How to Create CAM files from Eagle for PCB Manufacture

A Gerber file has to be created for each layer you want on your circuit board. If you just want a single sided board, with no solder mask or silk screen (ie just a copper layer), you just need to create two files, one for the copper layer, one for the board dimensions. More complex boards, eg double sided with solder mask and silk screen on both sides will require 7 Gerber files. A drill file is needed as well.

Creating Gerber files:

Select CAM Processor from the Eagle File Menu
Under Output > Device select GERBER_RS274X

Create the Top Copper Gerber
- Under Job> Section, choose a name for the layer.
- Under Output, choose a name for to save the file for this layer.
- In the layer list, choose 1 Top, 17 Pads, 18 Vias.

Create the Top Solder Mask Gerber.
- Click Add, this will create a new tab.
- Under Job > Section, choose a name for the layer.
- Under Output, choose a name for to save the file for this layer.
- In the layer list, choose 17 Pads, 18 Vias, 31 tCream

Create the Top Silk Gerber.
- Click Add
- Under Job > Section, choose a name for the layer.
- Under Output, choose a name for to save the file for this layer.
- In the layer list, choose 21 tPlace, 25 tNames, and 27 tValues if you want component values printed.

Create the Bottom Solder Mask Gerber.
- Click Add,
- Under Job > Section, choose a name for the layer.
- Under Output, choose a name for to save the file for this layer.
- In the layer list, choose 17 Pads, 18 Vias, 32 bCream.

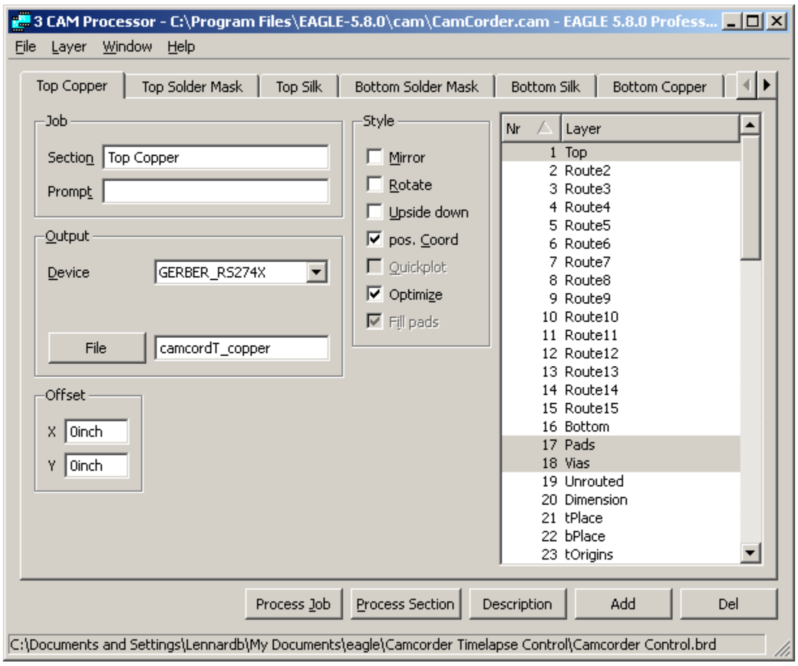
Create the Bottom Silk Gerber.
- Click Add.
- Under Job > Section, choose a name for the layer.
- Under Output, choose a name for to save the file for this layer.
- In the layer list, choose 22 bPlace, 26 bNames.

Create the Bottom Copper Gerber.
- Click Add.
- Under Job > Section, choose a name for the layer.
- Under Output, choose a name for to save the file for this layer.
- In the layer list, choose 16 Bottom, 17 Pads, 18 Vias.

Create the Dimension Gerber.
- Click Add.
- Under Job > Section, choose a name for the layer.
- Under Output, choose a name for to save the file for this layer.
- In the layer list, choose 20 Dimension

Create the Holes and Drills Gerber.
- Click Add.
- Under Job > Section, choose a name for the layer.
- Under Output, choose a name for to save the file for this layer.
- In the layer list, choose 44 Drills, 45 Holes

- Under File, choose Save Job, so you can use this for other boards in the future.
- Finally, click on Process Job, this will create lots of Gerber files.

