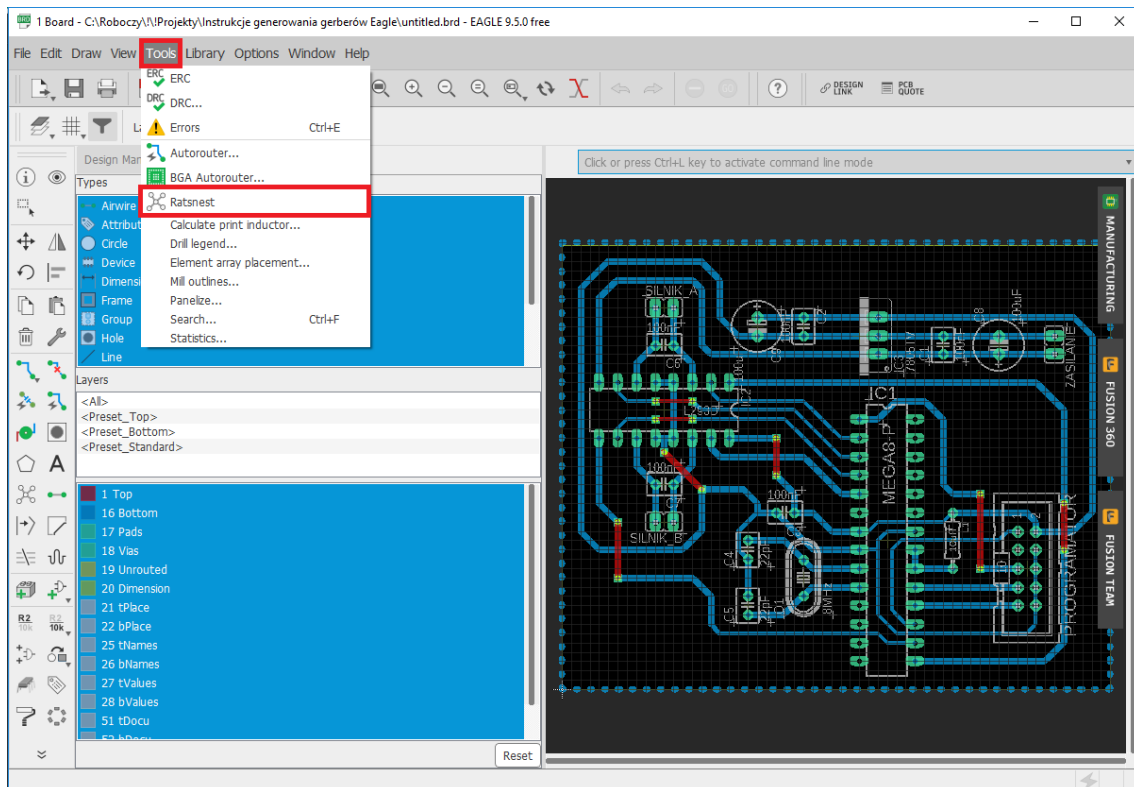


## Exporting Gerber files from the Eagle CAD software (version 9.2 and higher)

**This manual describes how to generate Gerber files with default Eagle configuration. Attention, please modify configuration when nondefault layer settings have been used!**

