



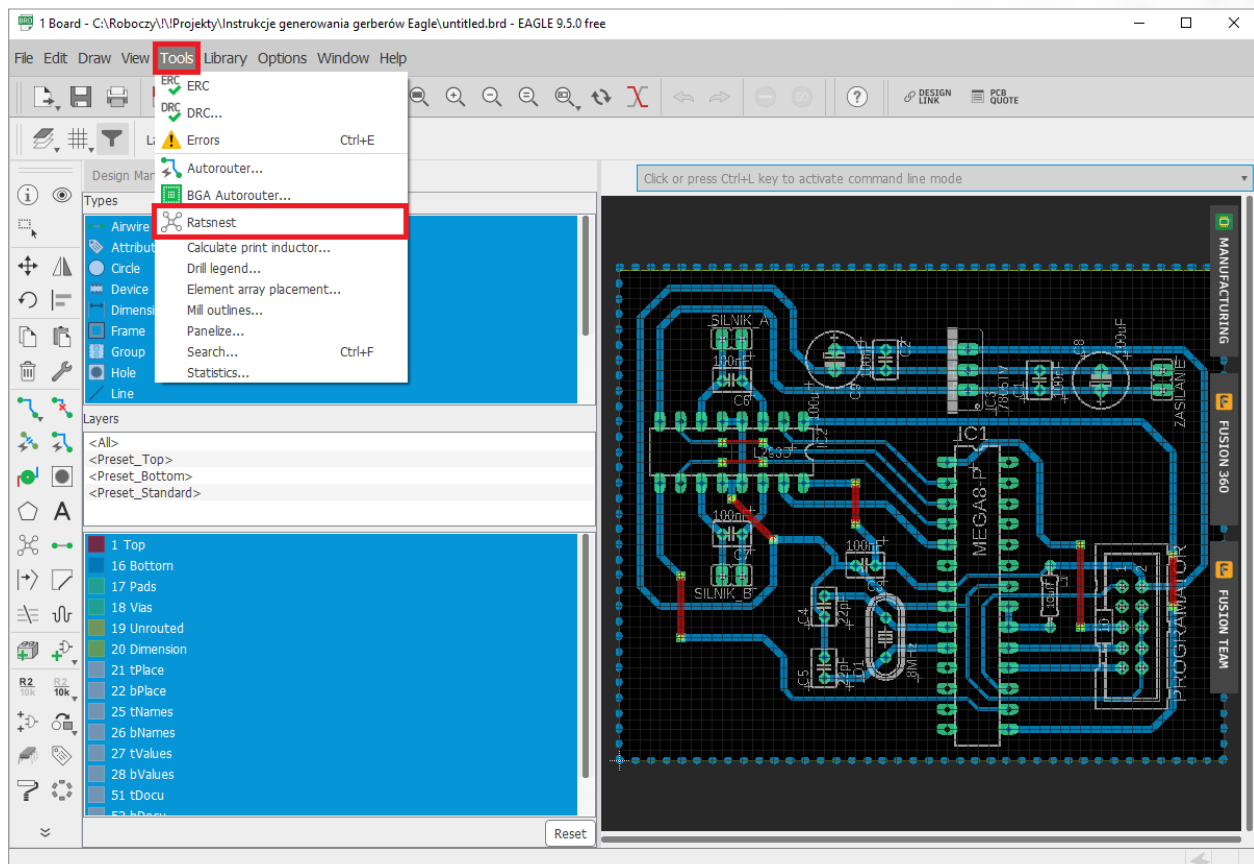
Exporting Gerber files from the Eagle CAD software

(version 9.2 and higher)



