Altium PCB mill Workflow

* In PCB, File->FabricationOutputs->Gerber: get Gerbers of necessary files (TopLayer, BottomLayer)
* In PCB, File->FabricationOutputs->NCDrill: get the drill outlines for the holes. Select option to include board outline
* Then open these in CAMtastic
* Then for each file, File->Export->BMP
  + Set DPI=1000
  + Confirm that size is correct
  + Monochrome
* Convert to PNG online