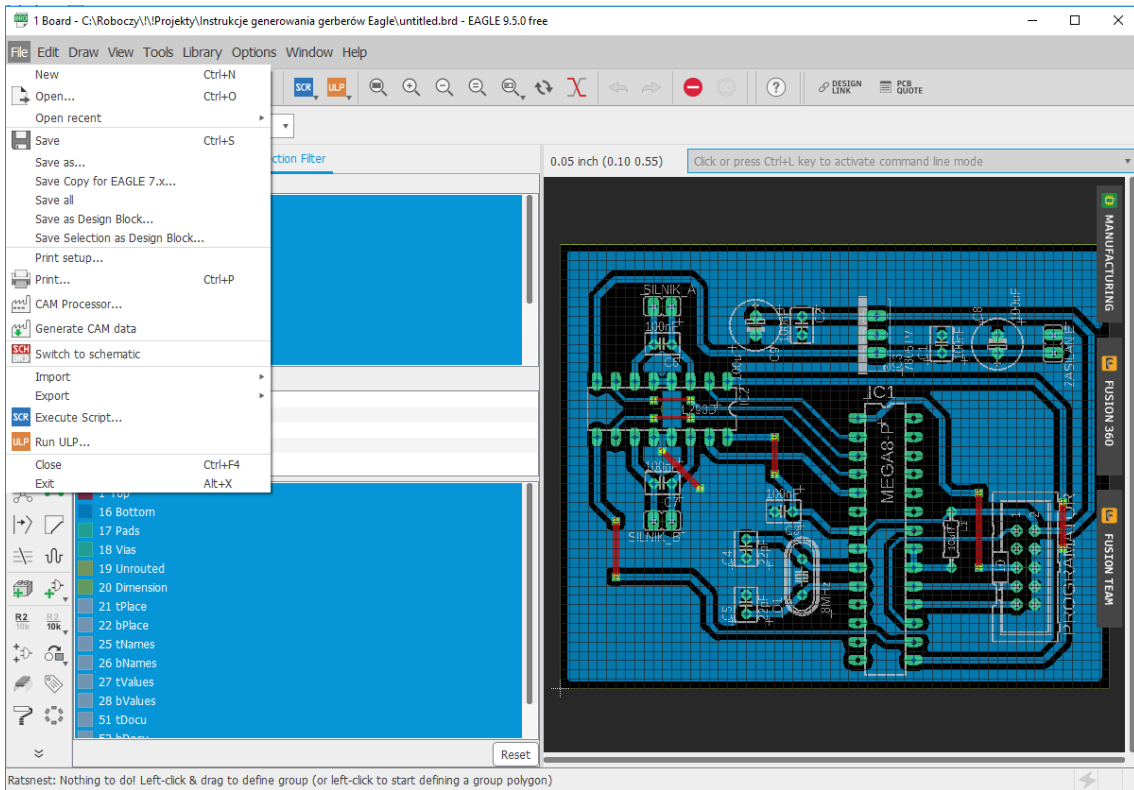
1. Open project in the Eagle program.
2. Open window with a view of the circuit.
3. Generate polygons for copper planes (*Tools -> Ratsnest*).



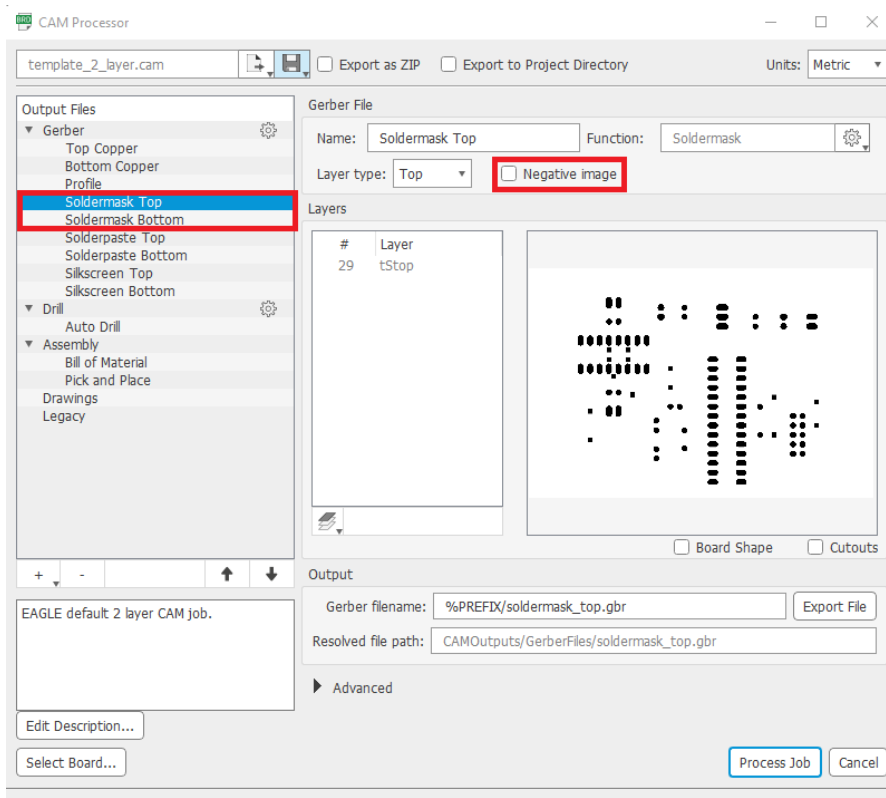
Benzynowa 21  
 83-011 Gdańsk  
 phone: (+48) 58 340 42 54  
 mail: [office@tspcb.pl](mailto:office@tspcb.pl)

