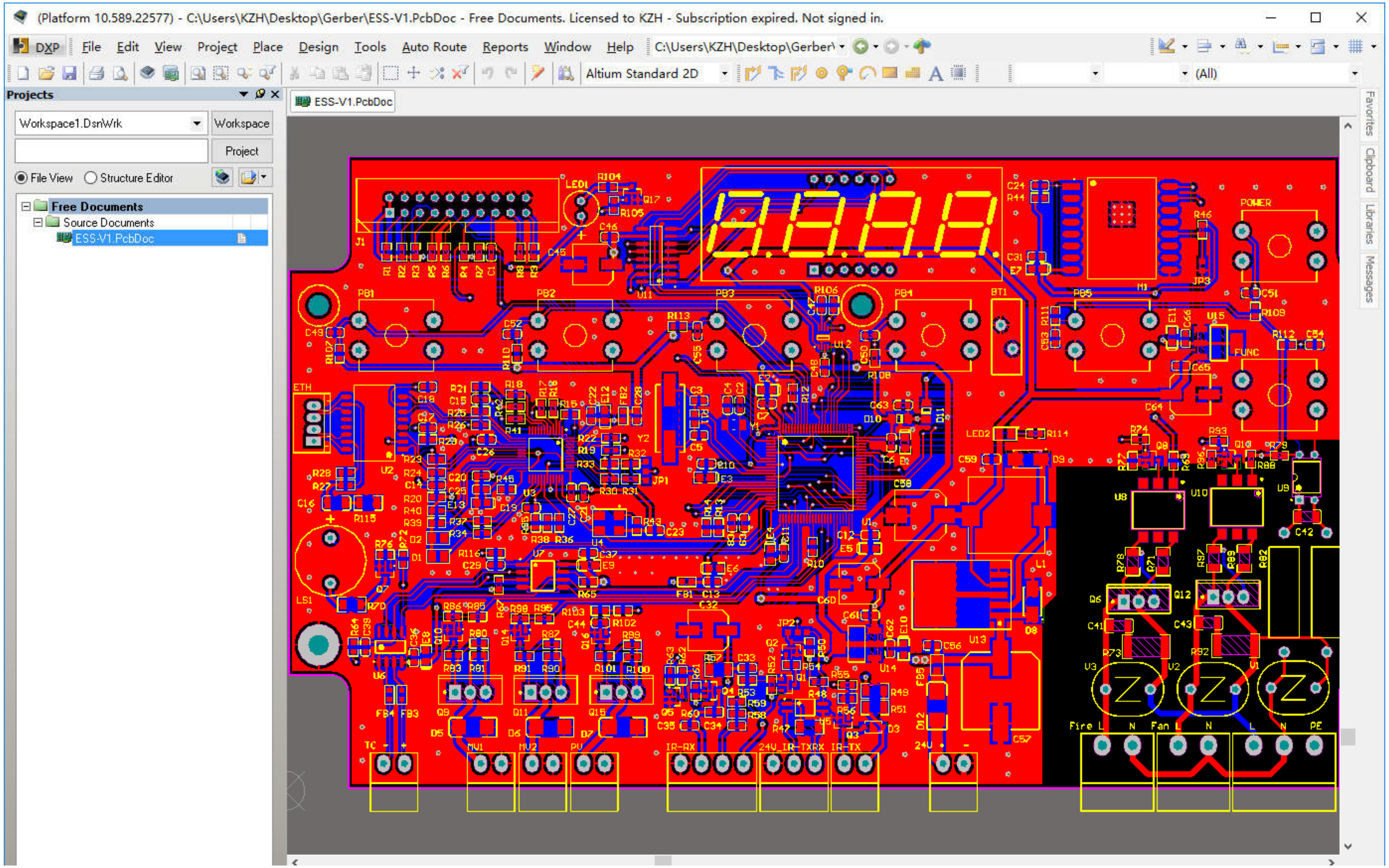
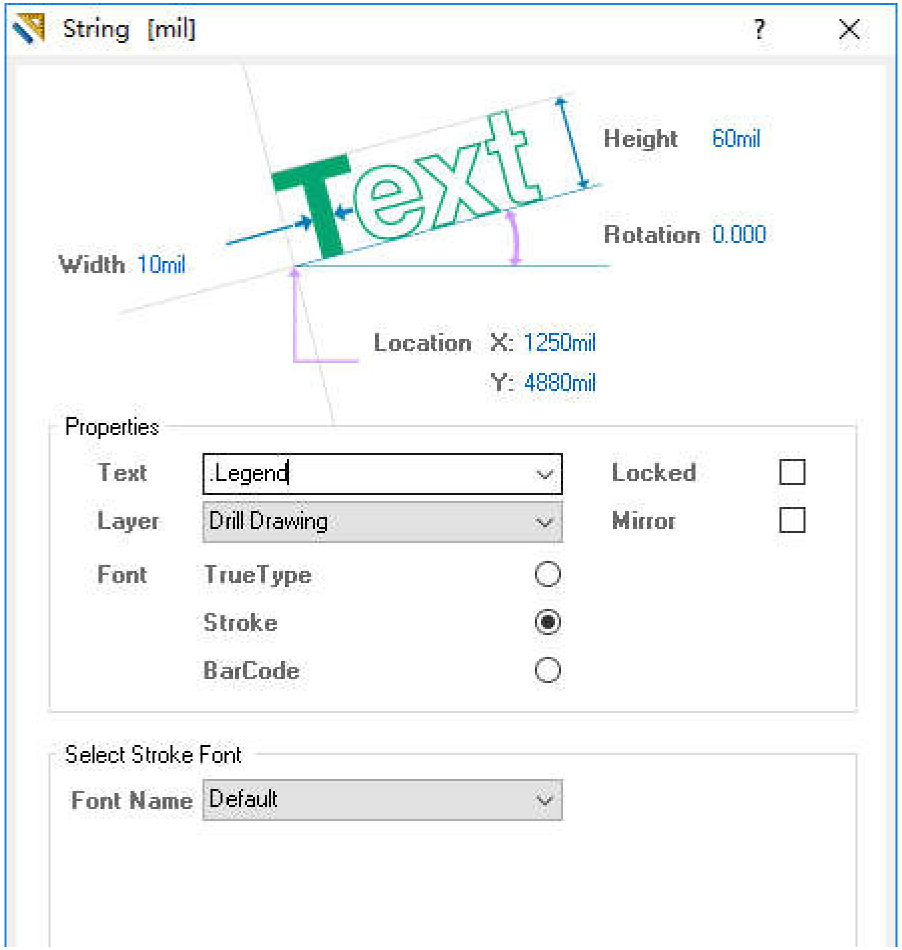
