

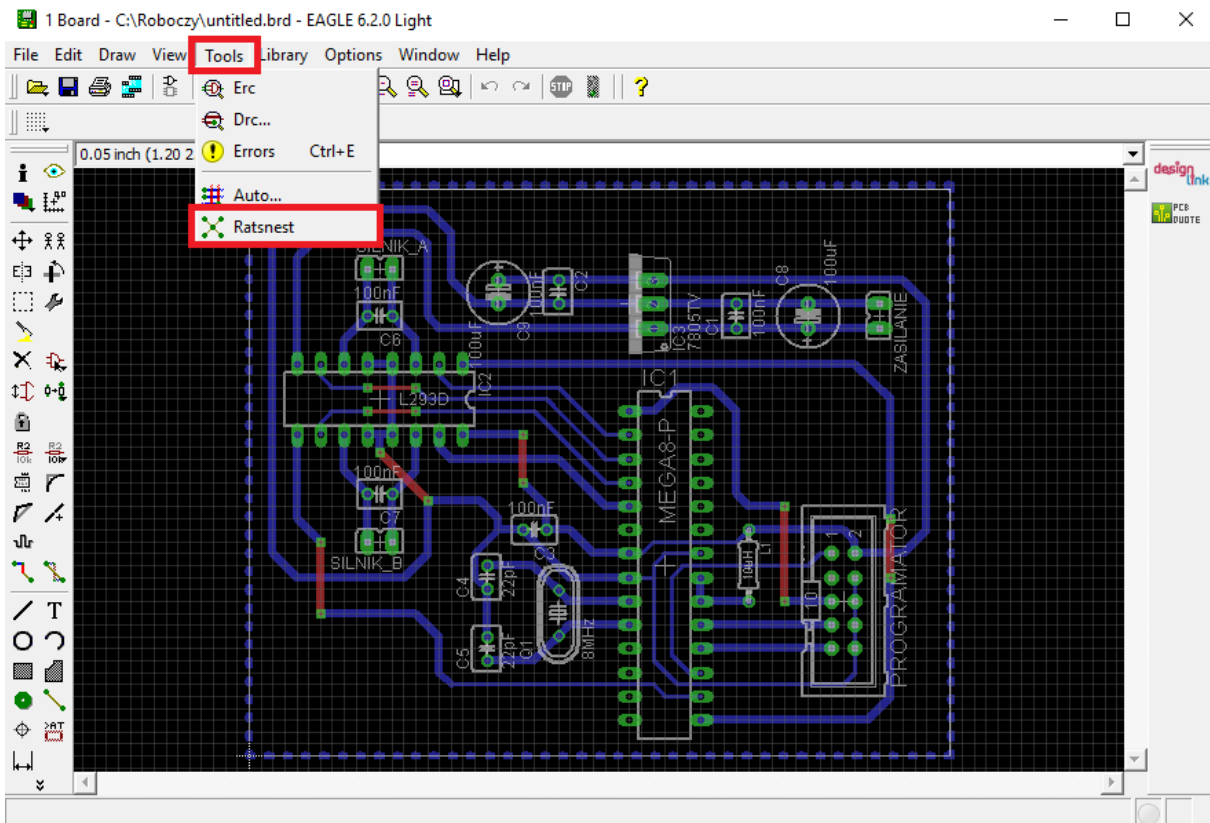
## Exporting Gerber files from the Eagle CAD software (up to version 9.1)

This manual describes how to generate Gerber files for one-, two- and four-layer PCBs.

