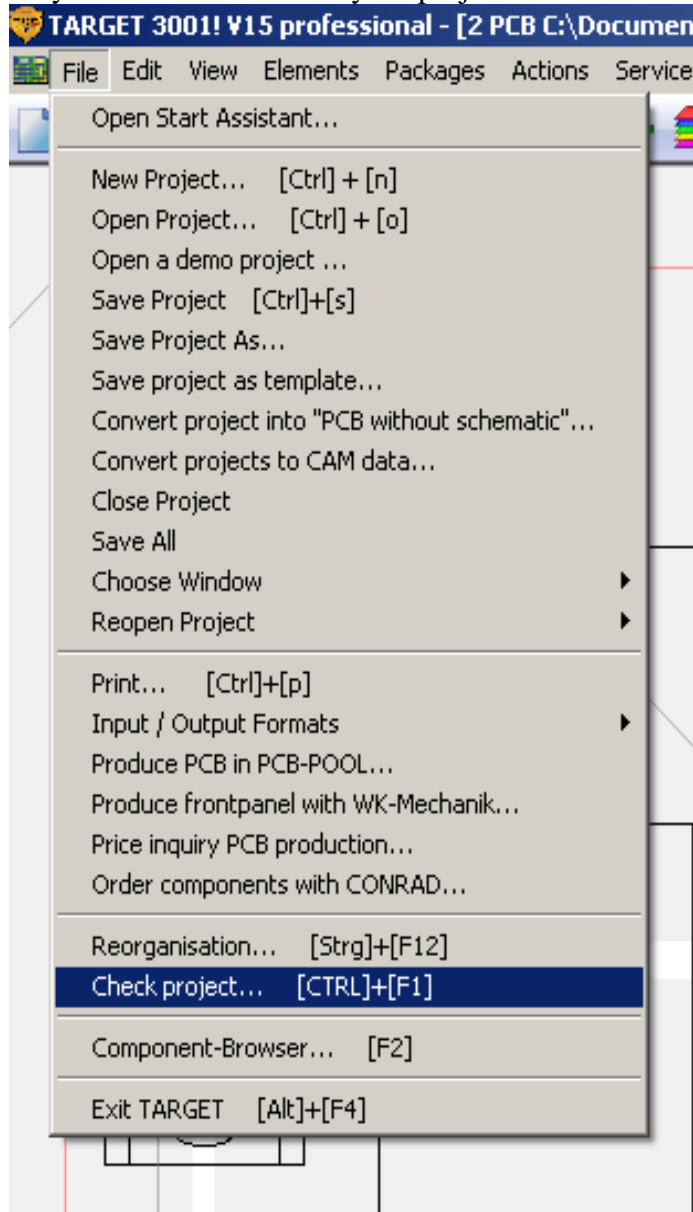


Exporting PCB production files fom Target3001

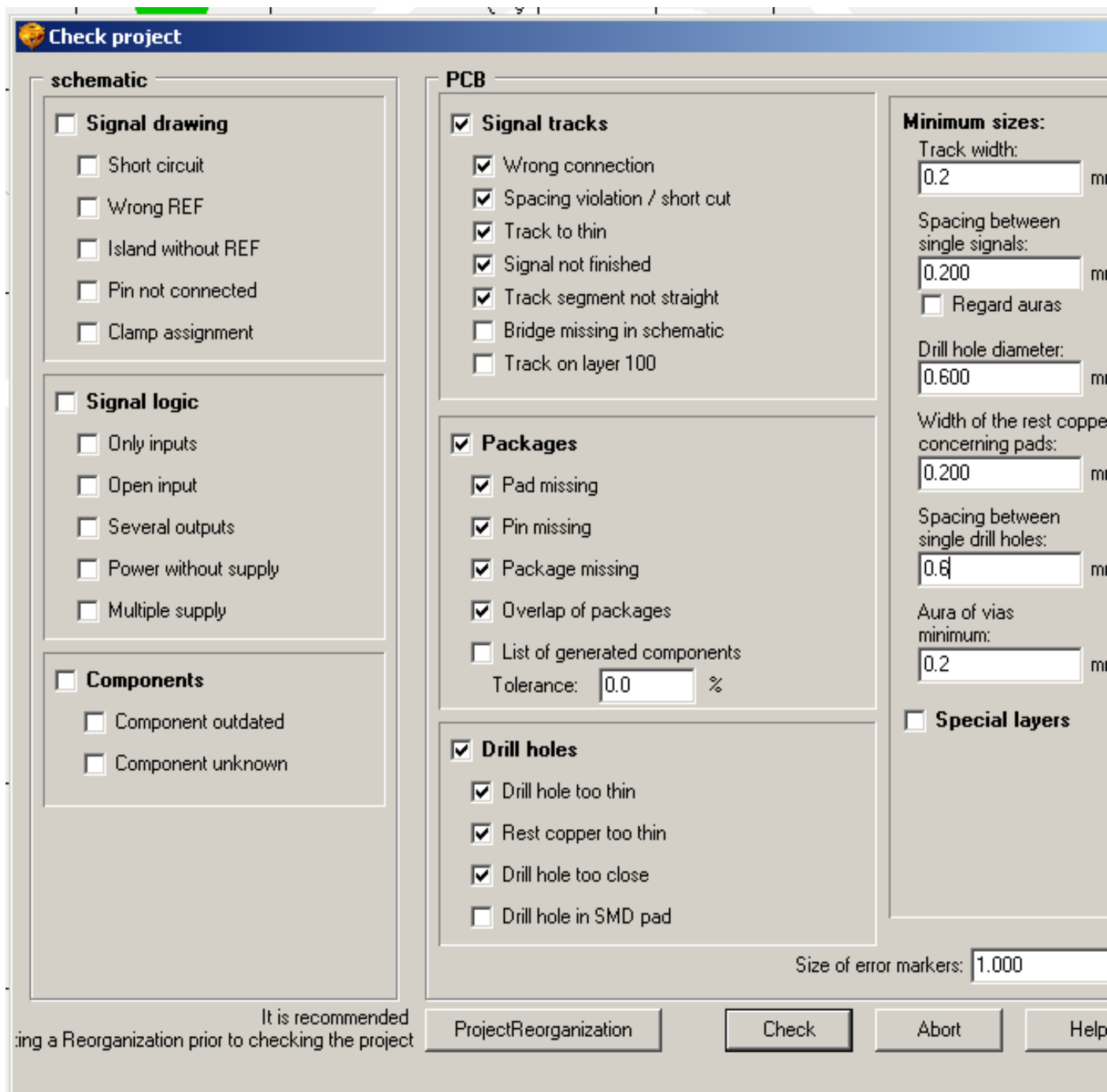
Written by Tsvetan Usunov
OLIMEX Ltd / Revision 1.0 / 03.09.2011