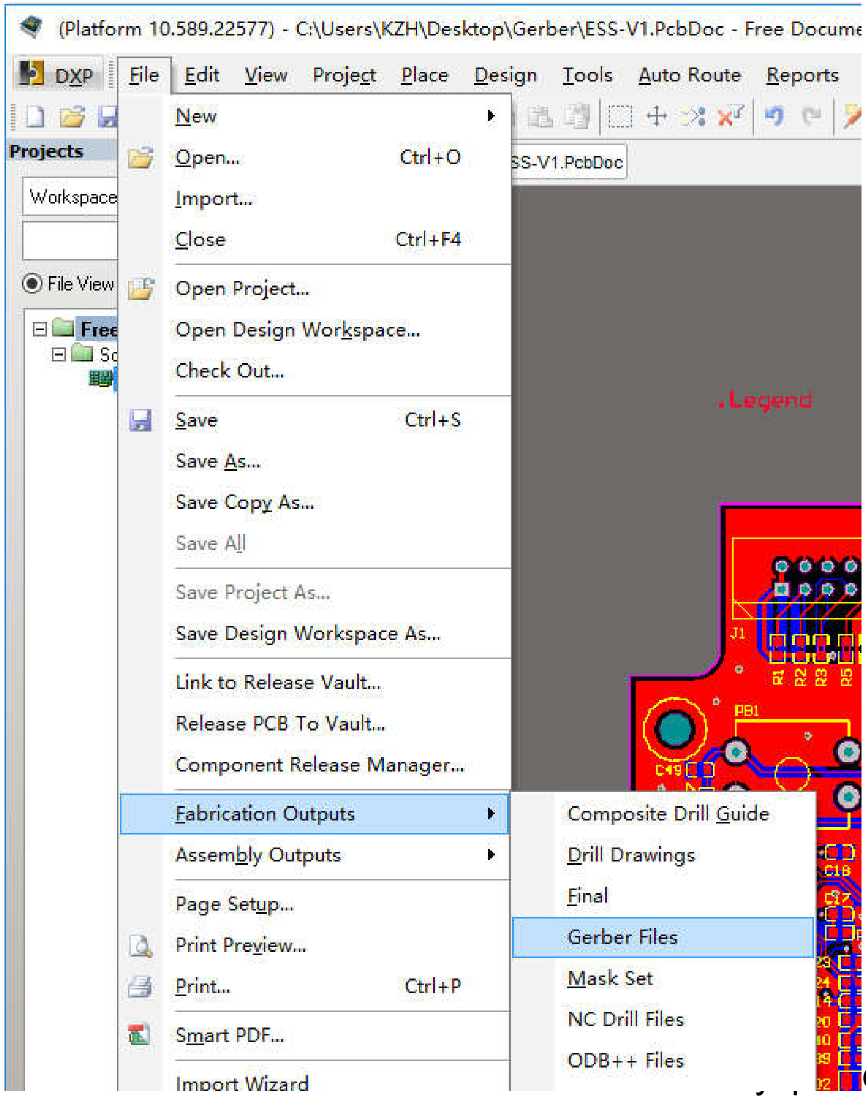
1. Start the Altium designer software and open the designed PCB file.



