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,

🕮 1 Board - C:\Roboczy\!\!Projekty	\Instrukcje generowania ge	rów Eagle\untitled.brd - EAGLE 9.5.0 free			- 0	×
File Edit Draw View Tools Lib	orary Options Window	Help				
New Open	Ctrl+N Ctrl+O	् .		P DESIGN E PCB QUOTE		
→ Open Open recent → Save as Save copy for EAGLE 7.x Save as Design Block Save see Copy for EAGLE 7.x Save as Design Block Save see Copy for EAGLE 7.x Save as Design Block Print. → Print → Print	Ctrl+P Ctrl+F4 Alt+X		r press Ctrl+L key to activate comme	and Ine mode		MANUFACTURING FUSION 360 FUSION TEAM
28 bValues 51 tDocu		Reset				
Ratsnest: Nothing to do!		(Keser)			4	

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

CAM Processor		- 0	>		
template_2_layer.cam	Þ.	Export as ZIP Export to Project Directory Units: Metric			
Output Files		Gerber File			
Gerber	53		5		
Top Copper	640	Name: Soldermask Top Function: Soldermask 성공	\$₽		
Bottom Copper					
Profile		Layer type: Top 🔻 🗌 Negative image			
Soldermask Top		Lavora			
Soldermask Bottom		Layers			
Solderpaste Top		# lavar			
Solderpaste Bottom					
Silkscreen Top		29 USLOP			
Silkscreen Bottom					
▼ Drill	ξ ³ ρ				
Auto Drill					
 Assembly 					
Bill of Material		nonining - 2 2			
Pick and Place					
Drawings					
Legacy					
		Board Shape	out		
+	† +	Output			
EAGLE dofult 2 byor CAM job		Gerber filename: %PREFIX/soldermask_top.gbr Export F			
CAULE VERVELZ RAVEL CAM TOD.					
ENGLE DEIBUIL Z IBYEL CAM JOD.			lie		
EAGLE DETAUL 2 layer CAM JOD.		Resolved file path: CAMOutputs/GerberFiles/soldermask_top.gbr	-lie		
ENGLE Derault 2 layer CAM JOD.		Resolved file path: CAMOutputs/GerberFiles/soldermask_top.gbr Advanced	-lie		
Edit Description		Resolved file path: CAMOutputs/GerberFiles/soldermask_top.gbr Advanced	lie		
Edit Description		Resolved file path: CAMOutputs/GerberFiles/soldermask_top.gbr Advanced	lie		

2

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,

CAM Processor	- O X
template_2_layer.cam	Export as ZIP Export to Project Directory Units: Metric •
Output Files	Gerber File
▼ Gerber ② Top Copper Bottom Copper	Name: Top Copper Function: Copper Image: Copper Laver type: Top Top Gerber laver number: L1 Negative image
Profile Soldermask Top Soldermask Bottom Solderpaste Top Solderpaste Bottom Silkscreen Top Silkscreen Bottom ▼ Drill Auto Drill ♥ Assembly Bill of Material Pick and Place	Layers # Layer 1 Top 17 Pads 18 Vias
Legacy	Board Shape Cutouts
+ ↓ - ↑ ↓	Output
EAGLE default 2 layer CAM job.	Gerber filename: %PREFIX/copper_top.gbr Export File Resolved file path: CAMOutputs/GerberFiles/copper_top.gbr Advanced
Edit Description Select Board C:/Roboczy/1/IProjekty/Instrukcie generowag	2 Process Job Cancel

- **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**)