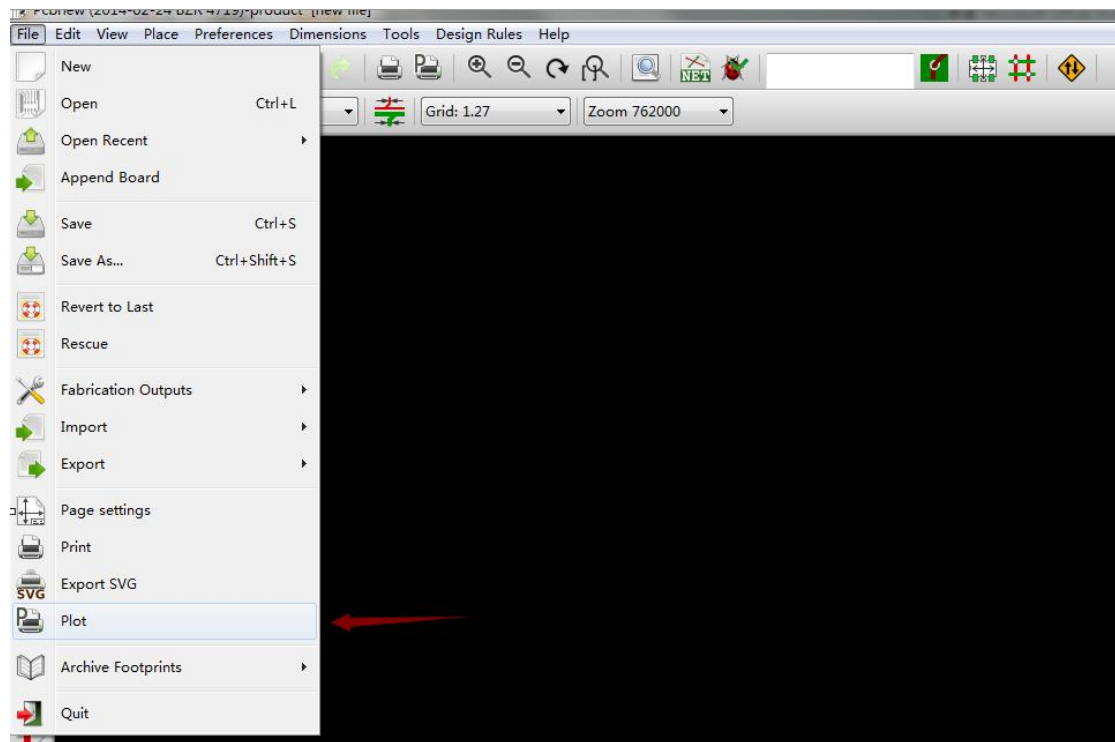
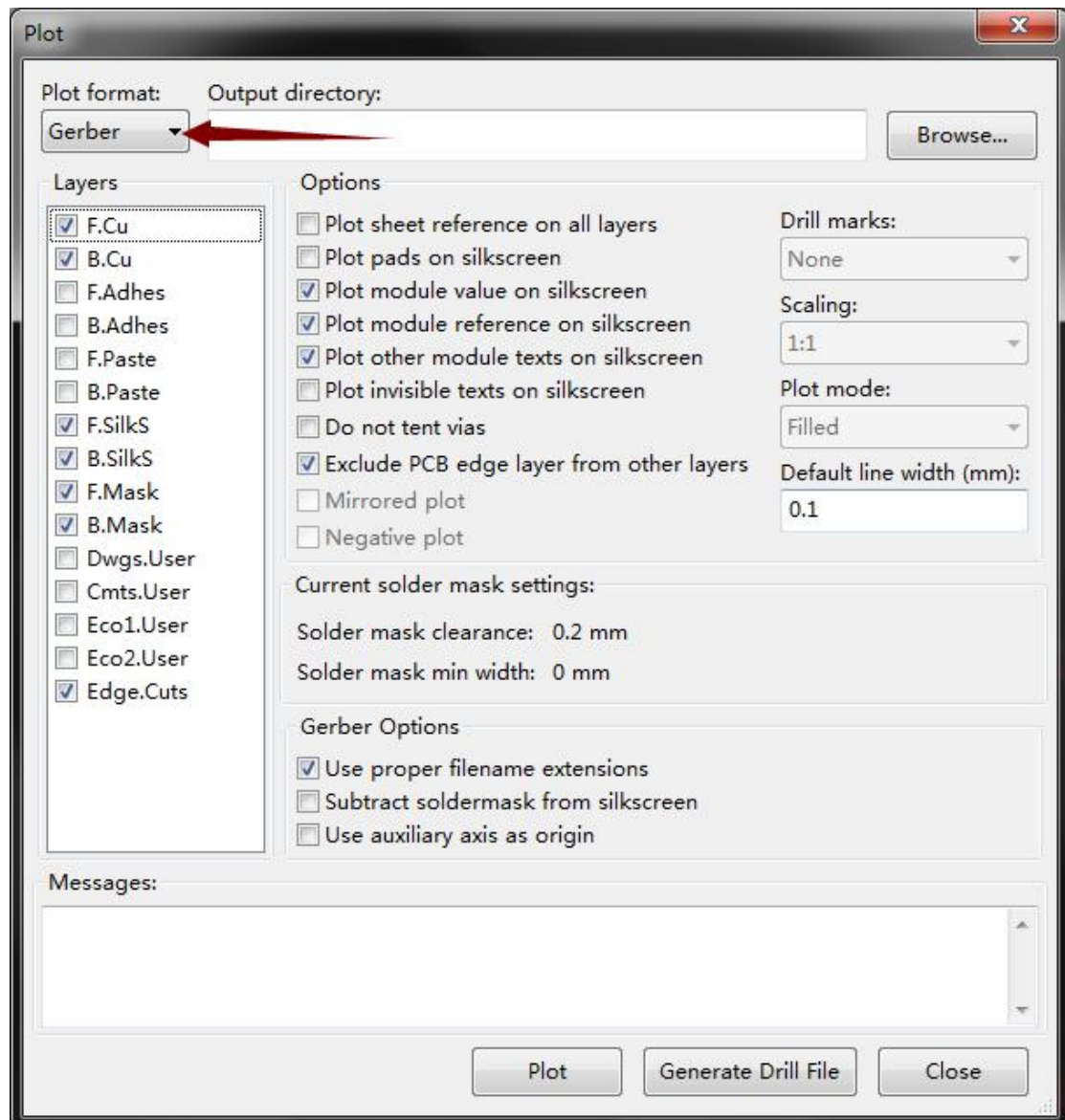