2. Place the hole chart list string, click Place -> String in the main menu, and then press the Tab key to select ".Legend" in the Text column, and select "Drill Drawing" in the Layer column, generally placed at the top or bottom right of the board. The corresponding position will generate a Drill Table. (You can also directly Place->Drill Table)

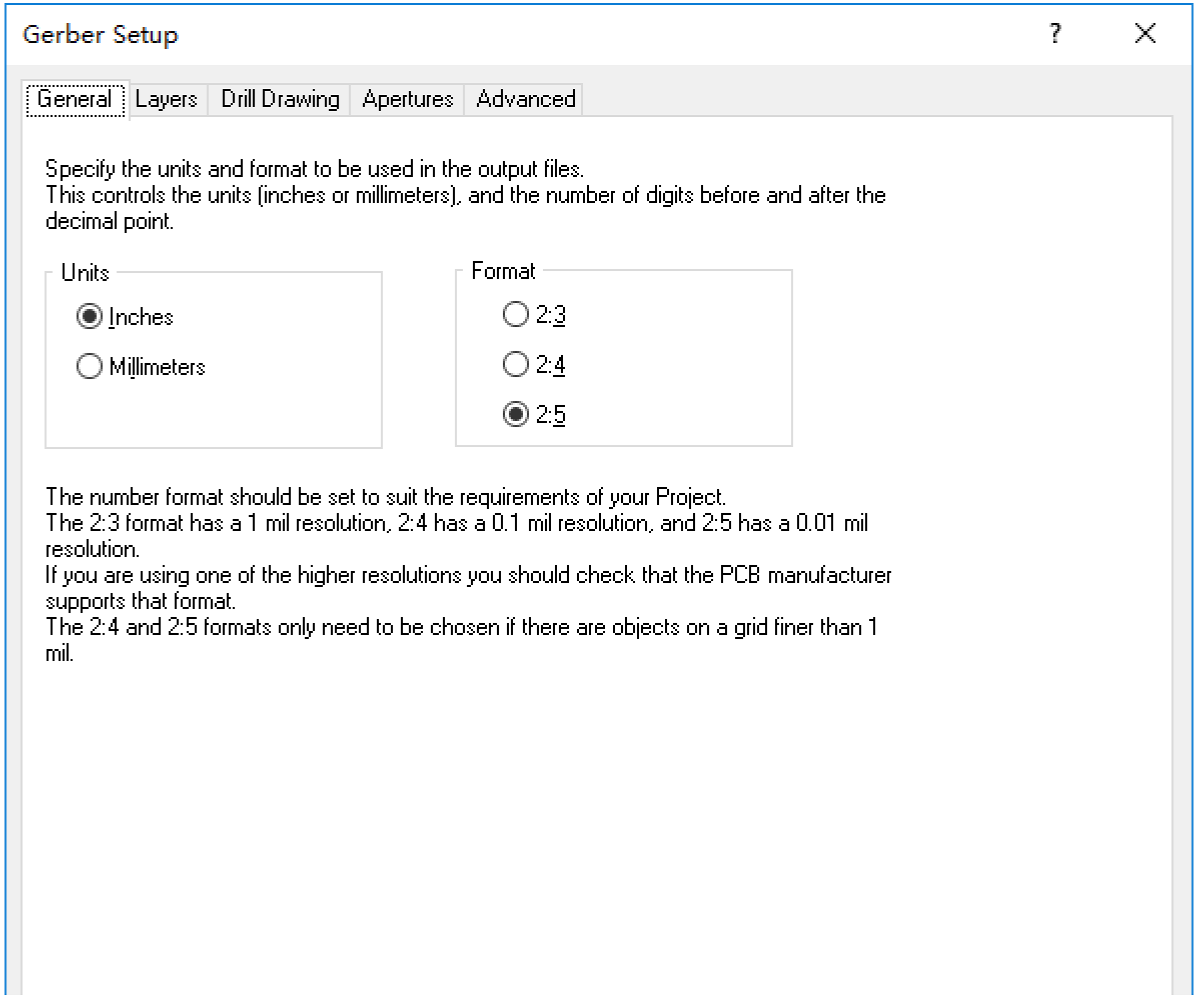


3. Start to transfer Gerber File. In the PCB file environment, left-click File -> Fabrication Outputs -> Gerber Files to enter the Gerber setup interface.

