How to Generate Gerber Files in EAGLE

*This document uses Windows 10 and EAGLE 9.2.2 free version.

1.) Download the CAM job file E301_Gerber_XXX.cam from iLearn under EAGLE_misc. "xxx" substitutes for the date that the .cam file was modified.

2.) In EAGLE, switch to the board layout window. Click File > CAM Processor. The CAM Processor window will come up.
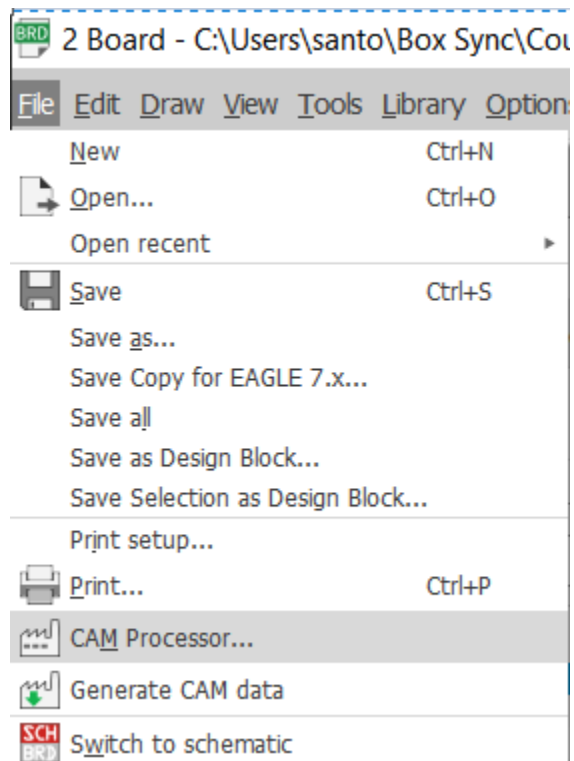

Figure 1

3.) On the top left corner, select Load job file > Open CAM file... > browse and select the CAM job file downloaded earlier > Open. On Silkscreen Top under Gerber, make sure that the Layer # 27 tValues is included. This is not included by default.
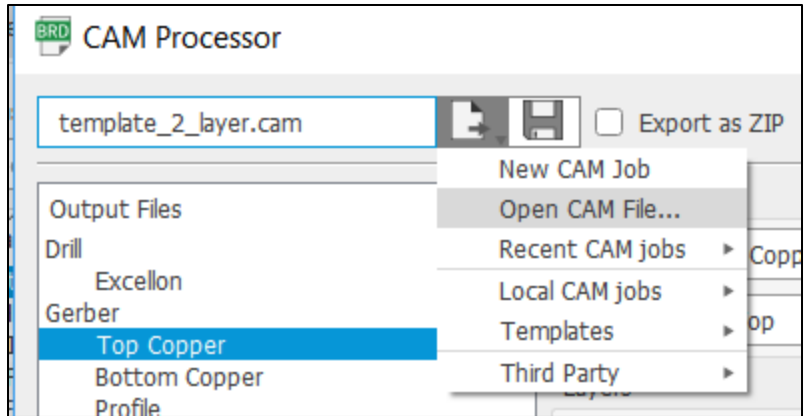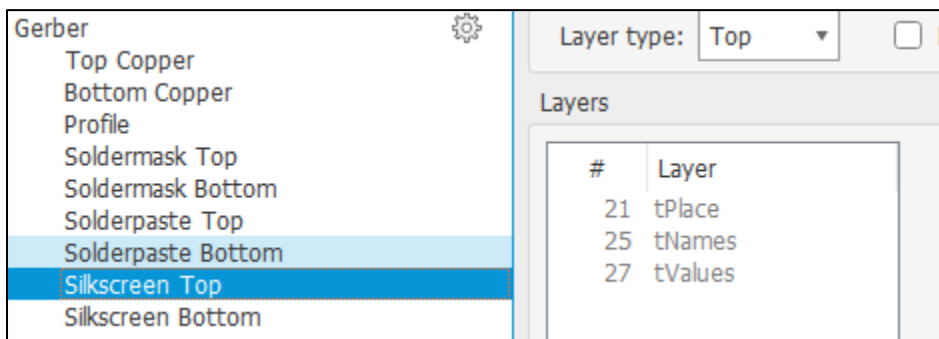
Figure 2

Figure 3

4.) Check "Export as ZIP" > Process Job. This will generate the .zip file in the same folder as your board file. You will turn this in along with the schematic (.sch) and the board (.brd) files, see Figure 6.
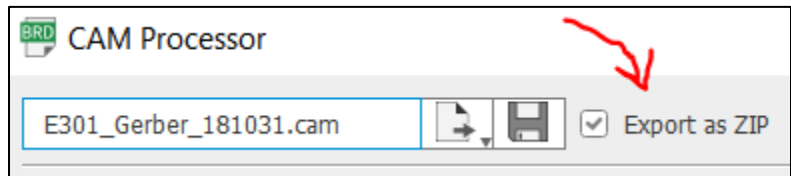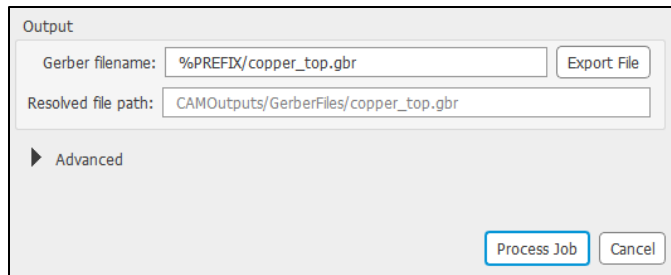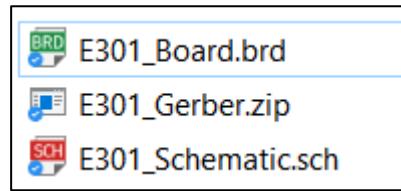

Figure 4

Figure 5

Figure 6