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:

Control Panel - EAGLE 5.2.0 Light File View Options Window Help			
File View Options Window Name Image: Campoint of the second	Help Description CAM Processor Jobs Example for cam2printer Generates EPS Format Generates Gerber Format Generates Extended Ge Generates Extended Ge Generates Excellon Drill Script Files User Language Programs Design Rules Libraries	Generates Extended Gerber Format This CAM job consists of five sections that genera You will get five gerber files that contain data for: component side *.cmp solder side *.sol silkscreen component side *.plc solder stop component side *.stc solder stop solder sid *.sts	

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

🔚 3 CAM Processor - C:\Program Files\EAGLE-5.2.0\cam\gerb2	74x.cam - EAGLE 5.2.0 Light		
File Layer Window Help			
Component side Solder side Silk screen CMP Solder stop m		1	
Section Component side Prompt Output Device GERBER_RS274X Gereination Gereinatio Gereinati	NrLayer1Top16Bottom17Pads18Vias19Unrouted20Dimension21tPlace23tOrigins25tNames27tValues29tStop30bStop39tKeepout41tRestrict42bRestrict43vRestrict44Drills45Holes46Milling47Measures48Document49Reference51tDocu		
Process Job Process Section	Description Add	Del	
C:\Program Files\EAGLE-5.2.0\projects\examples\hexapod\hexapod.brd			

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