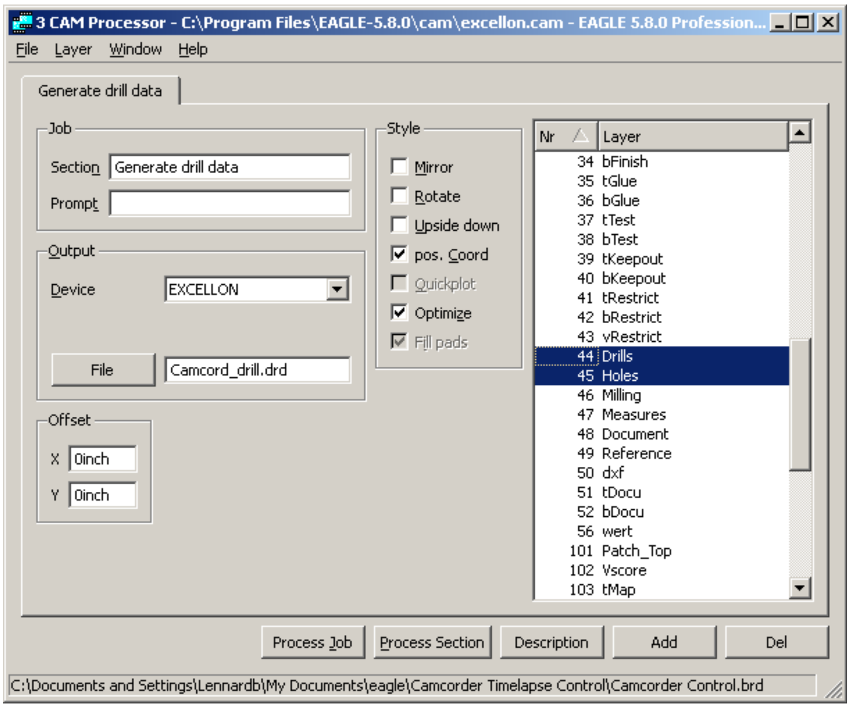


Creating the Drill File:

- In the CAM Processor Window, go to File > Open > Job, and select excellon.cam
- Under Job > Section call it Generate drill data
- Under Output > Device, select EXCELLON
- Under Output, in the text box next to File, choose a name to save as.
- In the layer list, choose 44 Drills, 45 Holes.

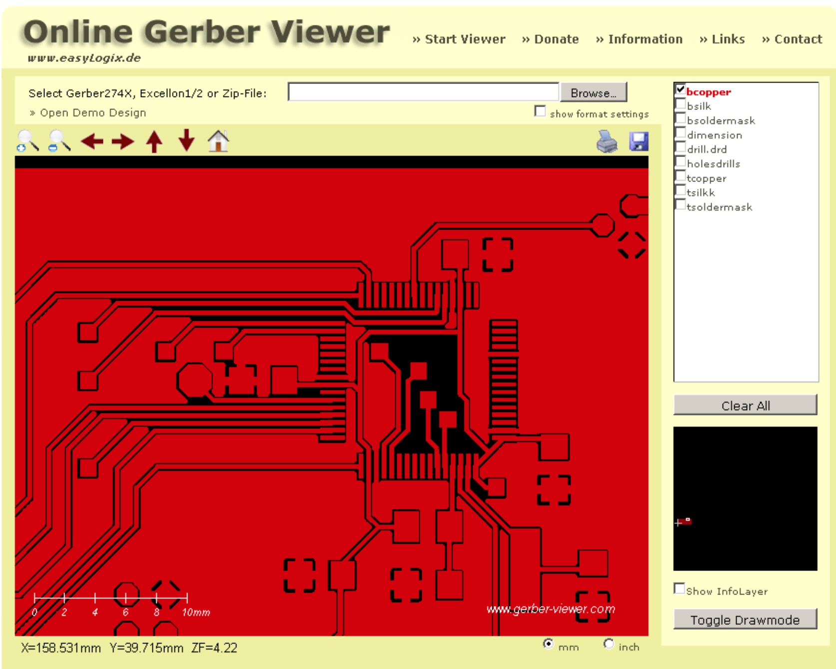
- Finally, click on Process Job, this will create a drill file.

- The .gpi files are not needed, but zip up all the other files that are created and send to the PCB manufacturer.



Checking the Gerber Files:

- You can check your files look OK by using this free online Gerber File viewer:
  http://www.gerber-viewer.com/default.aspx



Finally, add some nutmeg and egg whites and place in a warm oven for 40 minutes. Whisk briefly and let stand for 2 hours before icing.