerber Files

The gerber generation process should be pretty quick. Once it has run its course, have a look in your project directory, which should have loads of new files in it.

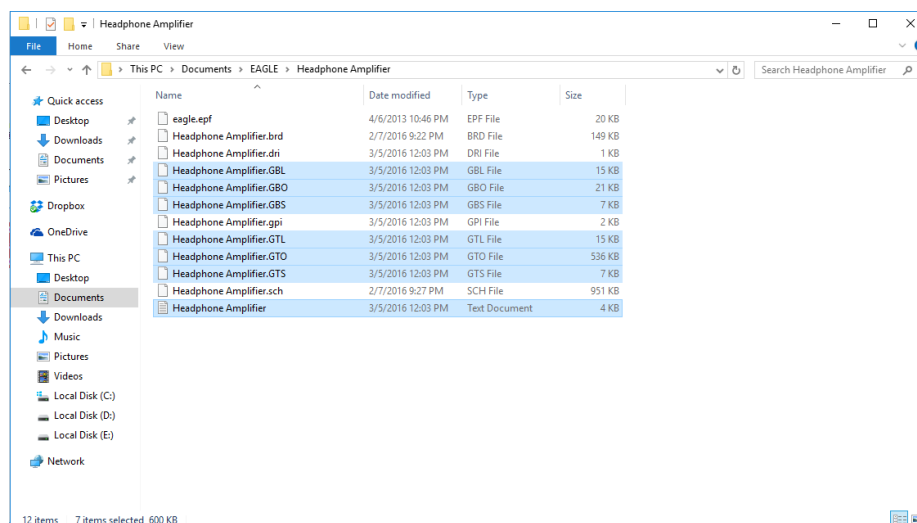


Fig. 5: Gerber Files

Step 4: delivering the gerber files

The process of sending gerber files varies by fab house. Most will ask you to send them a zipped folder of selected files.

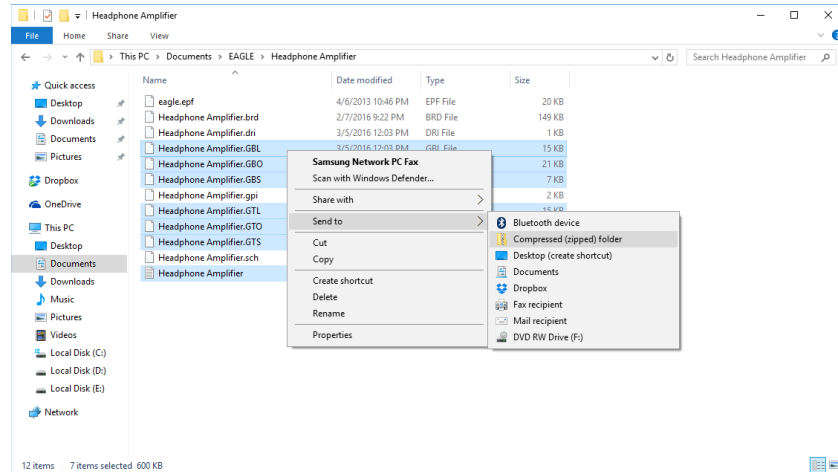


Fig. 6: Compressed the Gerber Files

Note: the final zipped folder MUST include all 7 gerber files (GTL, GTS, GTO, GBL, GBS, GBO, TXT). Missing one of these files, the fabrication process cannot be done.