The very first thing you have to do before export files for manufacturing is to first check if they pass DRC.

So you have first to check your project:

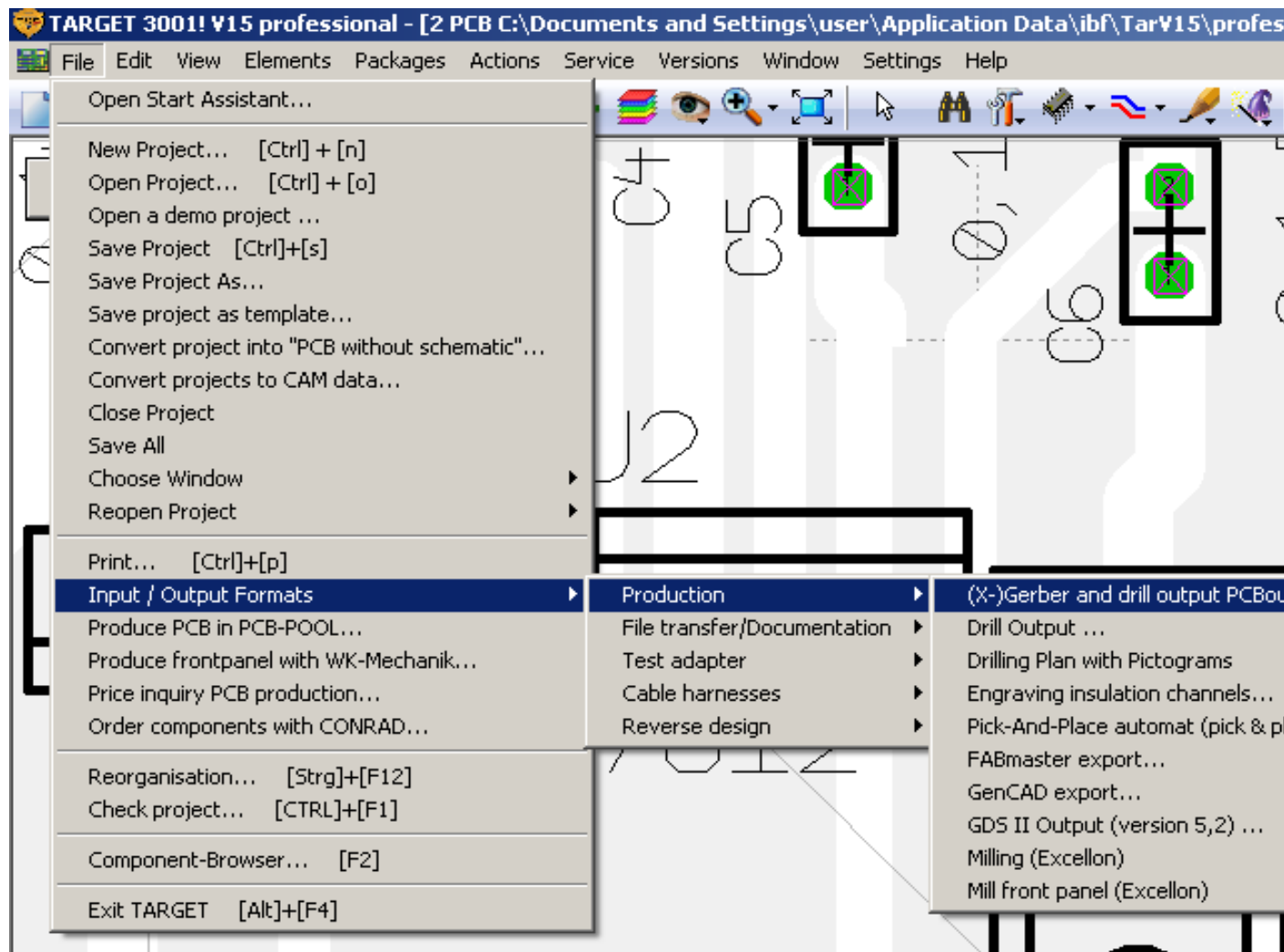


you will see this menu, set the parameters as on the picture below:

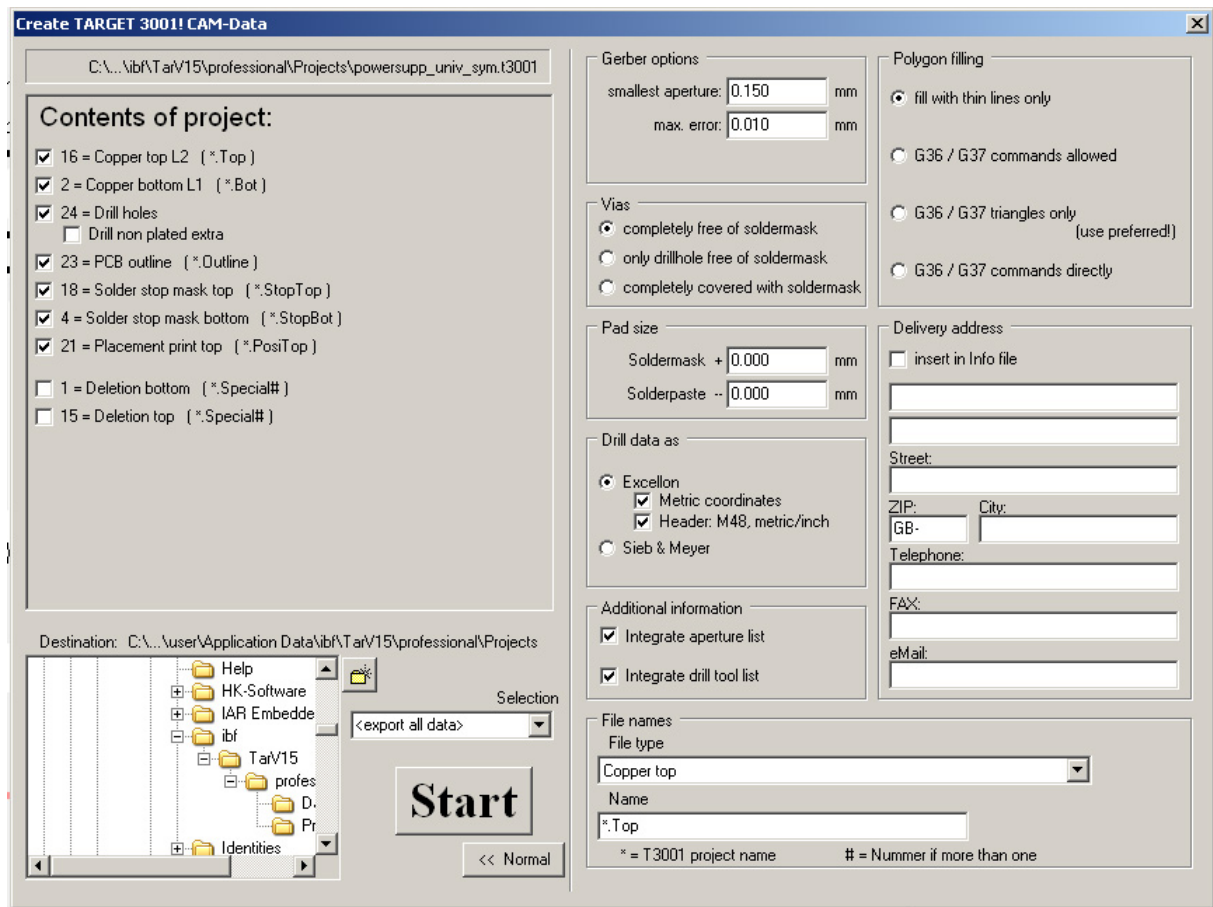


If the DRC check show no errors you can continue with the PCB manufacturing files generation.

You can do this from the File->Input/Output->Production->Gerbers:



you will see this menu, check everything as per this picture:



Now you are ready to send your files for manufacturing, put in archive the files you generated:

pcb-name.Drill, pcb-name.Top, pcb-name.Bot, pcb-name.Outline, pcb-name.PosiTop, pcb-name.StopBot, pcb-name.StopTop

if you want bottom silkscreen print you should also have *pcb-name.PosiBot*

and of course do not forget to add README.TXT as per our web template

and send them to fastpcb@olimex.com