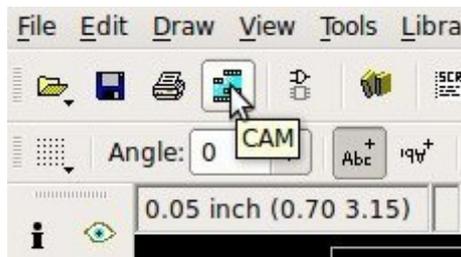


How To Create a Gerber File Using Eagle

A Gerber file for each section of your electronic circuit design is what you need if you want to create a PCB.

Step 1: Open the CAM Processor



In Eagle, open Board view. Click the “CAM” button or choose “File->CAM Processor”. This will open the CAM Processor tool that is used to generate the files.

Here you can define the sections you want to create files for.

Step 2: Open a predefined job

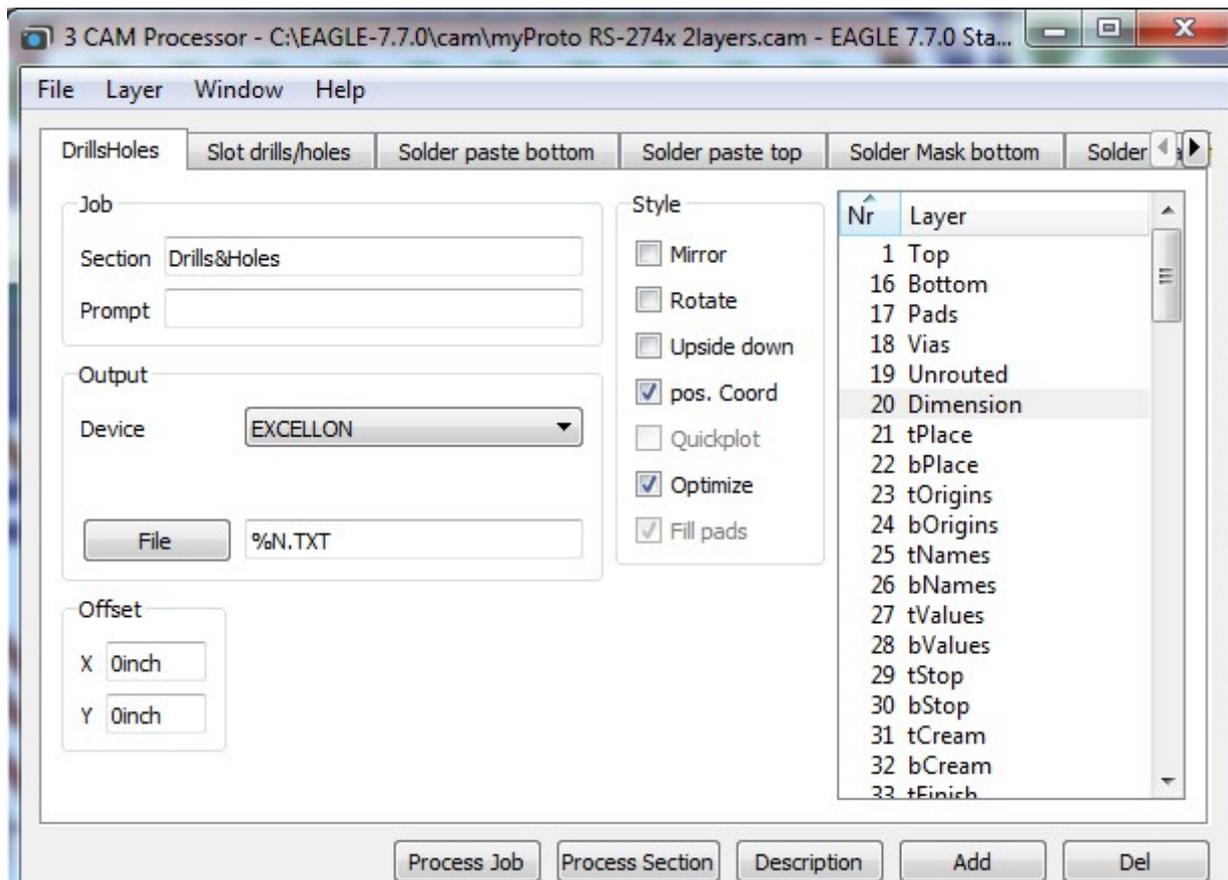
To simplify creating Gerber files, you can use the predefined job of myProto. This is the file myProto RS-274x 2layers.cam for two-layer circuits and the file myProto RS-274x 4layers.cam for four-layer circuits.

We recommend that you put these two files in the directory C:\EAGLE-x.x.x\cam

To open it in the CAM Processor click “File->Open->Job”

Browse to your /eagle/cam/ folder, and you should see a file called. Choose it and click “Open”.

You will now see several tabs in the CAM Processor. Each of these tabs will generate a Gerber file.



Step 3: Process job

Click on « Process Job » to output the Gerbers files. You will find the files in your project directory folder

Step 4: Check output files

You will get 10 RS274-x format gerber files with myProto RS-274x 2layers :

Top Layer: pcbname.GTL

Silk Top: pcbname.GTO

SolderMask Top: pcbname.GTS

SMD paste Top: pcbname.GTP

Bottom Layer: pcbname.GBL

Silk Bottom: pcbname.GBO

SolderMask Bottom: pcbname.GBS

SMD paste Bottom: pcbname.GBP

Solt Drills/Hole: pcbname.GML ----> the Milling Layer(layer46)

NC Drill file:pcbname.TXT

Board Outline: pcbname.GKO