Output Gerber

1.

File>Plot





Select **gerber** tab and tick the layers.

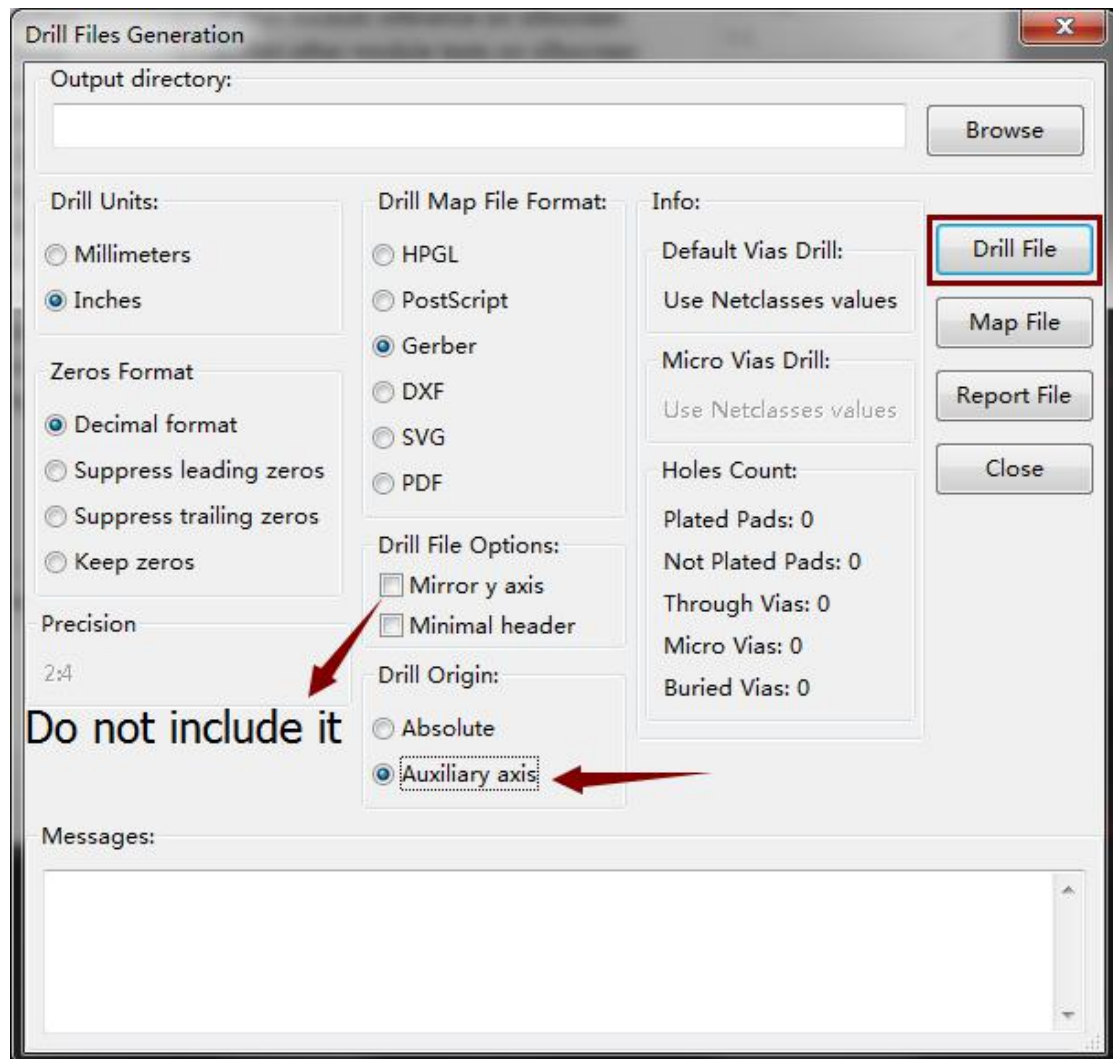
For a 2-layer PCB, we need

- **F.Cu&B.Cu** (Top layer& Bottom layer),
- **F.Silks&B.Filks** (Silk Top & Silk Bottom),
- **F.Mask&B.Mask** (Solder Stop Mask top & Solder Stop Mask Bottom)
- **Edge.Cuts** (Mechanical layer)

Other options can be adjusted by the designer, or leave the default settings.

2.

>Generate Drill File



Click **Drill File** and **Map file** to get drilling data

- Name.drl
- Name-drl_map.gbr

Therefore, we can output the file needed for production.