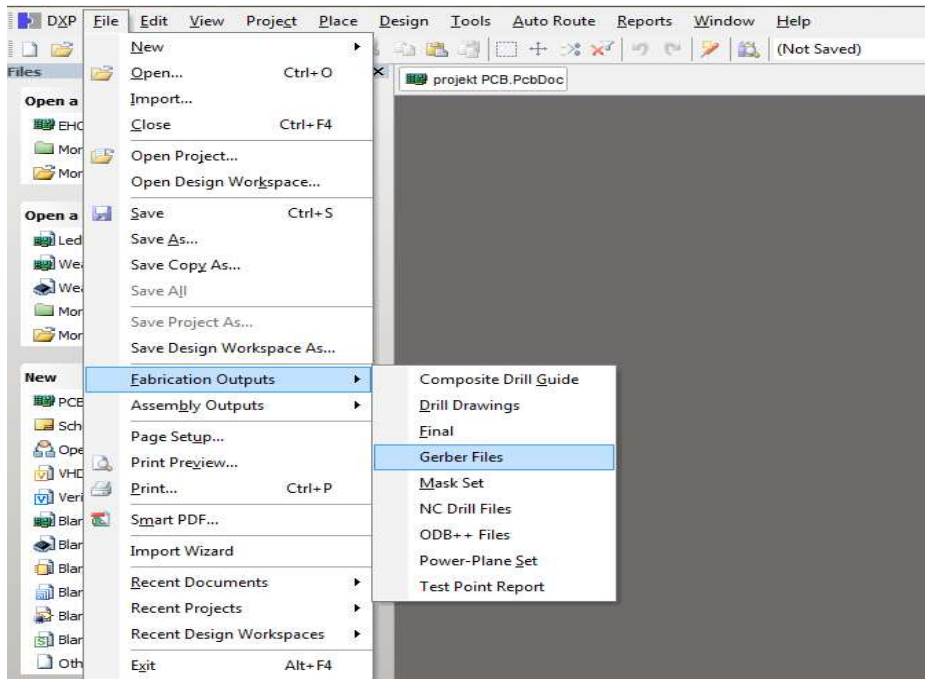
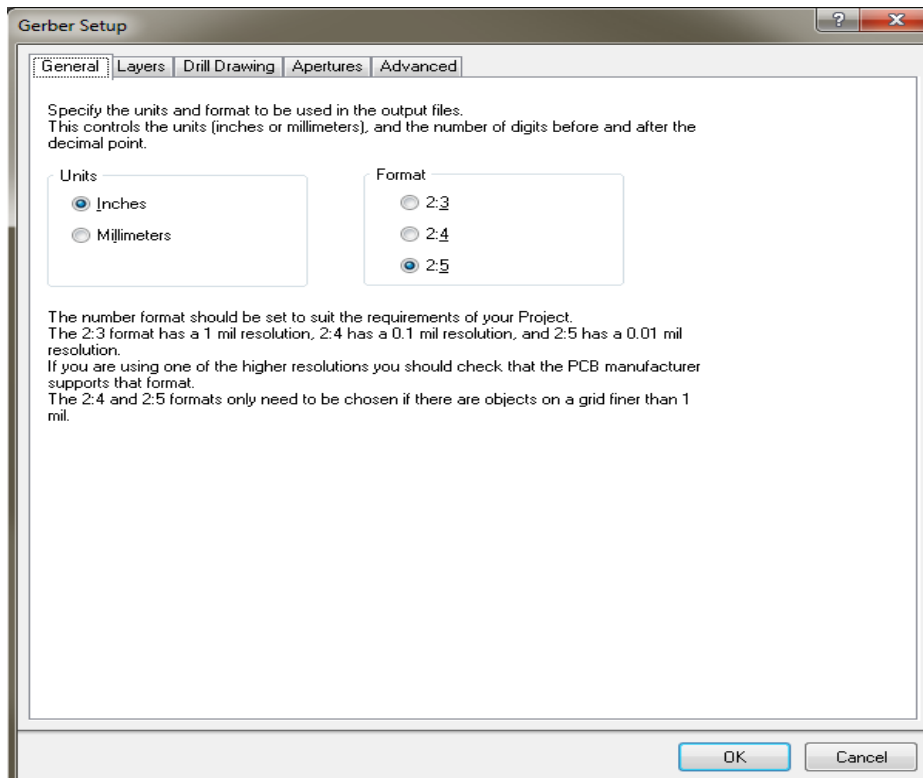