1. Download the gerber\_TSPCB.cam configuration file from our website ([link](#)).

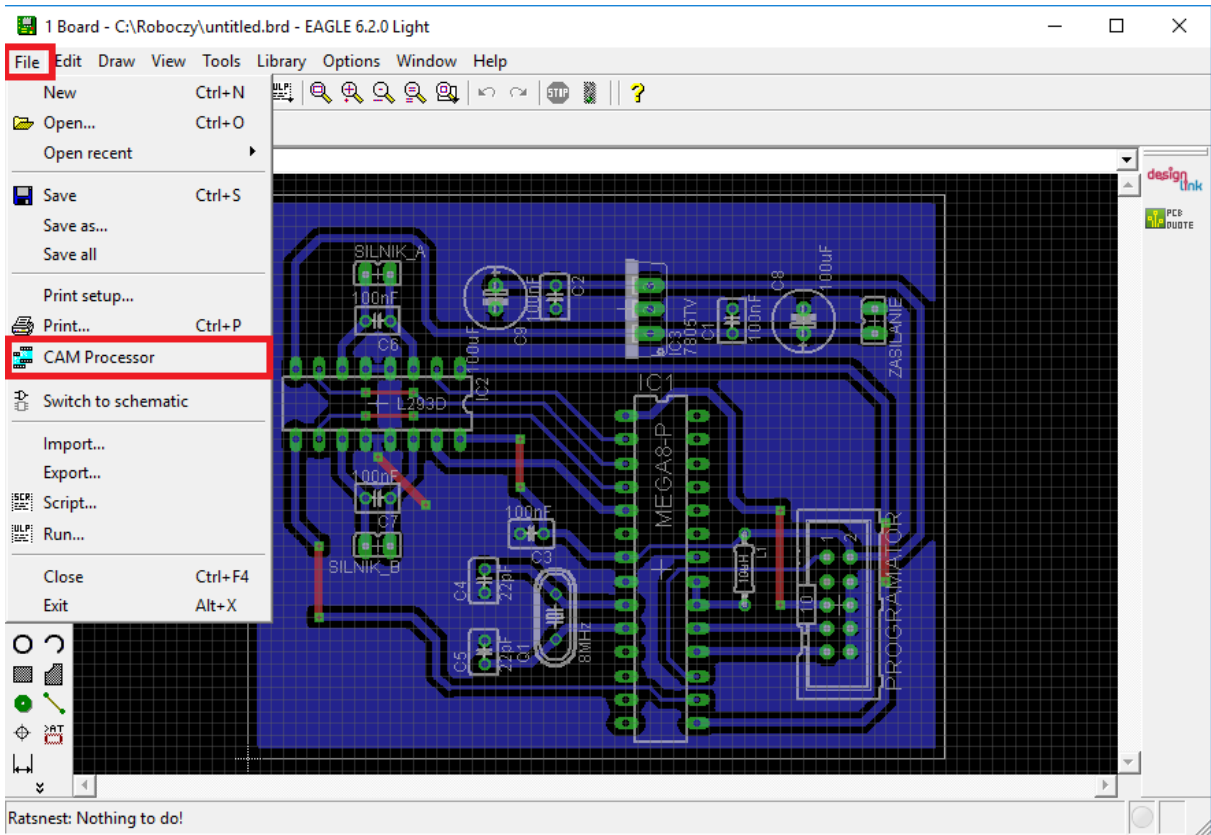
**ATTENTION! Please check whether the configuration file includes all layers to be made (especially the elements on the silkscreen layers).**

2. Open project in the Eagle program.
3. Open window with a view of the circuit.
4. Generate polygons for copper planes (*Tools -> Ratsnest*).

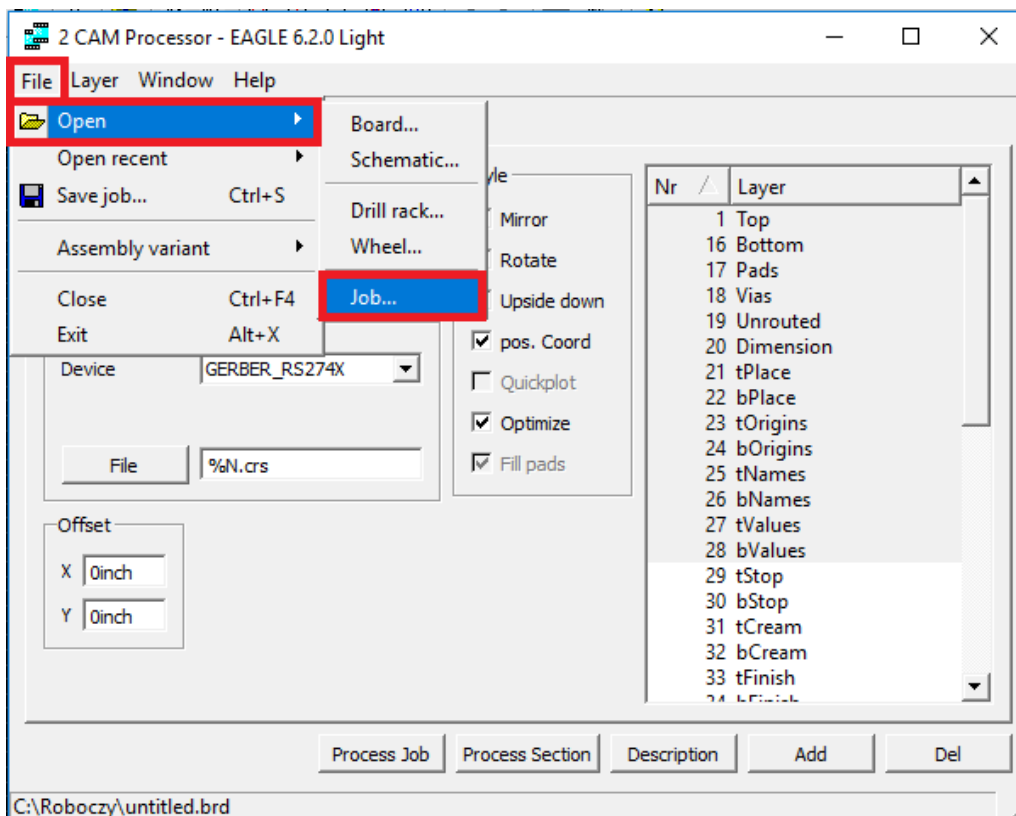


Benzynowa 21  
83-011 Gdańsk  
phone: (+48) 58 340 42 54  
mail: [office@tspcb.pl](mailto:office@tspcb.pl)

