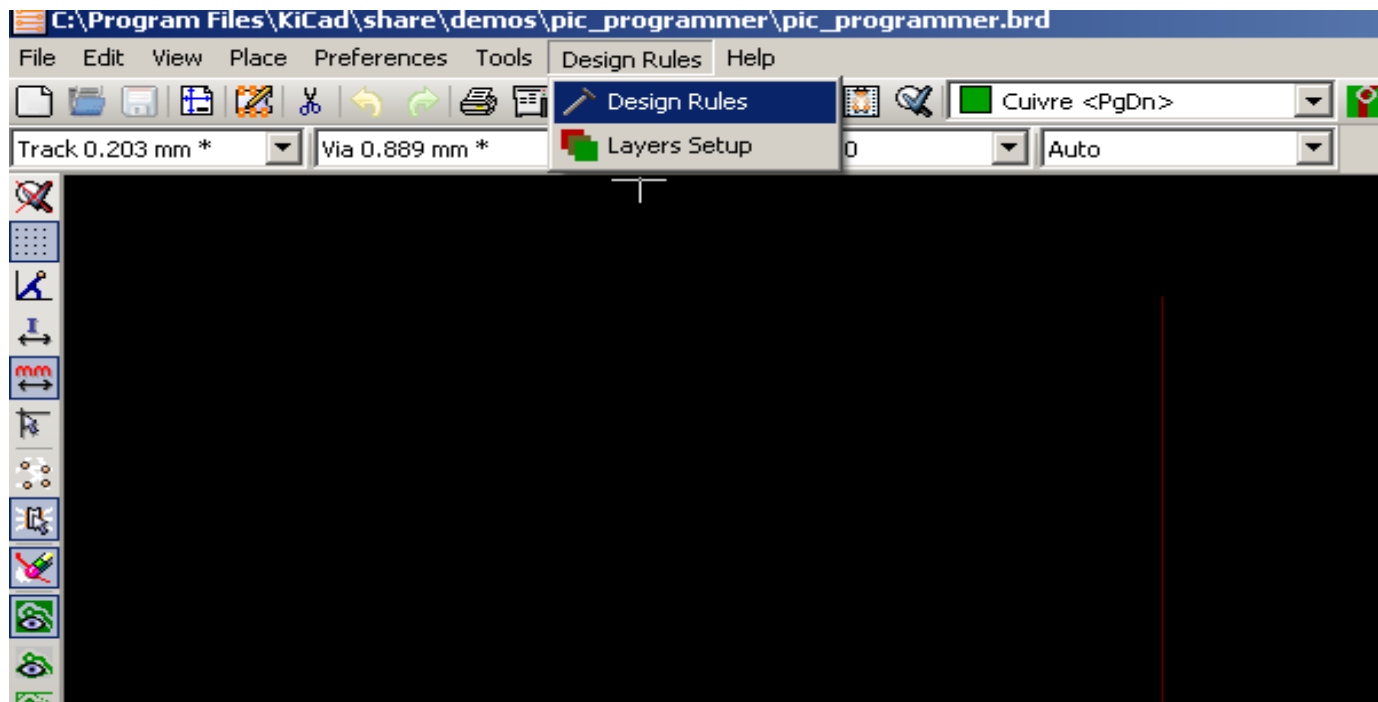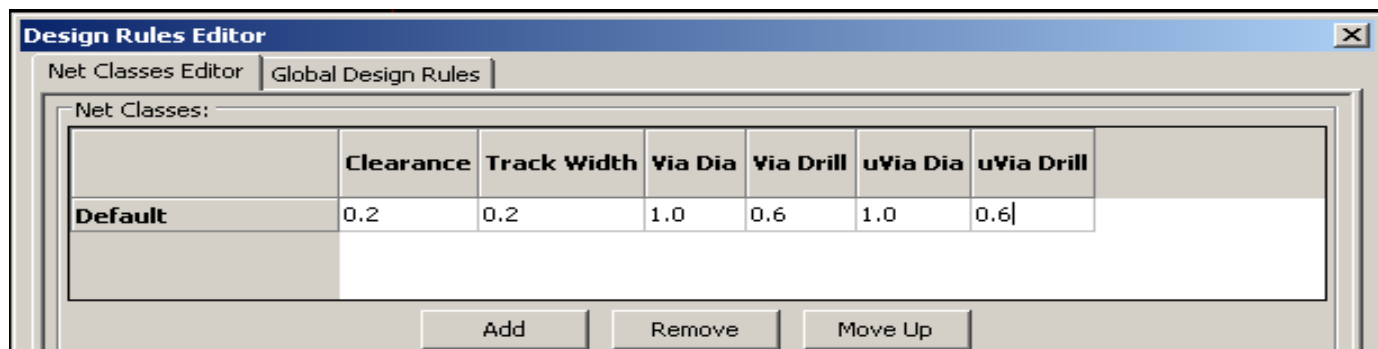# Exporiting PCB production files fom KiCAD

