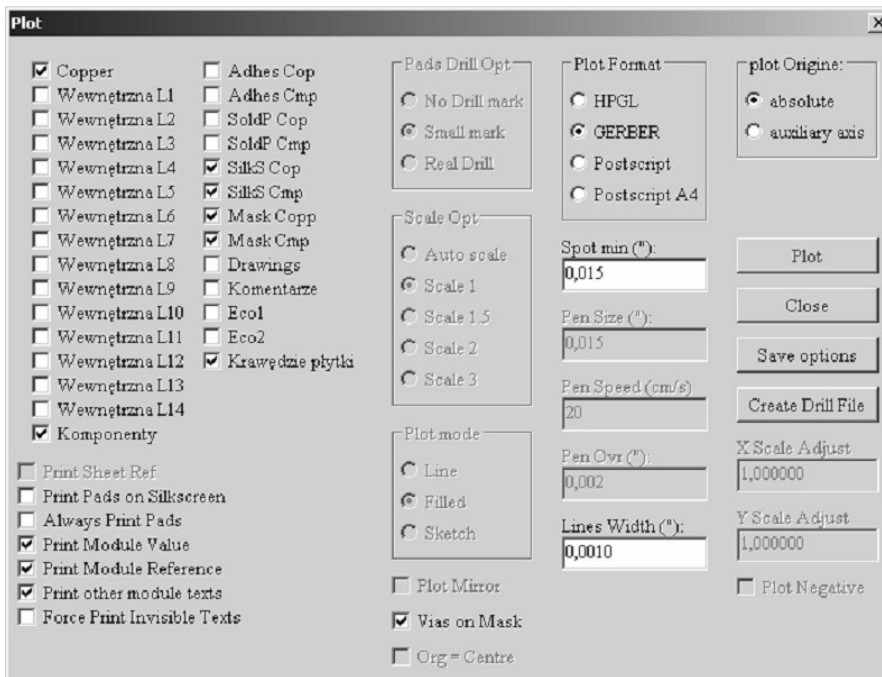
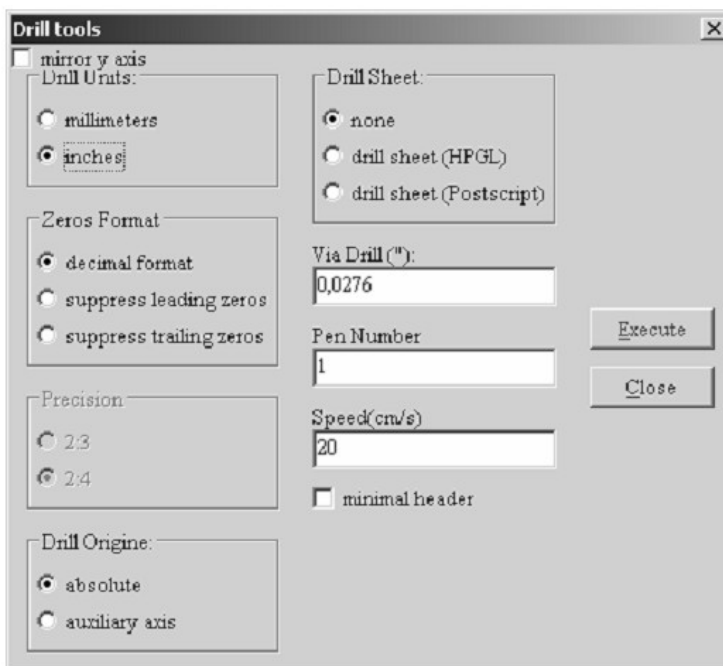


1. Open the file in KiCad
2. Choose:
  - Files
  - Plot
  - Check the boxes as shown:



**ATTENTION:** The "Vision Mask" should be chosen only in case of boards that are not meant to be metallized; then the grommets won't be covered by soldermask.

- Click "Create drill file" and choose:



- Then, click "Execute"
    - Save the "Wiertla.drl" file in your project's catalog
  - Then, click "Plot"
3. In the project's catalog now we have the gerber and drill files (.pho & .drl)