- 5. Generate Gerber files:
  - a. select *File -> CAM Processor* from the window with a circuit view,

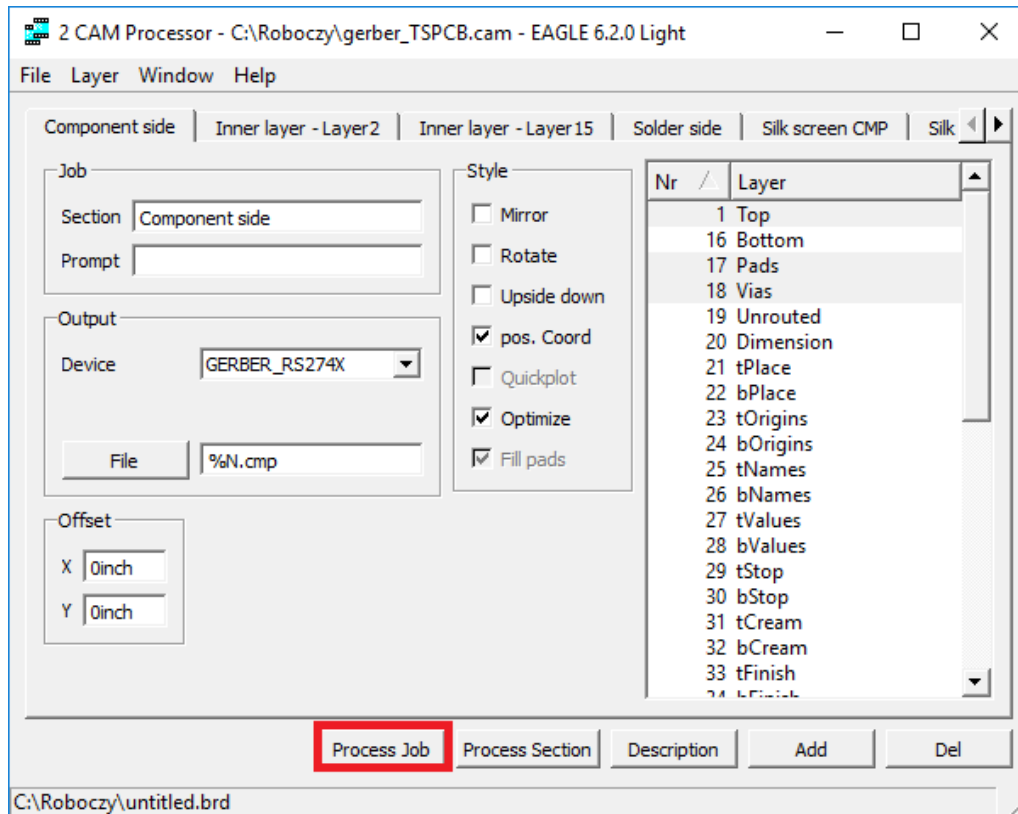


- b. in the *CAM Processor* window select *File -> Open -> Job...*



Benzynowa 21  
83-011 Gdańsk  
phone: (+48) 58 340 42 54  
mail: [office@tspcb.pl](mailto:office@tspcb.pl)

- c. in the folder selection window search for previously downloaded *gerber\_TSPCB.cam* file and then click *Open* button,
- d. in the *CAM Processor* window click *Process Job* button.



6. Close Eagle program. The generated Gerber files were saved in the same directory as the project. The names of generated files, their function and layer numbers from Eagle program are as follows:
  - \*.cmp – top copper – layer numbers: 1, 17, 18,
  - \*.ly2 – internal 1 copper (internal top copper) – layer numbers: 2, 17, 18,
  - \*.l15 – internal 2 copper (internal bottom copper) – layer numbers: 15, 17, 18,
  - \*.sol – bottom copper – layer numbers: 16, 17, 18,
  - \*.stc – top soldermask – layer numbers: 29,
  - \*.sts – bottom soldermask – layer numbers: 30,
  - \*.plc – top silkscreen – layer numbers: 21, 25,
  - \*.pls – bottom silkscreen – layer numbers: 22, 26,
  - \*.crc – top paste – layer numbers: 31,
  - \*.crs – bottom paste – layer numbers: 32,
  - \*.gko – mechanical treatment (outline + cutouts) – layer numbers: 20, 46,
  - \*.drd – drills – layer numbers: 44, 45,
  - \*.dri – information file,
  - \*.gpi – information file.

**For two-layer PCBs \*.ly2 and \*.l15 files are useless and can be deleted before sending.**

All files have be packed (into .zip, .rar, .7z archive) and together with completed technological card ([link](#)) sent by mail or uploaded via our website ([link](#)).