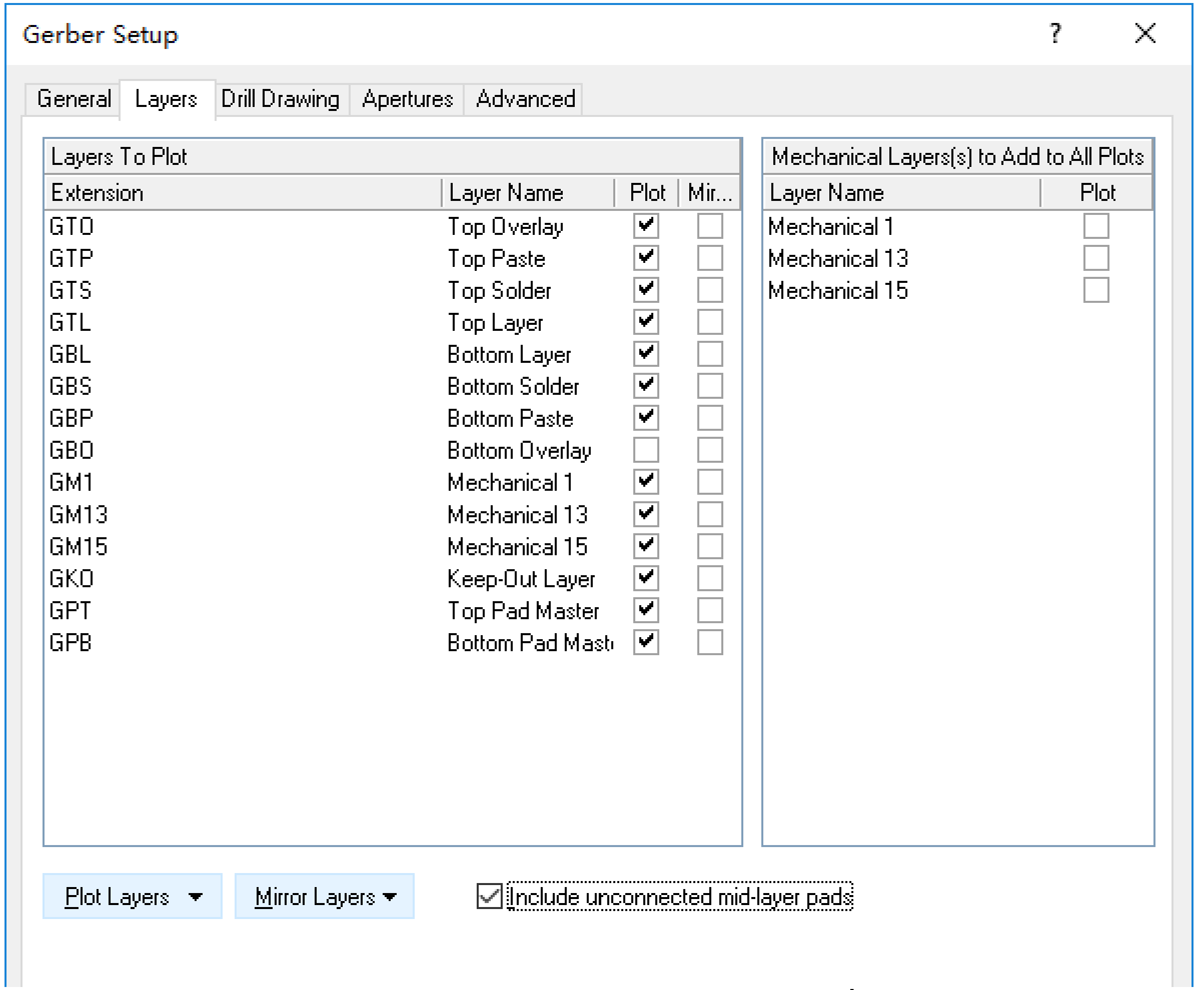


com

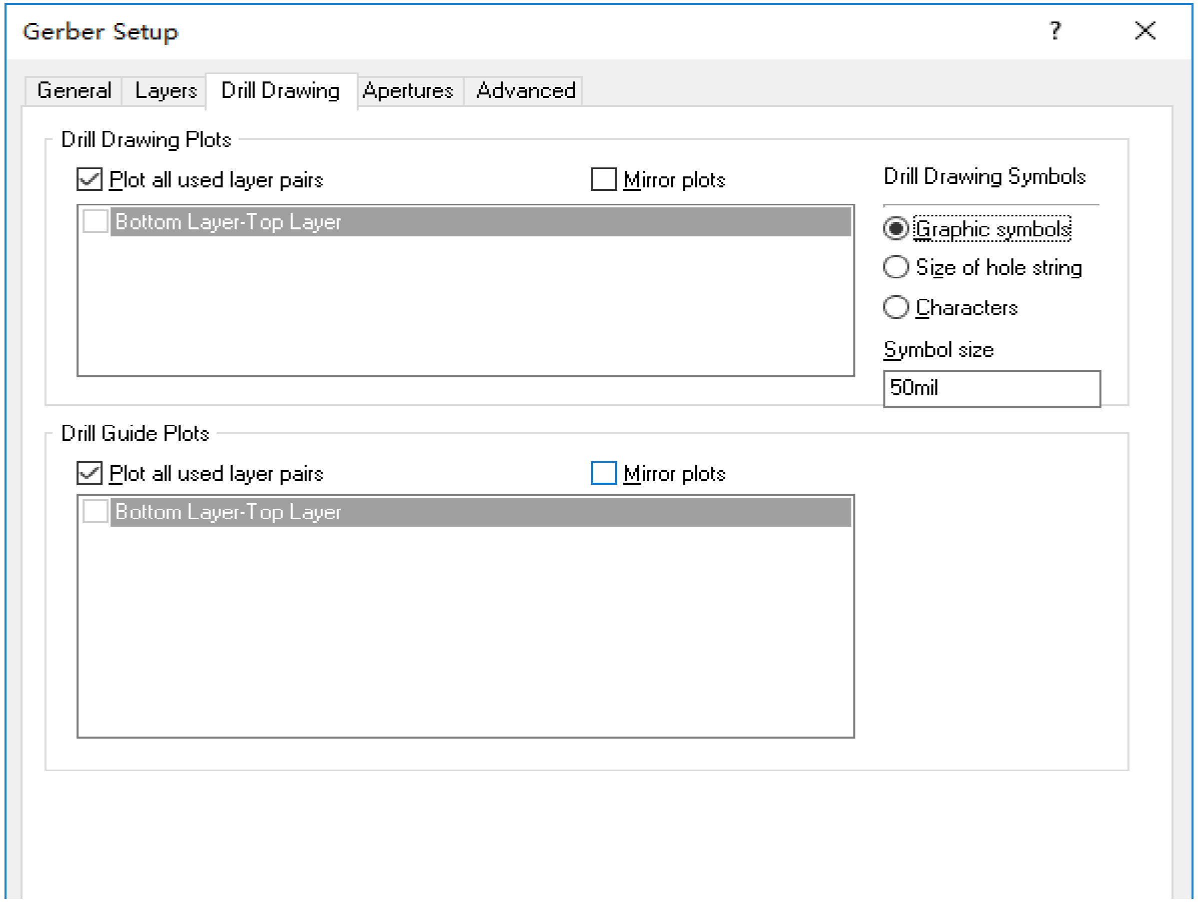
4. In the "General" option, select "Inches" for "Units" and 2:5 for Format. This size accuracy is relatively high.