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.
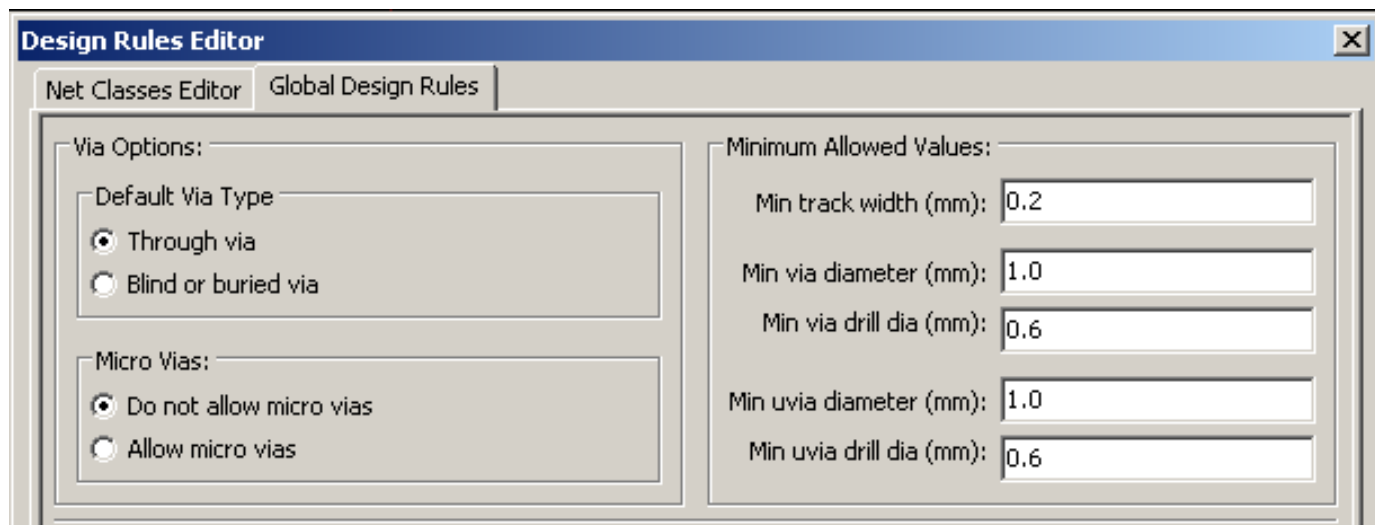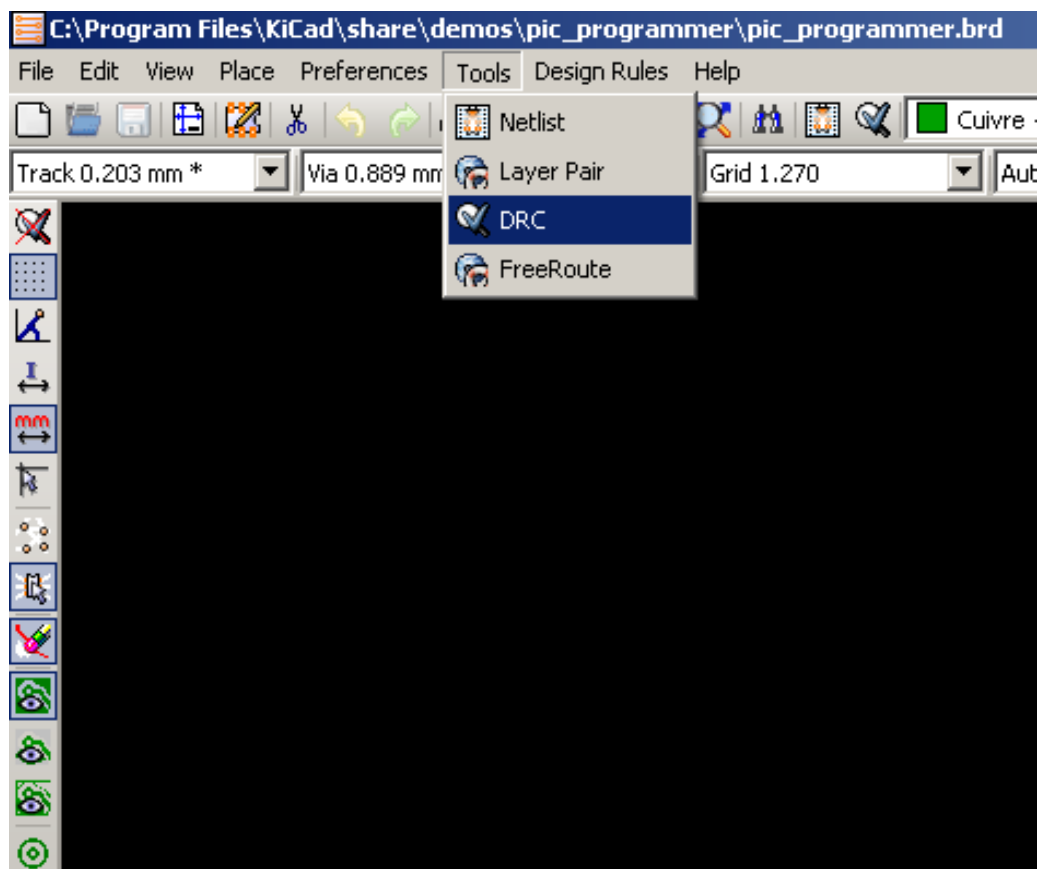First step is to set your Design Rules:

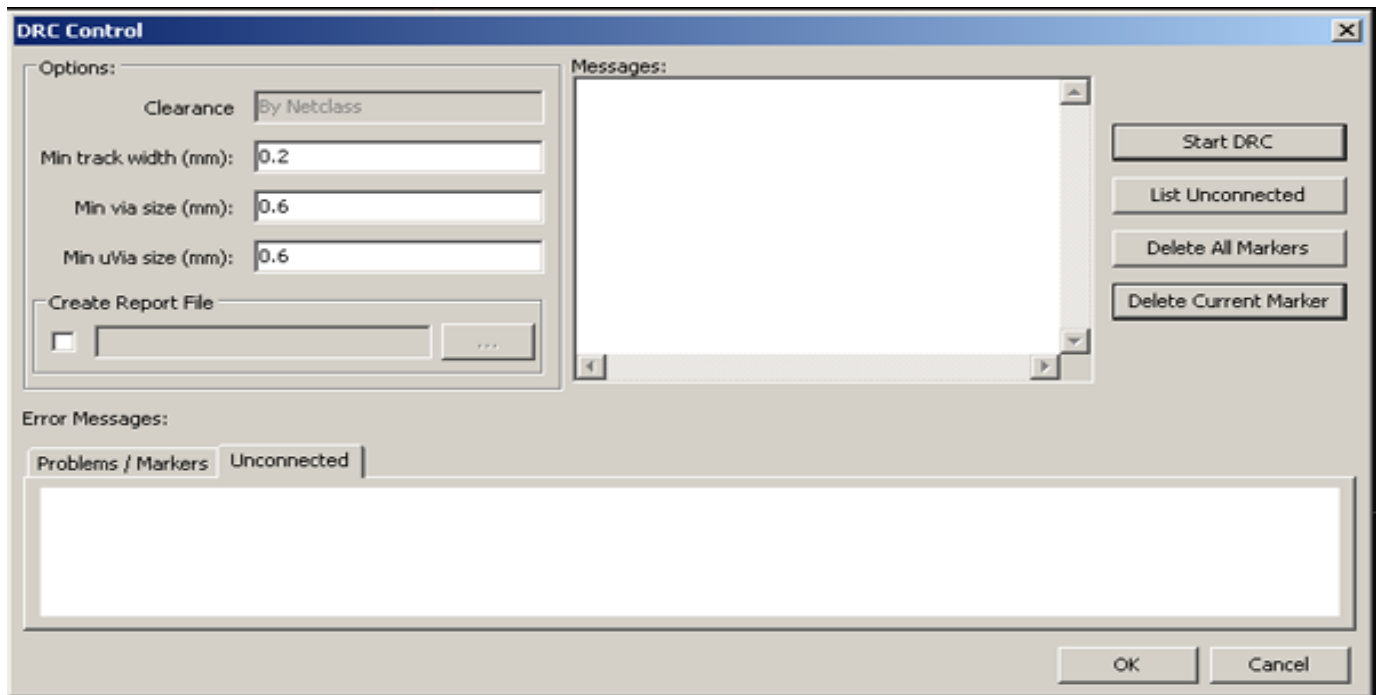you will see this menu, set the parameters as on the picture below:

the clock on Global Design Rules and set them as on the picture below:

Once you set your Design rules you can run DRC check, this is done in Board view you use the menu *Tools->DRC* as on this picture:
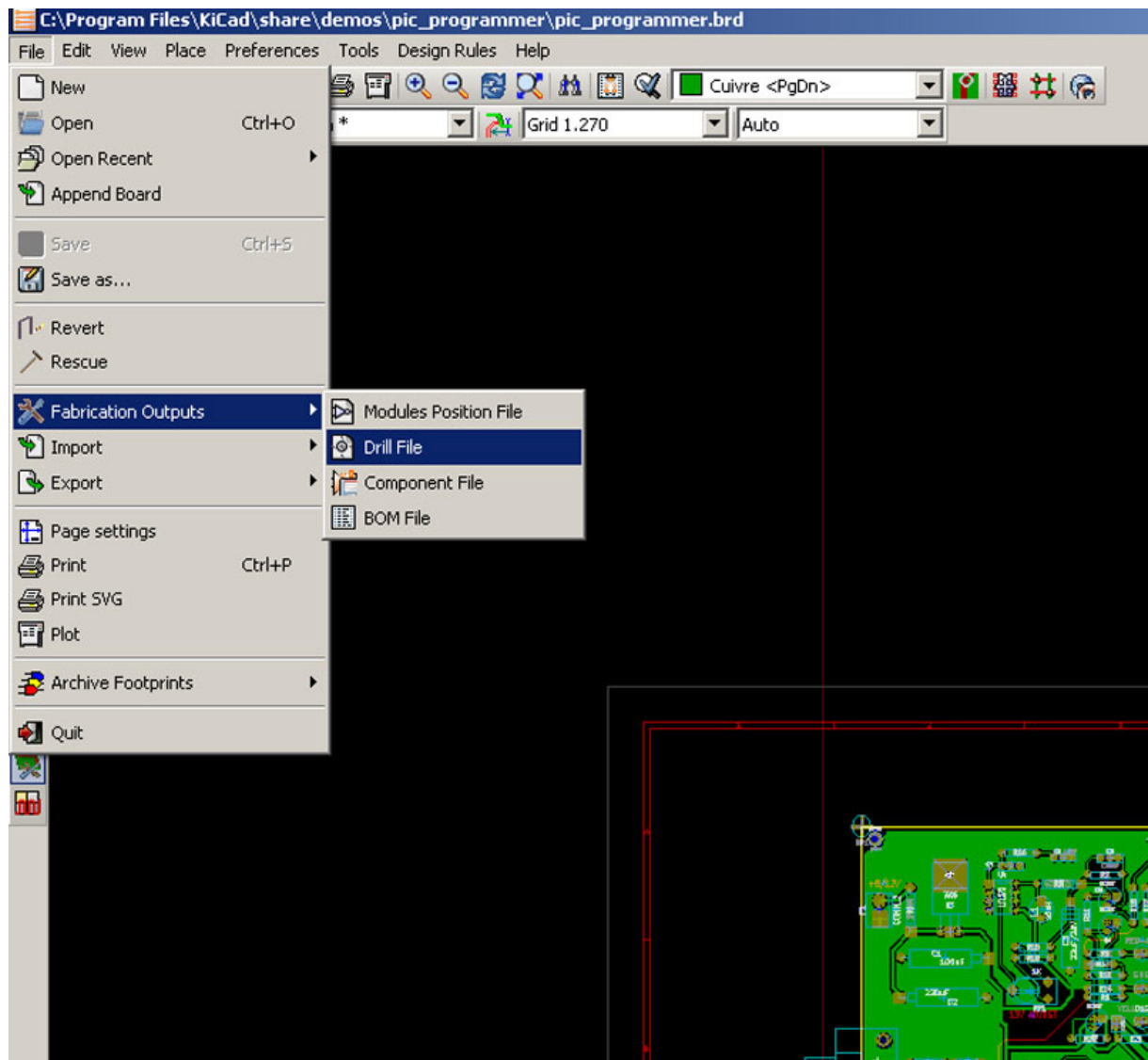
you will see this menu:

set the minimum track to 0.2 mm, minimum via to 0.6 mm and minimum uVia to 0.6 mm, then StartDRC, if errors show please fix them, until no error is displayed.

