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 (<u>link</u>).

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*).



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...

2 CAM Processor - EAGLE 6.2	0 Light		_		
File Layer Window Help					
Open > Open recent > Save job Ctrl+S Assembly variant > Close Ctrl+F4 Exit Alt+X Device GERBER_RS27 File %N.crs Offset X Y Oinch	Board Schematic Drill rack Wheel	vle Mirror Rotate Upside down I♥ pos. Coord I♥ Quidxplot I♥ Optimize I♥ Fill pads	Nr Layer 1 Top 16 Bottom 17 Pads 18 Vias 19 Unrouted 20 Dimension 21 tPlace 23 tOrigins 24 bOrigins 25 tNames 26 bNames 27 tValues 28 bValues 29 tStop 30 bStop 31 tCream 33 tFinish	×	
	Process Job	Process Section	Description Add	Del	
2:\Roboczy\untitled.brd					

- **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.

🚰 2 CAM Processor - C:\Roboczy\gerber_TSPCB.cam - EAGLE 6.2.0 Light — 🛛 🛛 🗙					
File Layer Window Help					
Component side Inner layer - Layer2 Inner layer - Layer15 Job Section Component side Style Prompt Inner layer - Layer2 Inner layer - Layer15 Output Inner layer - Layer2 Inner layer - Layer15 Output Inner layer - Layer2 Inner layer - Layer15 Output Inner layer - Layer2 Inner layer - Layer15 Output Inner layer - Layer2 Inner layer - Layer15 Prompt Inner layer - Layer2 Inner layer - Layer15 Output Inner layer - Layer2 Inner layer - Layer15 Prompt Inner layer - Layer2 Inner layer - Layer15 Output Inner layer - Layer2 Inner layer - Layer15 Prompt Inner layer - Layer2 Inner layer - Layer15 Output Inner layer - Layer2 Inner layer - Layer15 Prompt Inner layer - Layer2 Inner layer - Layer15 Output Inner layer3 Inner layer3 File %N.cmp Inner layer3 Offset X Inner layer3 X Inner layer3 Inner layer3 Offset X Inner laye	Solder side Silk screen CMP Silk Nr Layer 1 Top 16 Bottom 17 Pads 18 Vias 19 Unrouted 20 Dimension 21 tPlace 23 tOrigins 24 bOrigins 25 tNames 26 bNames 27 tValues 28 bValues 29 tStop 30 bStop 31 tCream 32 bCream 33 tEinish				
Process Job Process Section	Description Add Del				
:\Roboczy\untitled.brd					

- **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,
 - *.115 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**).