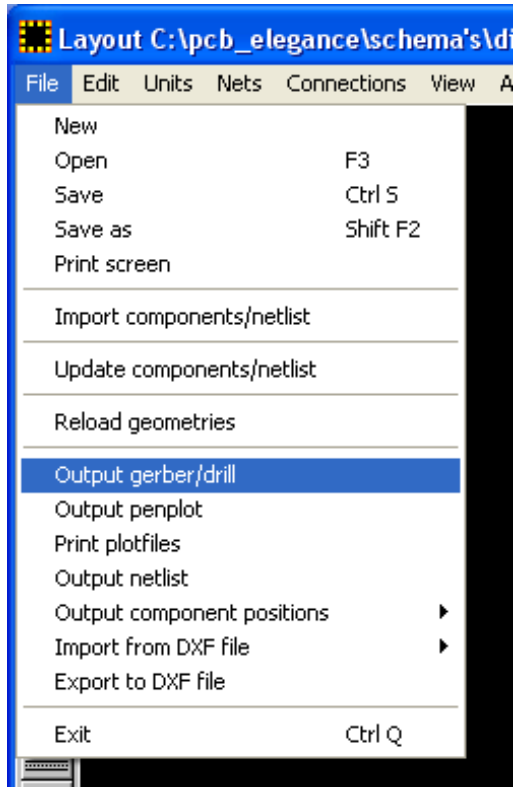


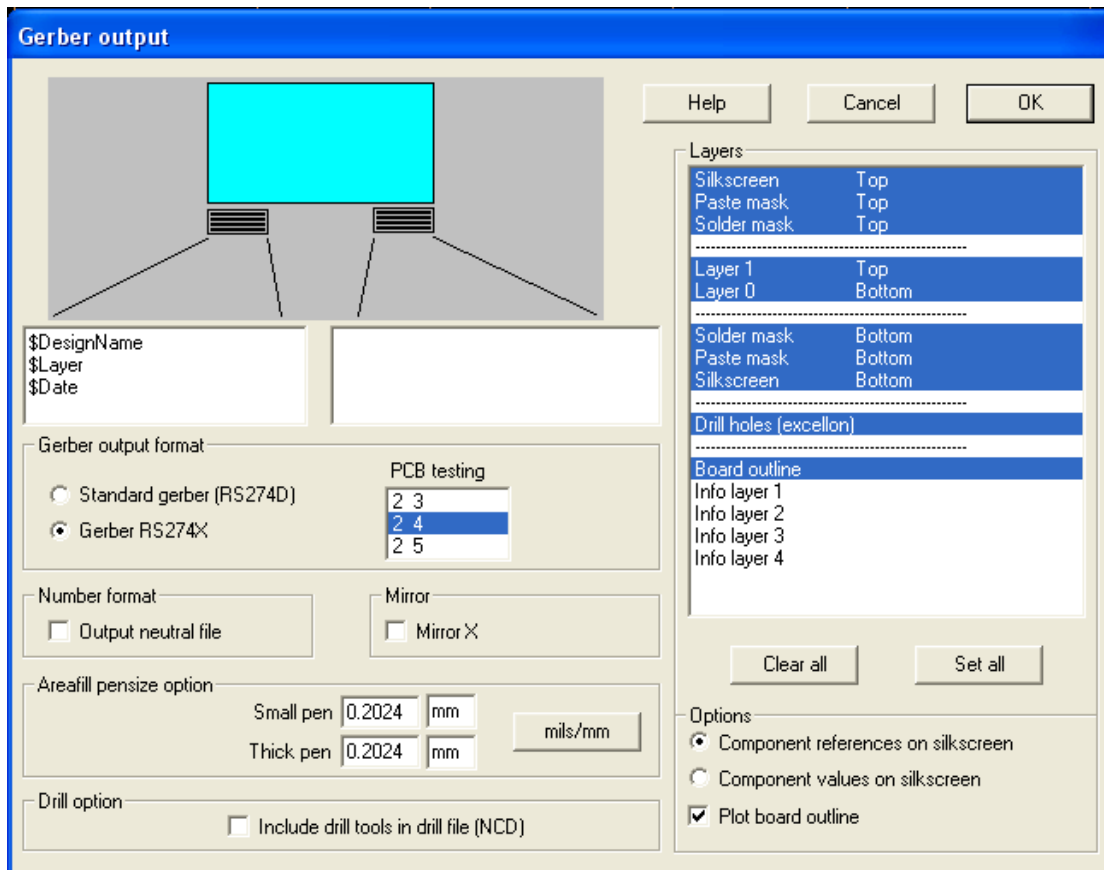
### Export Gerber files with PCB-Elegance:

Load your file, and go to Layout Editor in design manager.

Export the Gerber files with output Gerber/Drill files under File.



Select the following options in the output Gerber files. If you use one of the info layers, for example for peel off info, also select this layer. In this case you have to rename the layer in explorer to the function where you want to use the layer for, for example "peel off info top".



After click OK you can find the Gerber RS-274X files in the directory where you have the design in the PCB directory and then Gerber. The directory can look like this:  
C:\pcb\_elegance\schematics\powersupply\pcb\gerber