Please do not send Board files which contain DRC errors, we will not accept for manufacturing boards with DRC violations and when you send boards with errors this will delay your order and rise your price.
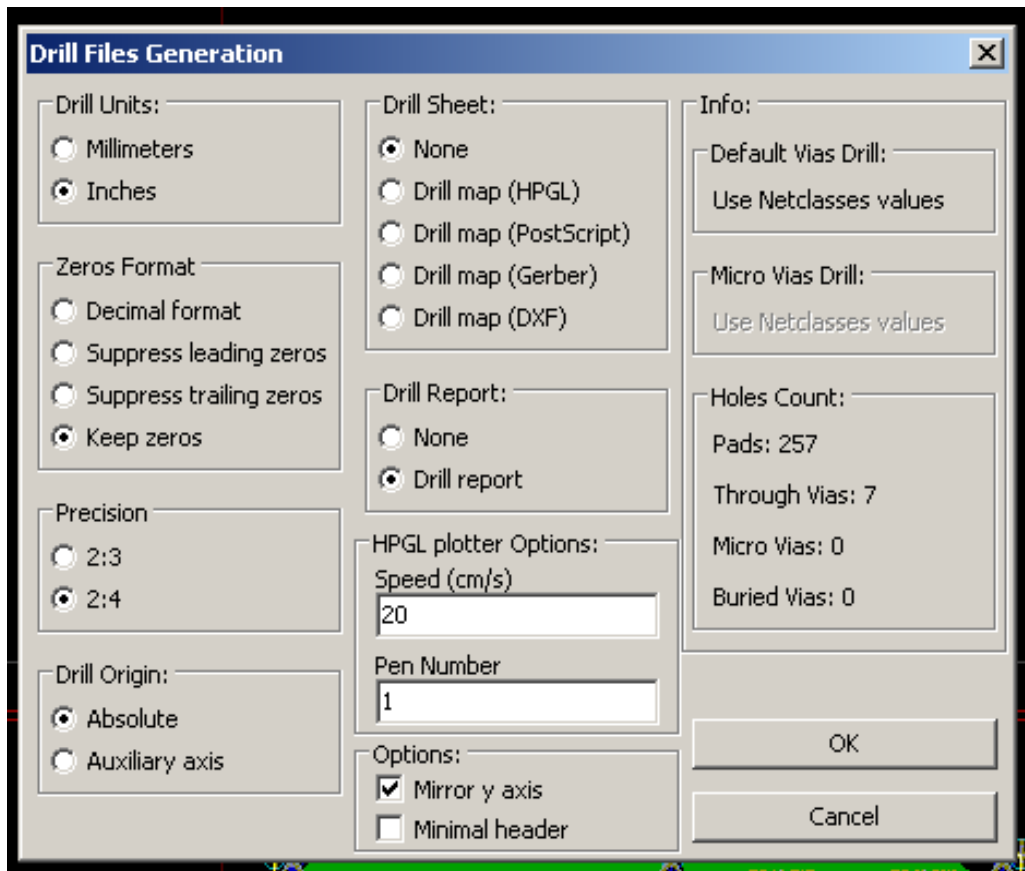
Once your board is DRC checked and OK we can proceed with PCB production files generation.