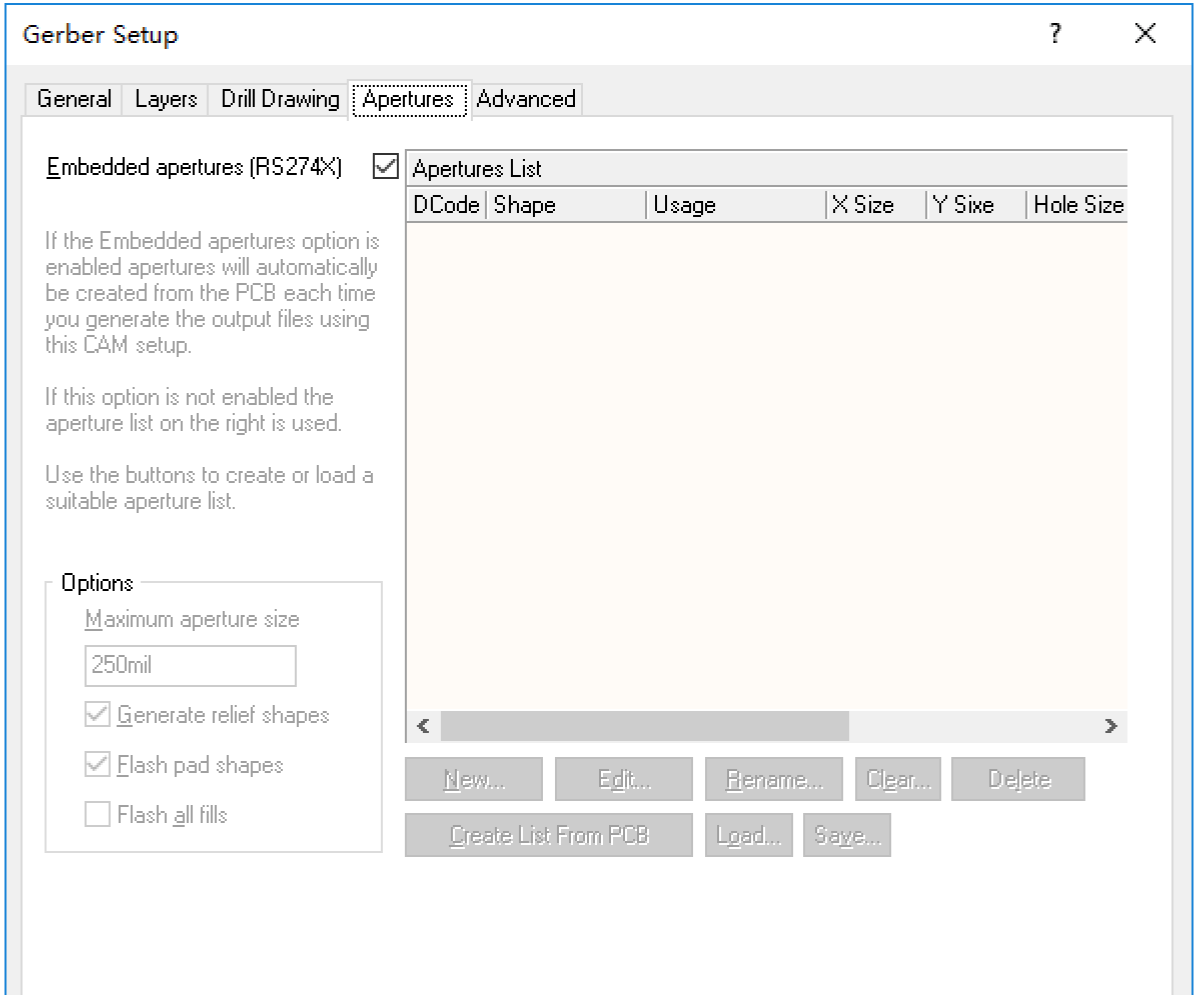
5. In "Layers", select "Include unconnected mid-layer pads", select "Used On" in the "Plot Layers" drop-down menu, check it, and don't drop the layer. Select "All Off" in the "Mirror layers" drop-down menu, and do not select the mechanical layers on the right.