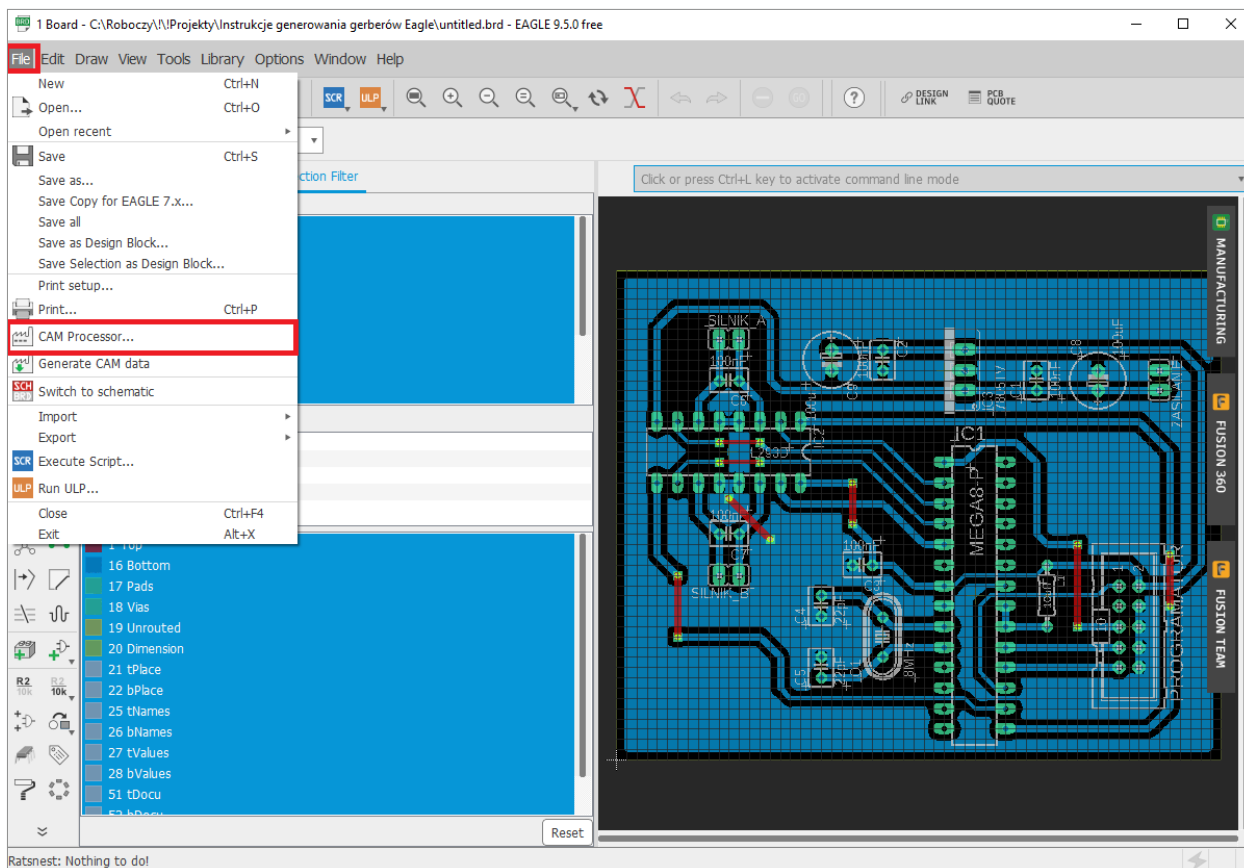
This manual describes how to generate Gerber files with default Eagle configuration. Attention, please modify configuration when nondefault layer settings have been used!

