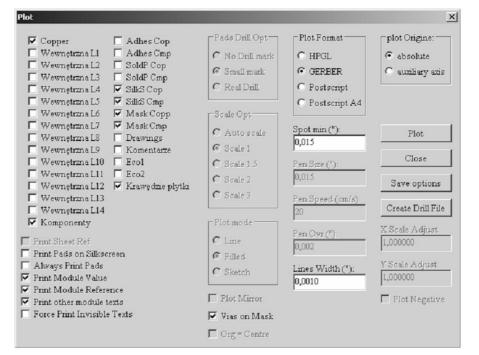


- 1. Open the file in KiCad
- 2. Choose:

•

- 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 soldiermask.

Click "Create drill file" and choose:

Drill tools		×
mirror y axis Dull Urats: millimeters inches Zeros Format decimal format suppress leading zeros suppress trailing zeros Precision 2:3 2:4 Drill Origine: absolute auxiliary axis	Drill Sheet: • none • drill sheet (HPGL) • drill sheet (Postscript) Via Drill ("): 0,0276 Pen Number 1 Speed(cm/s) 20 • minimal header	Execute

- 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)