4. Generate Gerber files:

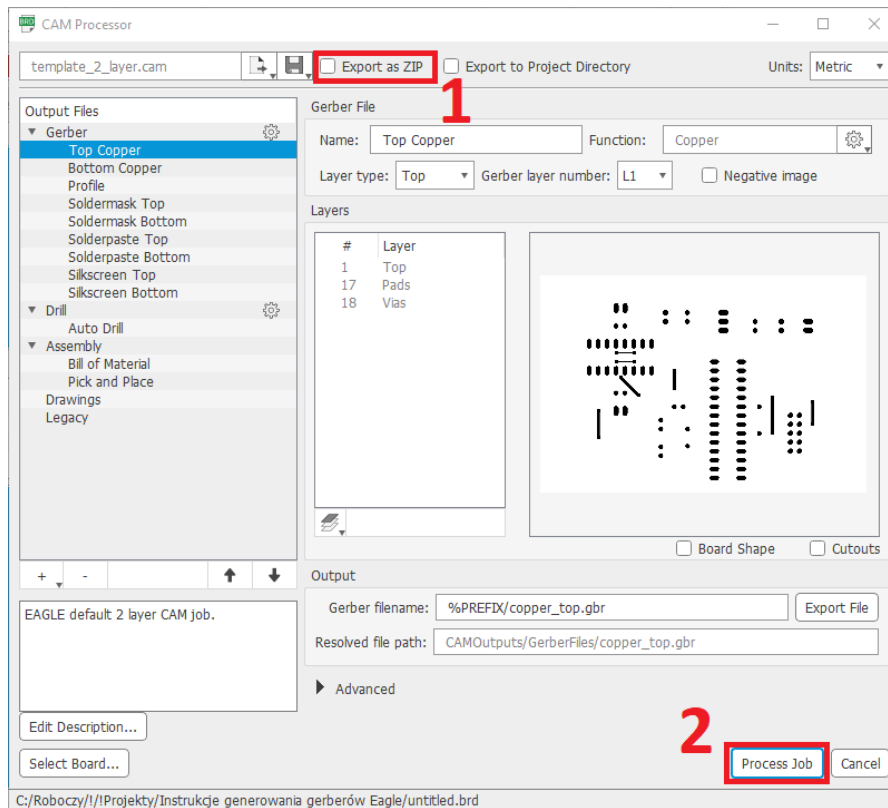
- a. select *File -> CAM Processor* from the window with a circuit view,



- b. in the *CAM Processor* window select *Soldermask Top* and *Soldermask Bottom* and check if *Negative image* is unchecked,



- c. in the CAM Processor window check *Export as ZIP* (1) and then click *Process Job* button (2). In the folder selection window choose directory for and then click *Choose Folder* button,



5. Close Eagle program. Generated Gerber files were saved in the directory chosen in point 4a. The names of generated files, their function and layer numbers (for 4-layer PCB) from Eagle program are as follows:

- copper\_top\_l1.gbr – top copper – layer numbers: 1, 17, 18,
- copper\_top\_l2.gbr – internal 1 copper (internal top copper) – layer numbers: 2, 17, 18,
- copper\_top\_l3.gbr – internal 2 copper (internal bottom copper) – layer numbers: 15, 17, 18,
- copper\_top\_l4.gbr – bottom copper – layer numbers: 16, 17, 18,
- soldermask\_top.gbr – top soldermask – layer numbers: 29,
- soldermask\_bottom.gbr – bottom soldermask – layer numbers: 30,
- silkscreen\_top.gbr – top silkscreen – layer numbers: 21, 25,
- silkscreen\_bottom.gbr – bottom – layer numbers: 22, 26,
- solderpaste\_top.gbr – top paste – layer numbers: 31,
- solderpaste\_bottom.gbr – bottom paste – layer numbers: 32,
- profile.gbr – mechanical treatment (outline + cutouts) – Board Shape and Cutouts layers,
- drill\_1\_16.xln – drills – layer numbers: 44, 45,
- gerber\_job – information file.

**Files copper\_top\_IX.gbr (where X is a consecutive number for copper layer 1,2, etc.) are generated depending on the number of copper layers in the project. For a 2-layer PCB we get copper\_top\_l1.gbr and copper\_top\_l2.gbr files, for a 4-layer PCB we get copper\_top\_l1.gbr to copper\_top\_l4.gbr files respectively.**

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