Quick tutorial

## Generating gerber files in Eagle cadsoft

Mikael Larsmark

EISLAB

Lulea University of Technology

Creating gerber files in Eagle

1. Run the command "run drillcfg" which will create a tool file. Save the file to a sub folder called Gerber or something similar.

2. After this is done you need to start the CAM processor. Select to open a job called excellon to create the NC drill files (There should be a pre-made job in the <eagle folder>/cam). Click the File button and select your sub folder that you created earlier and the file should end with .drd. If you skip writing the file name and just write .drd for example it will be named as your board file but with the file ending you wrote. To produce the drill file just press the "Process Job" button and it will be produced.
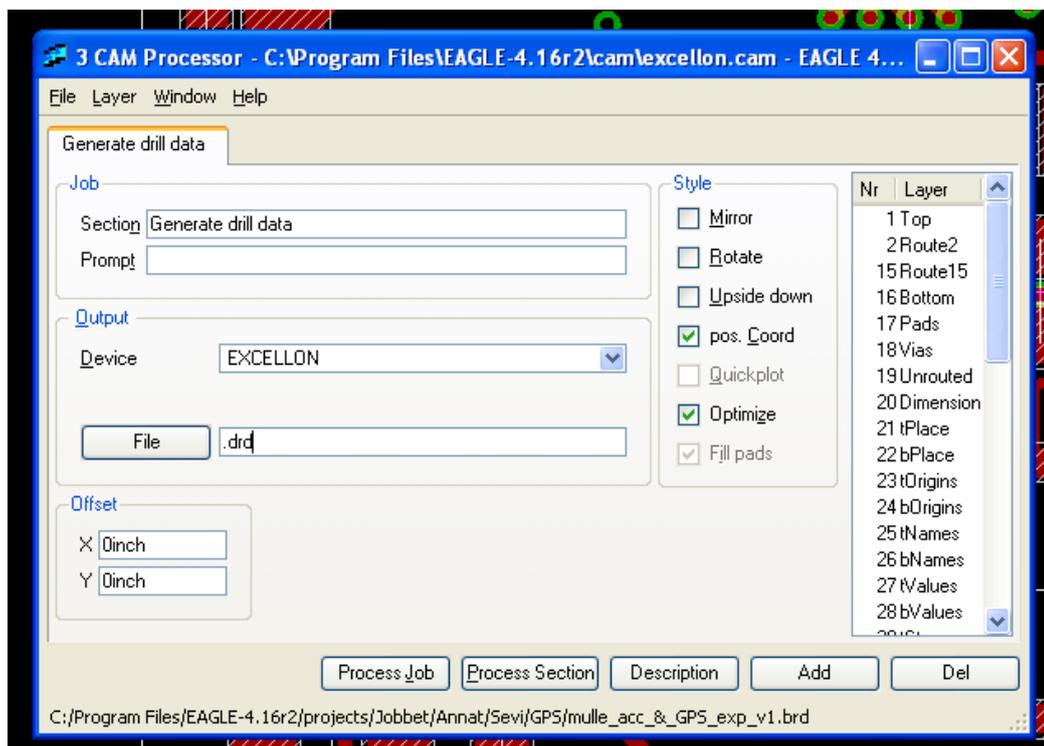
3. Next we wish to produce all the gerber files that represent the different layers on your board. Select the job named "gerb274x.cam" (<eagle folder>/cam).
   The layers that are usually used for 2 layer boards are
   - Component side (.cmp) – The top layer
   - Solder side (.sol) – The bottom layer
   - Silkscreen CMP (.plc) – The silk screen on the component side
   - Silkscreen SOL (.pls) – The silk screen on the solder side
   - Solder stop CMP (.stc) – The solder stop mask for the component side
   - Solder stop SOL (.sts) – The solder stop mask for the solder side
   - Dimmension (.dim) – The dimmension layer (layer 20 in eagle)