First step is to generating the DRILL files:

Open your project, then switch to Board view and run *File->Fabrication output->Drill File:*
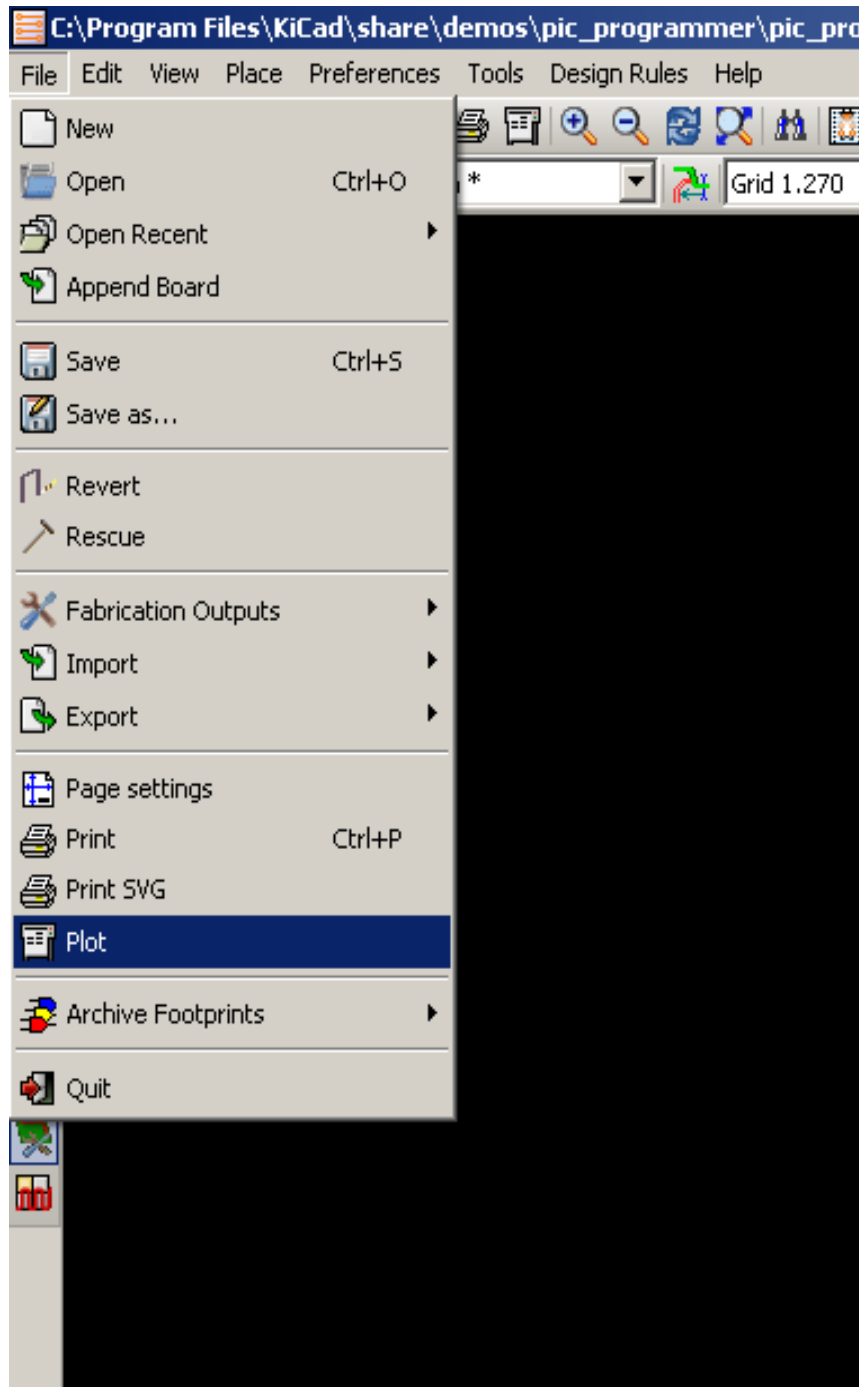
you will see this menu:

for Drill Units you should select <u>Inches or Millimeters</u> depend on what you have used when *designed* your board, i.e. if you have designed your libraries and boards in Inches export in Inches, if you have used Millimeters export in Millimeters, this is important because ensures that you do not lost precision due to conversion rounding.

All other parameters should be as on the picture, hit OK and it will generate your drill files: *pcb-name.DRL* and *pcb-name.RPT* we need both of these to be provided for manufacturing

