

1. Open the file in Protel 99SE
2. Open "File Catalogue"
3. Open the board view file
4. Double-click on the "Keep out layer"
 - A "Track - Propertis" window will open
 - As „Width” - set 0,2mm
 - Uncheck "Keepout"
 - Check "Global"
 - Attributes to match by:
 - Width - same
 - Layer - same
 - „Change scope”
 - Choose "All primitives"
 - OK
 - „You are about change 3 object(s). Continue?” – select “Yes”
5. Choose:
 - File
 - CAM manager
 - Next
 - Choose "Gerber (Generates Gerber Files)"
 - Next
 - Type in the file's name (eg. Gerber Output 1)
 - Next
 - Next
 - Choose inches as units
 - And "format - 2:3"
 - Next
 - Choose:
 - Menu
 - Plot layers
 - All on
 - Menu
 - Mirror layers

- All off
 - Next
 - Choose
 - Uncheck "Do you want to generate any drill drawing plots"
 - Also uncheck "Do you want to generate any drill guide plots"
 - Next
 - Next
 - Finish
6. A "Gerber Output 1" file has been created
 7. Right-click the file
 8. Choose "Insert NC drill" and
 - Choose:
 - Options:
 - Units as inches
 - And 'format – 2:3"
 - Advanced:
 - Check "Keep leading and trailing zeroes"
 - Check "Reference to relative orgin"
 - Check "Optimize change location commands"
 - And OK
 9. A "NC drill output 1" file has been created
 10. Left-click "gerber output 1"
 11. Press F9
 12. Now a "CAM for... (file name)" folder has been created in your catalogue
 13. Right-click on the created folder
 - Choose "export"
 - Then "Select a folder", where you need to choose your catalogue
 - And OK
 14. Close Protel 99SE
 15. The folder has been exported to your catalogue with the needed file
 16. In the "CAM for..." catalogue you will find generated gerber files