How to create a gerber file - Eagle

SATLAND Prototype
ul. Sarnia 5
80-336 Gdańsk
tel. +48 58 5540764
www.prototypy.com.pl

1. Download the "Gerbery2010.cam" file from our website
2. Open file in the Eagle program
3. Onto the board view
4. Choose:
   - „Tools”
     - „Ratsnest”
5. Choose:
   - „File”
     - „CAM Procesor”
6. Choose from the CAM Processor window menu
   - „File”
     - “Open”
       - “Job”
7. Choose the downloaded "Gerbery2010.cam" file and choose:
   - „Open”
8. Then click "Process Job" below.
9. Close Eagle program.
10. The gerber file has been exported to the catalogue in which we will find the PCB project (files: b.gbl, t.gtl, d.nc, m.mil, ts.gts, bs.gbs, opis.gbl, opis.gtl).