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

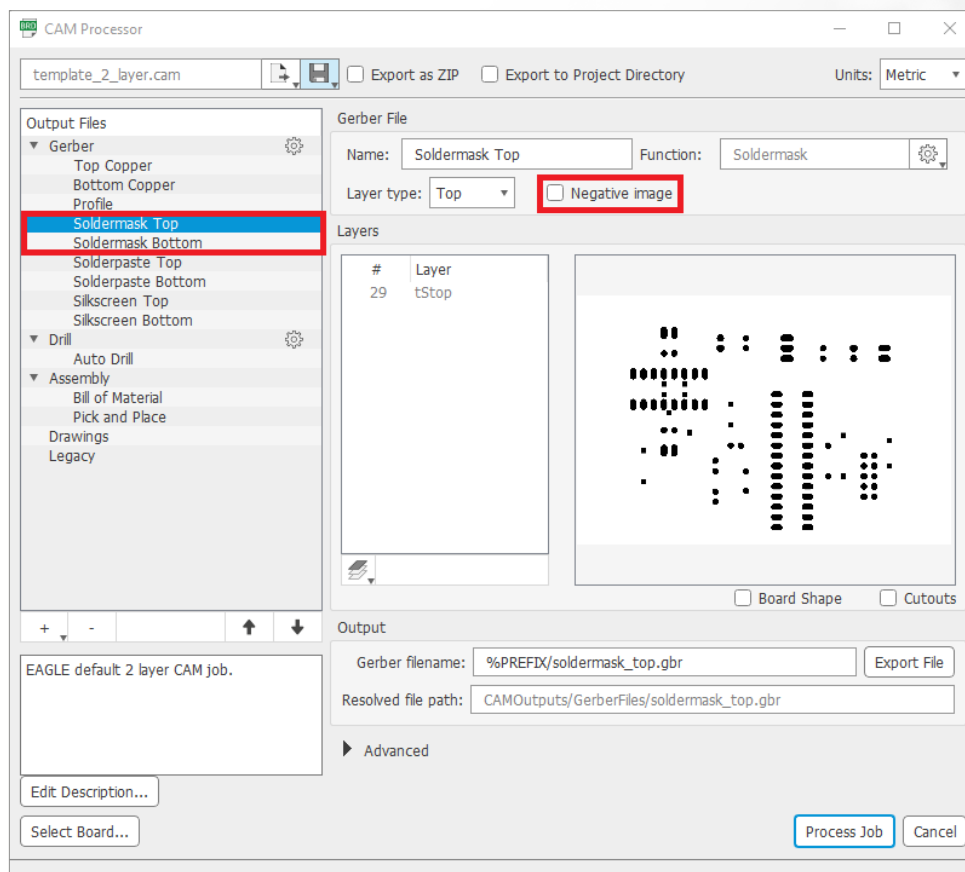


4. Generate Gerber files:

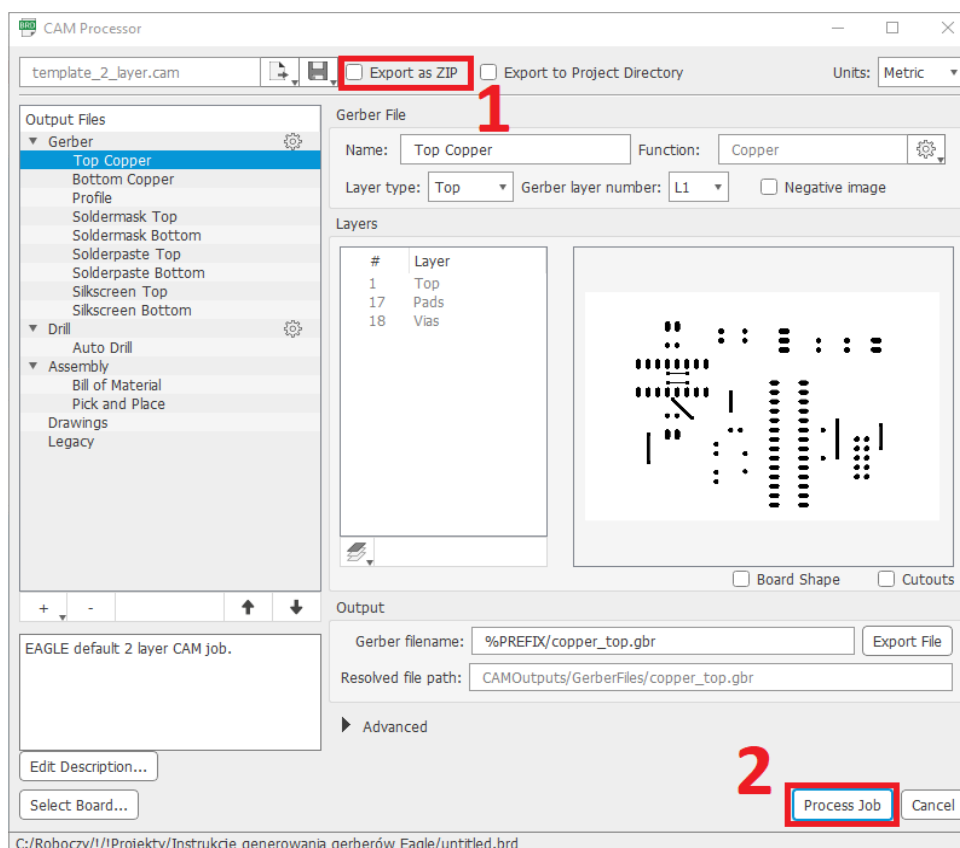
- a. select **File -> CAM Processor** from the window with a circuit view,



- 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_lX.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](#))