

Eagle:

We recommend exporting the data to the Gerber RS-274X and keeping the same name of layers that can be learned for future pcb.

CAM procesor – GATEMA

Number layer	Layers
1, 17, 18	TOP
16, 17, 18	BOTTOM
29	solder mask Top
30	solder mask Bottom
21, 25	silkscreen Top
22, 26	silkscreen Bottom
20	dimension / routing
44, 45	drilling
46	routing
31	solder paste Top
32	solder paste Bottom

Note:

If the signal layers are without text, we assume that the component layer (TOP) is readable imported!

It is necessary to indicate the version Eagle (e.g. 6.5.0)!

Use our
[CAM procesor](#)
for optimal layers labelling and export.