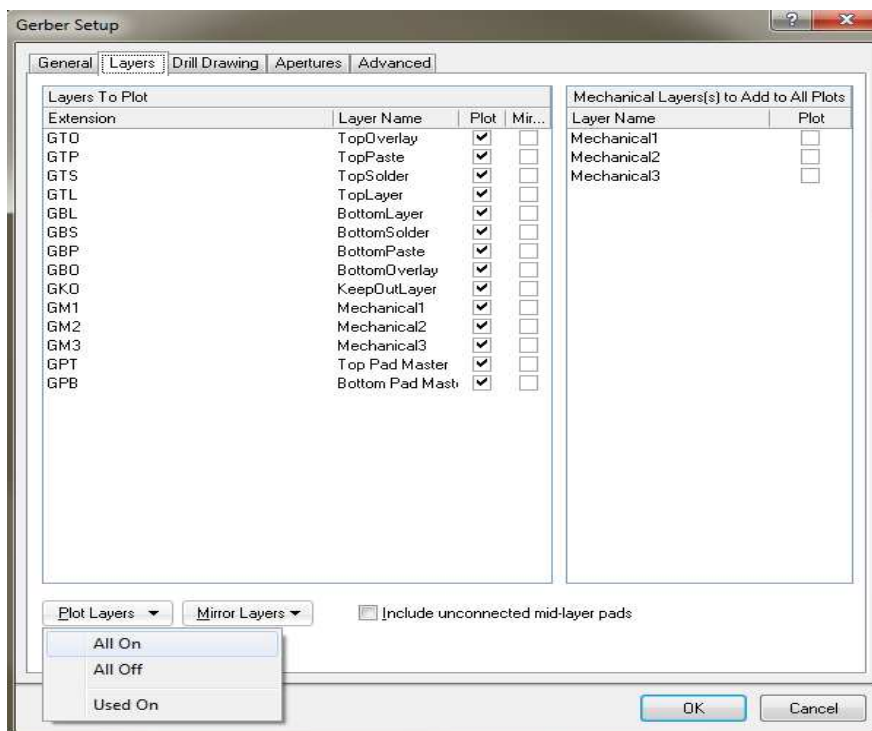
File → Fabrication Outputs → Gerber files