So far so good, next step is to generate your gerbers, in File menu select Plot as on this picture:

you will see this menu, check the boxes exactly as per this picture:
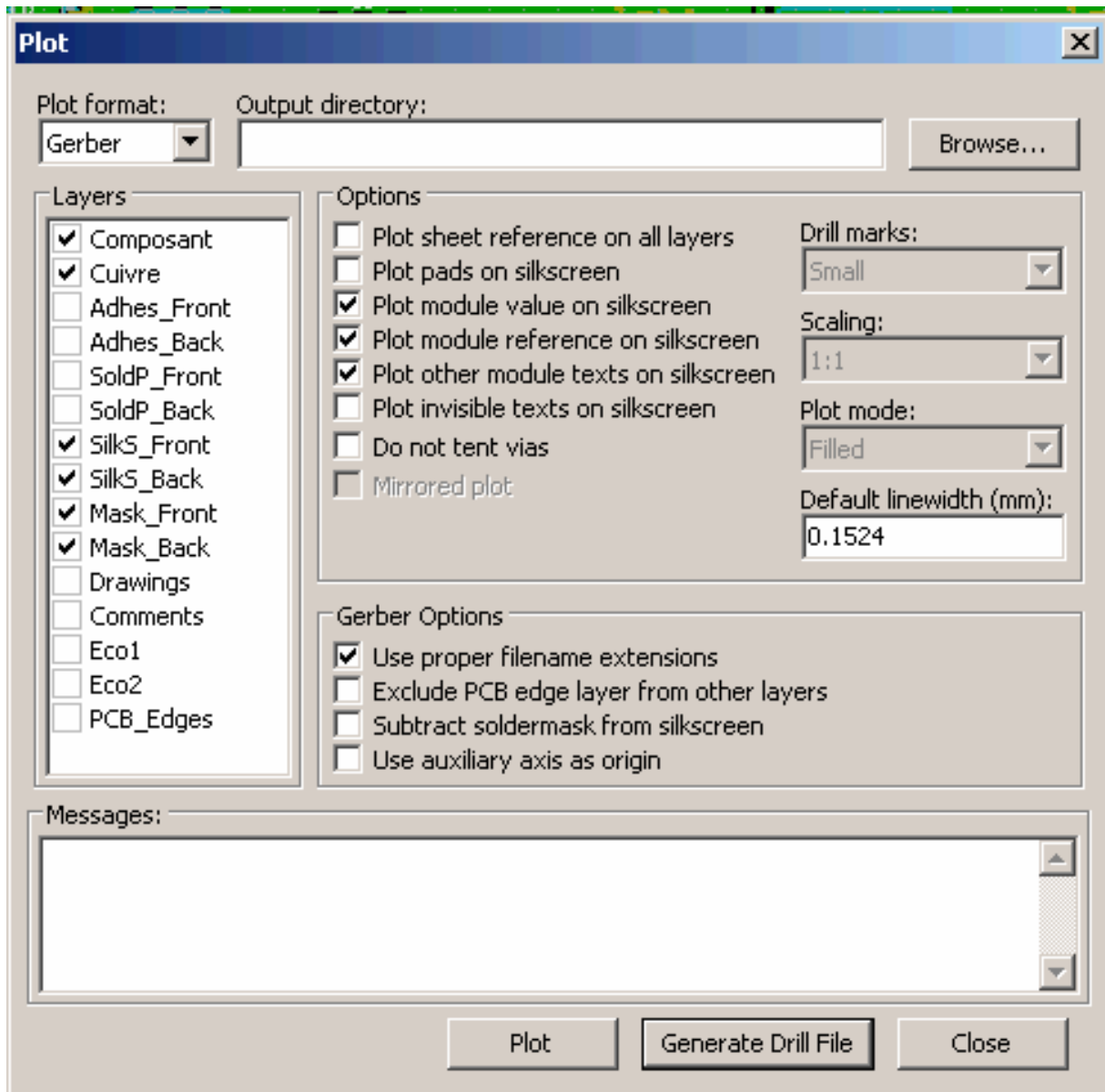
then hit Plot button, you will generate these files:

pcb-name-Composant.gtl, pcb-name-Cuivre.gbl, pcb-name-SilkS_Back.gbo, pcb-name-SilkS_Front.gto, pcb-name-Mask_Back.gbp, pcb-name-Mask_Front.gtp.

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

pcb-name.DRL , pcb-name.RPT, pcb-name-Composant.GTL, pcb-name-Cuivre.GBL, pcb-name-SilkS_Front.GTO, pcb-name-SoldP_Back.GBP, pcb-name-SoldP_Front.GTP
and of course README.TXT as per our web template

and send them to fastpcb@olimex.com

these 8 files are enough to produce your board, if you want it with silkscreen on bottom side too you can add in archive *pcb-name-SilkS_Back.GBO*.