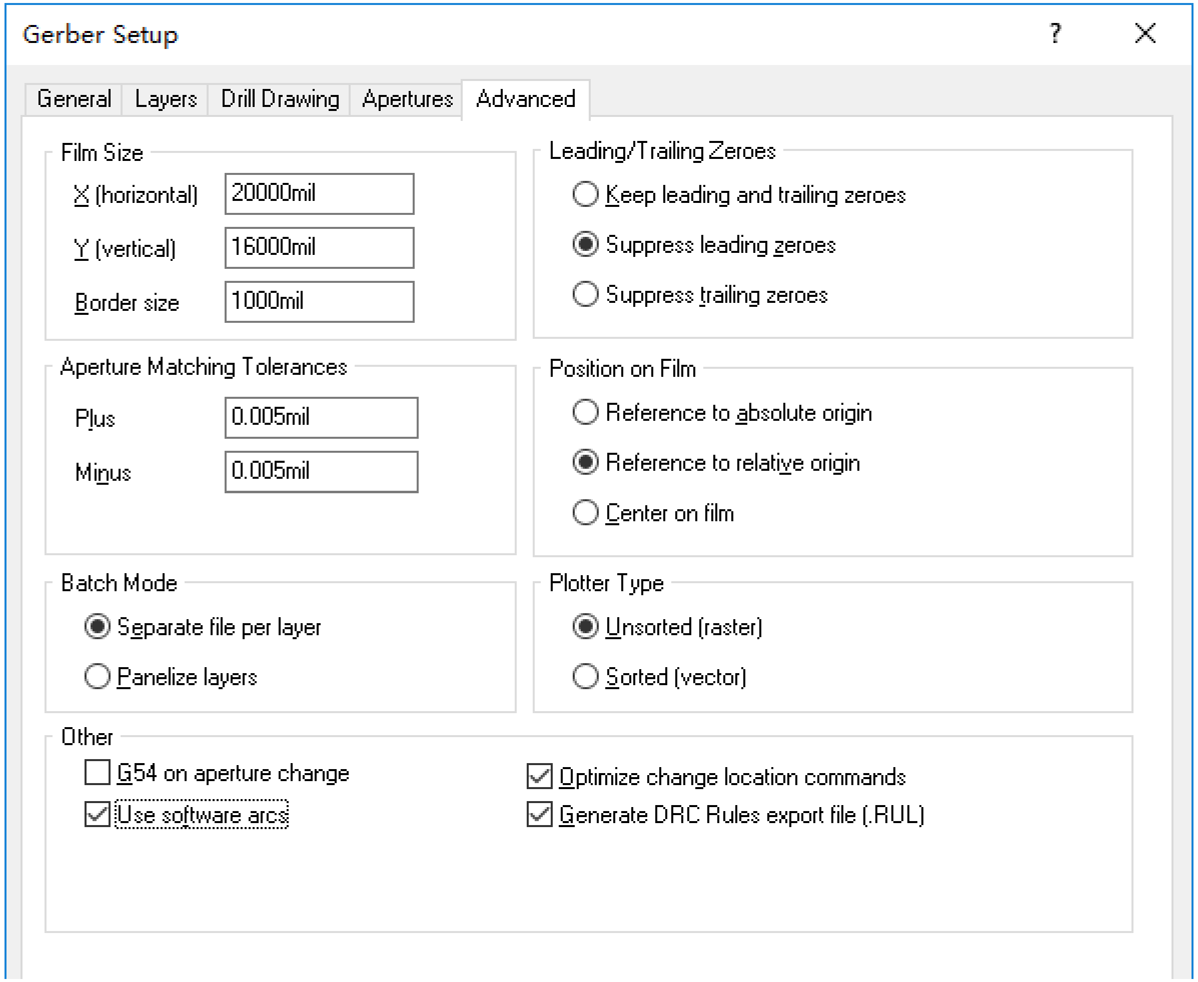
6. In "Drill Drawing", set the parameters of the hole layer and select the layer pair you want to export. Generally select "Plot all used layer pairs", and do not select "Mirror plots". Make the same selection for Drill Drawing plots and Drill Guide Plots.



