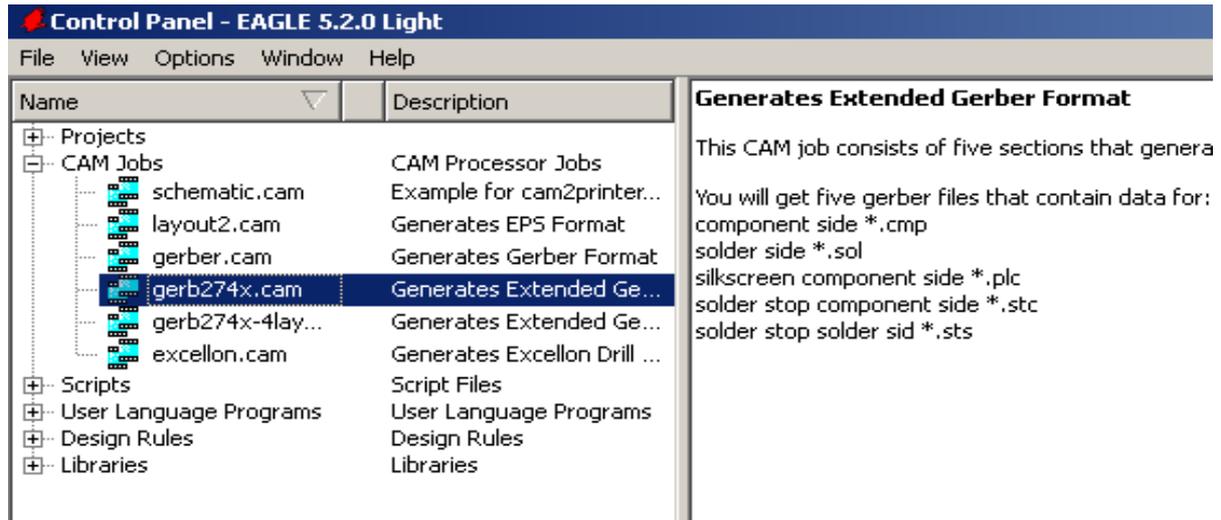# Exporiting PCB production files fom EAGLE

Written by Tsvetan Usunov
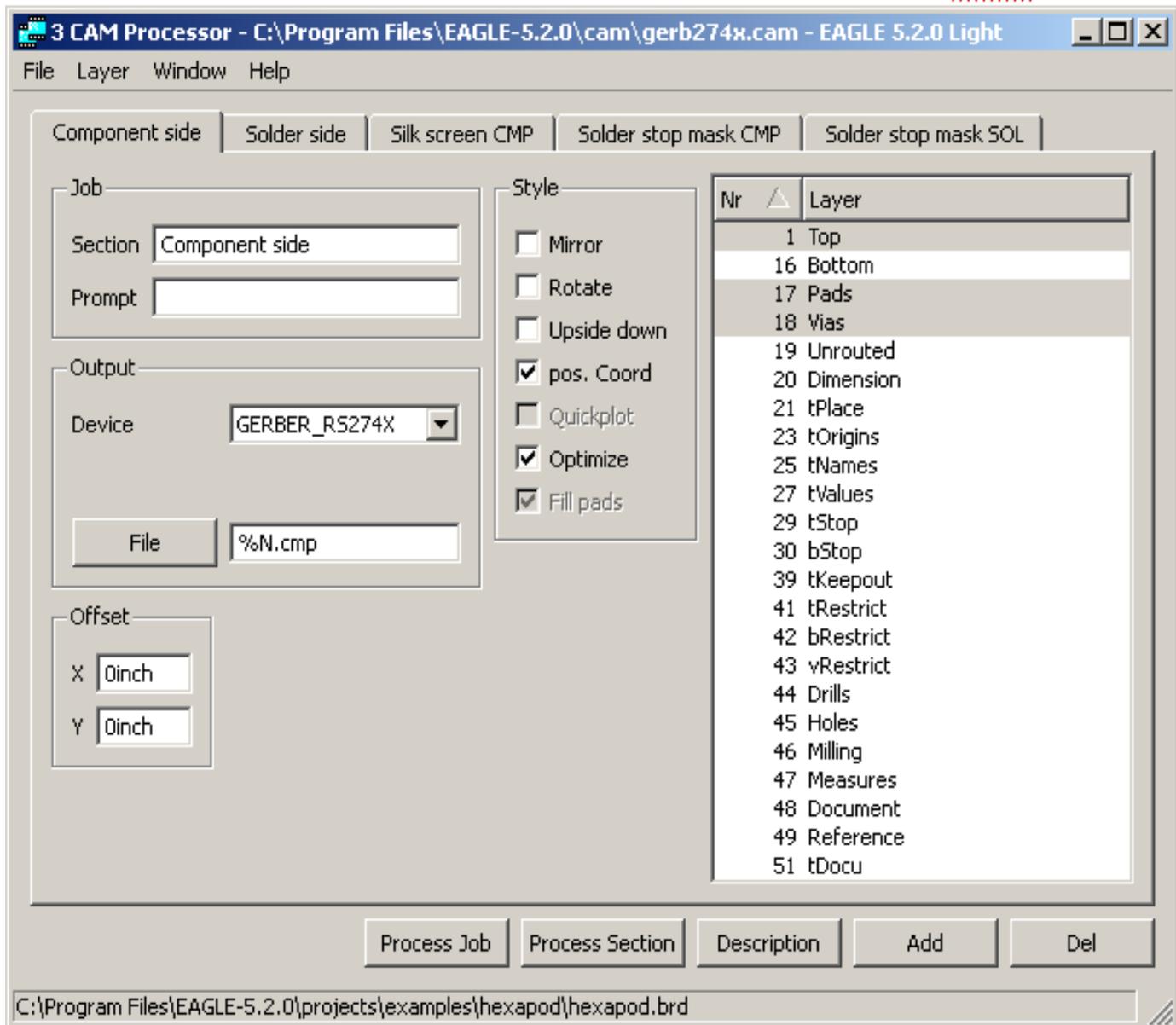 OLIMEX Ltd / Revision 1.0 / 03.09.2011

The very first thing you have to do before export files for manufacturing is to first check if they pass DRC. Download the 8mils DRU from our web and DRC check your board.

If DRC show 0 errors you can proceed to gerber and nc drill generation:



run excelon.cam load your BRD and clock Process Job. Eagle will generate *pcb-name.DRD* which contain your NC drill data

next is to generate the gerbers: run gerb274x.cam and load your BRD



and click on Process Job

Eagle will generate *pcb-name.cmp, pcb-name.plc, pcb-name.sol, pcb-name.stc, pcb-name.sts*

If you have bottom silkscreen too you have to ADD new section and to select 20 Dimension 22 bPlace and 26 bNames and make new file extension $N.pls, then Eagle will generate also *pcb-name.pls.*

Now you are ready to send your files for manufacturing, put in archive the files you generated and of course do not forget to add README.TXT as per our web template, then send them to fastpcb@olimex.com