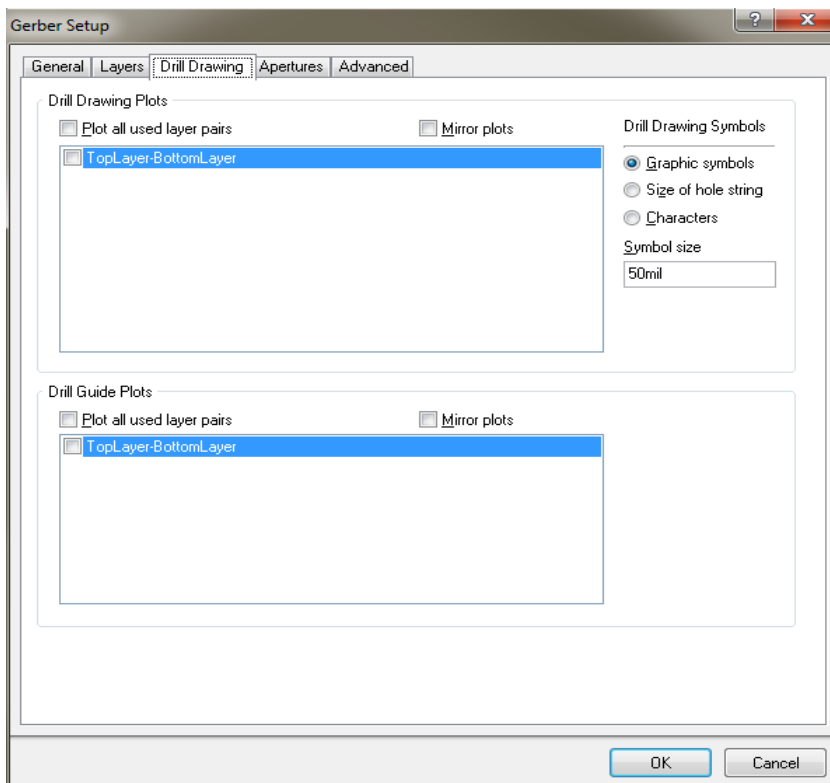
General → Units: Inches → Format: 2:5



Layers Plot Layers → All On

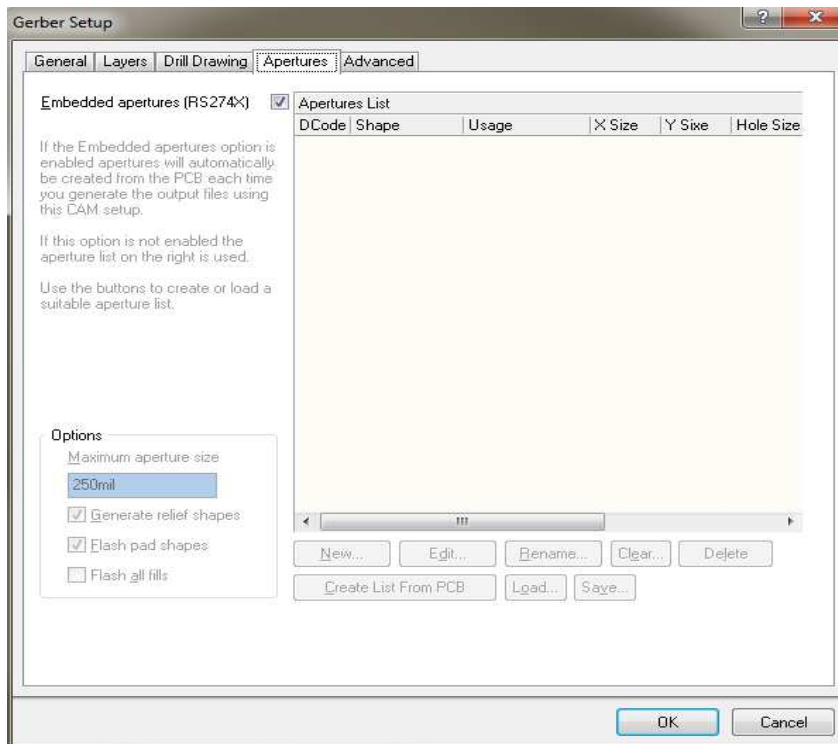


Drill Drawing Drill Drawing symbols Graphic symbols



Apertures

Embedded apertures (RS274x)



Advanced

Leading / Trailing Zeros:

Keep leading and trailing zeros

Position on Film:

Reference to relative origin

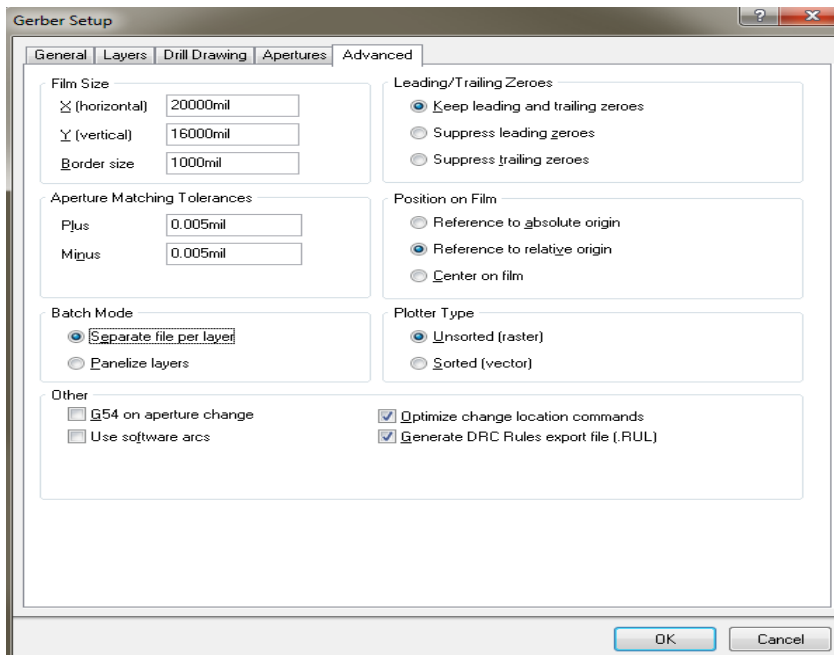
Plotter Type:

Unsorted (raster)

Optimize change location commands

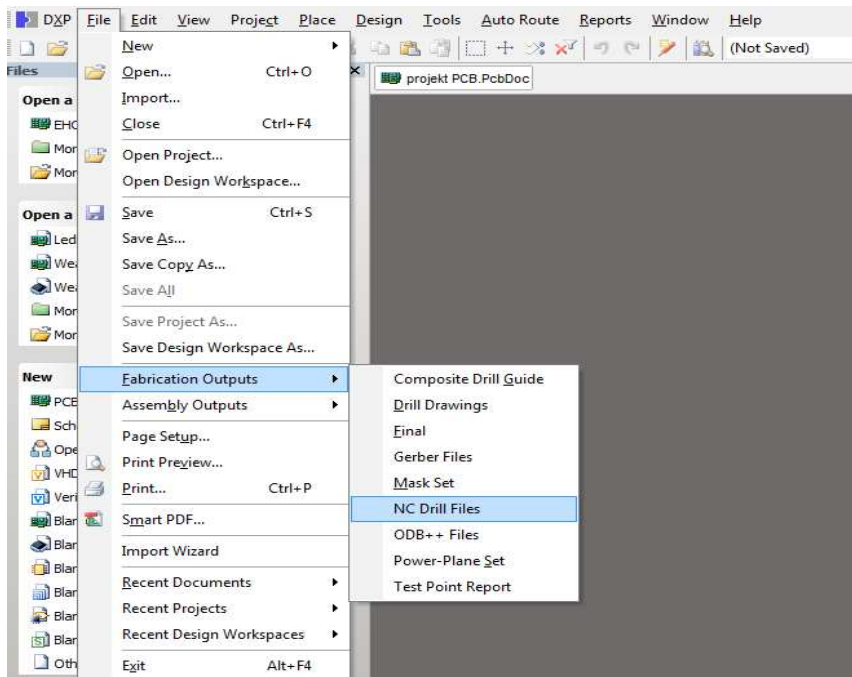
Generate DRC Rules export file (RUL)

→ OK



Pliki wierceń

File → Fabrication Outputs → NC Drill Files



NC Drill Setup

Options:

Units: Inches
 Format: 2:3
 Leading / Trailing Zeros: Keep leading and trailing zeros
 Coordinate Positions: Reference to relative origin
 Other: Optimize change location commands

→ **OK**

Pliki zostały zapisane w katalogu, w którym znajdował się projekt PCB.