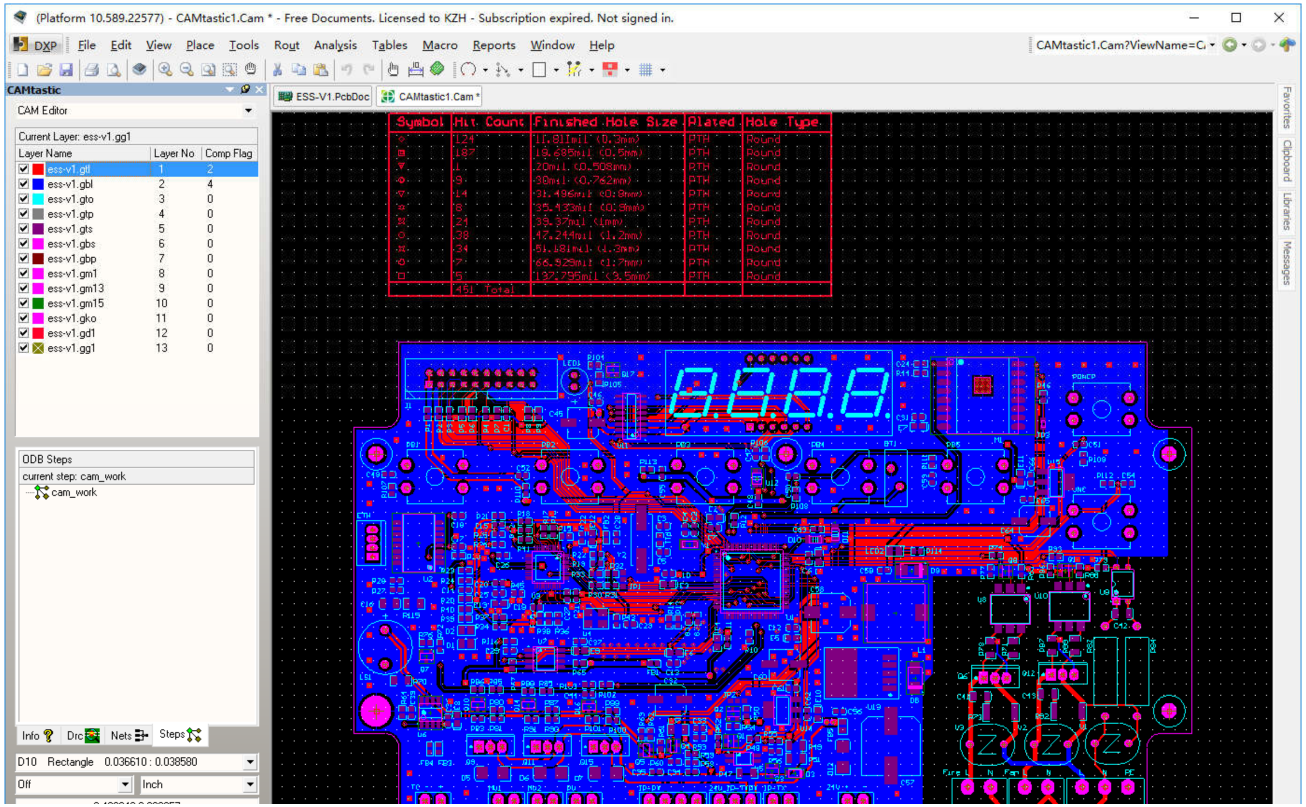
7. In "Apertures", check "Embeded apertures(RS274X)", which contains D code format.



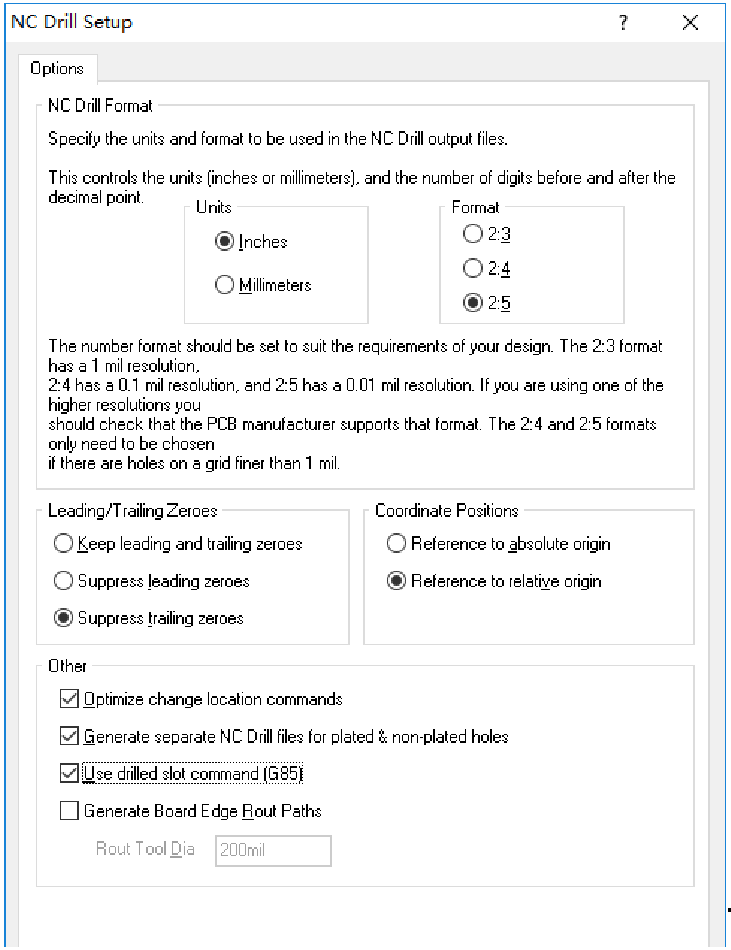
8. In "Advanced", in the "Leading/Trailing Zeroes" area, select "Suppress leading zeroes". By default, the line that is designed as an arc is turned out and still is an arc. If you tick the "Use software arcs" under the Other column, the designed arc will be replaced by a line segment.



