1. In schematic editor use tools->generate legacy bom. Use bom2\_grouped.csv generator script.
2. Use generate\_boms.bat
3. In PCB editor, plot gerber. Use these setings: A screenshot of a computer

   Description automatically generated
4. Generate component placement file. (File->Fabricatiob\_Outputs.
5. Use fab/zip\_gerbers.bat
6. Edit thors\_arsenal\_cpl.xlsx. Use Data->Refresh All. Then save.
7. Done.