   Do not mirror any of these layers and remember to put the files in the correct folder as when you created the drill files. Press the button "Process Job" and the gerber files will be produced. If you need to produce files for more than 2 layers you just add the right layers (the same as component/solder side but insted of Top/bottom you select your new layer).
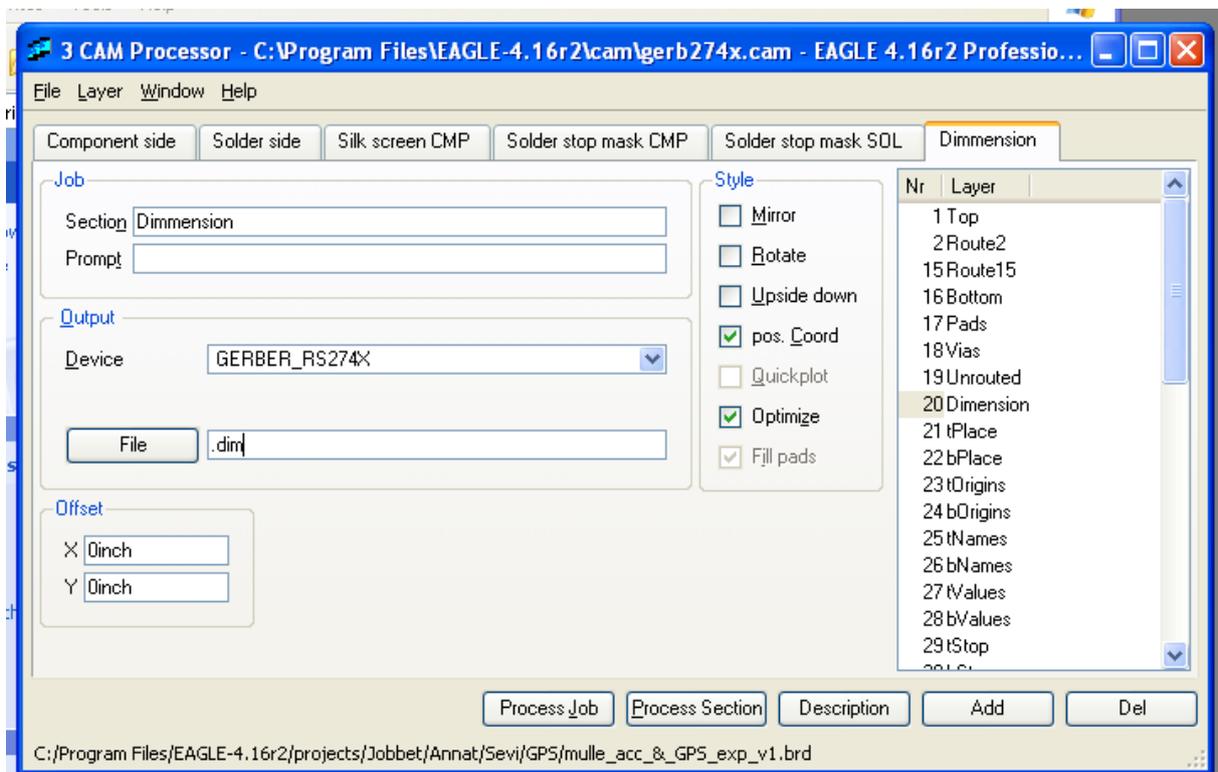
4. Use a gerber viewer to preview the files. There are a bunch of freeware viewers avaible on the market. The one I prefer using is called GcPrevue and can be downloaded from their website: http://www.graphicode.com/
   The drill files are usually default in eagle in 1:4 inch format but this might differ depending on the setting in the define file located in the bin folder (<eagle folder>/bin/eagle.def). You can add all the layers to see that everything lines up properly. The most interesting thing is to check that the drill holes are aligned properly with the pads.

5. I usually also make a description file which explains what the files are. This is to simplify for the PCB manufacturer, my file looks like:

.cmp - Top copper layer (Layer 1)
.sol - Bottom copper layer (Layer 16)
.plc - Silkscreen TOP
.pls - Silkscreen BOTTOM
.stc - Solder stop mask TOP
.sts - Solder stop mask BOTTOM
.dim - Outer Dimmension
.drd - NC drill file
.drl - Tool size specification

Put the files in a zip-file and send it to the PCB manufacturer.