9. Click the "OK" button, the software converts the Gerber File and automatically opens the converted Gerber File with the embedded CAM2000 software. Note: The transferred file will automatically lead each layer to the folder where the original file is located, so it is recommended that the original file be placed in an empty folder separately.

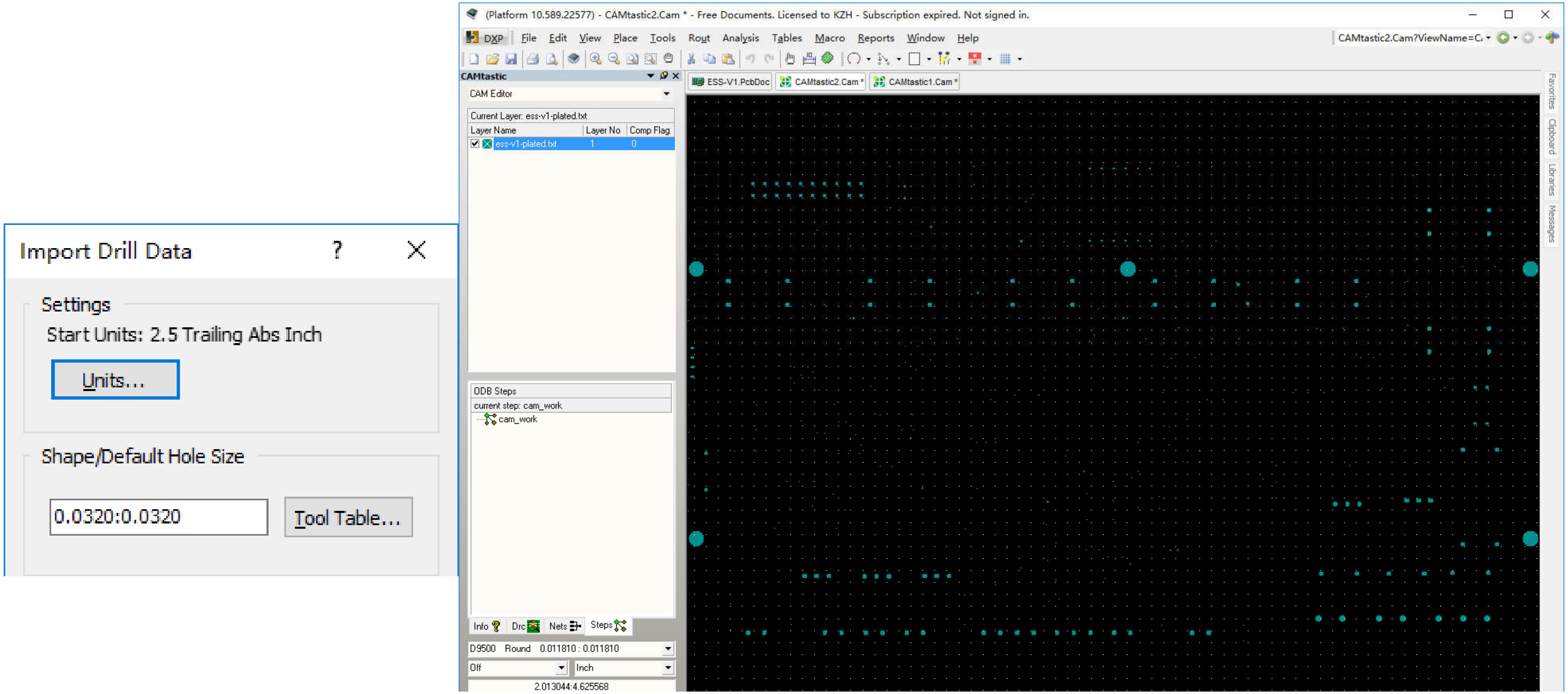


10. In the PCB file environment, left-click File -> Fabrication Outputs -> NC Drill Files to enter the NC Drill Setup interface, select "Inches" for Units, and 2:5 for Format. In the "Leading/Trailing Zeroes" area, select "Suppress trailing zeroes", which should be consistent with the previous "Advanced" of Gerber Setup, and other default options remain unchanged.



.syspcb.com

11. Click the "OK" button, and in the pop-up "Import Drill Data" interface, left-click and click "OK" to output.



12. Pack and compress all the files generated in the current project directory (excluding the PCB file itself) and send them to the PCB manufacturing factory for processing.