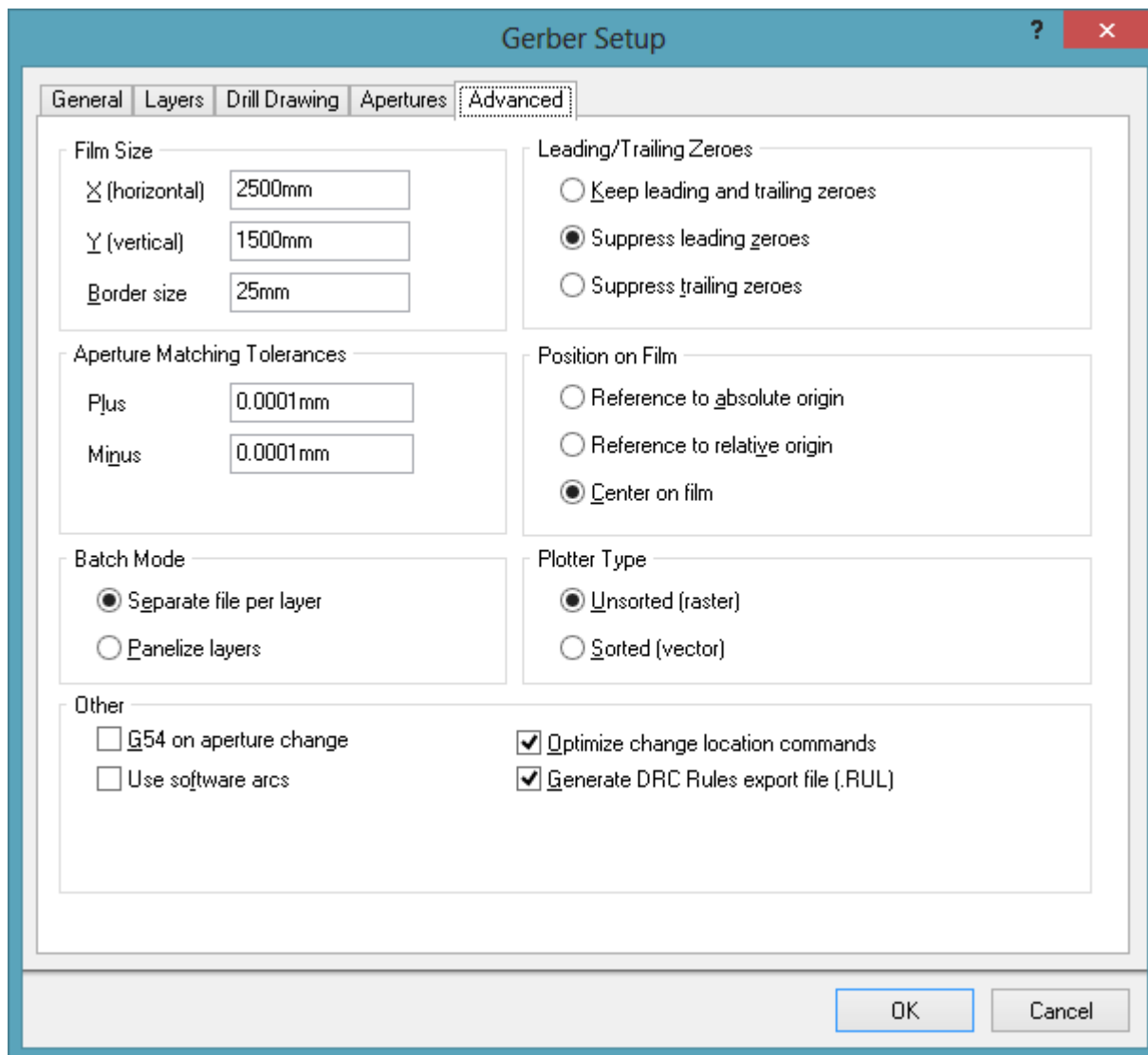


Problem: The film is too small for this pcb during generate Gerber File

Finally I set this Film size and able to generate gerber BUT..... Please refer second page,.....



The image shows a 'Gerber Setup' dialog box with the 'Advanced' tab selected. The dialog is divided into several sections with various input fields and checkboxes.

General | **Layers** | **Drill Drawing** | **Apertures** | **Advanced**

Film Size

- X (horizontal): 2500mm
- Y (vertical): 1500mm
- Border size: 25mm

Leading/Trailing Zeroes

- ☐ Keep leading and trailing zeroes
- ☒ Suppress leading zeroes
- ☐ Suppress trailing zeroes

Aperture Matching Tolerances

- Plus: 0.0001mm
- Minus: 0.0001mm

Position on Film

- ☐ Reference to absolute origin
- ☐ Reference to relative origin
- ☒ Center on film

Batch Mode

- ☒ Separate file per layer
- ☐ Panelize layers

Plotter Type

- ☒ Unsorted (raster)
- ☐ Sorted (vector)

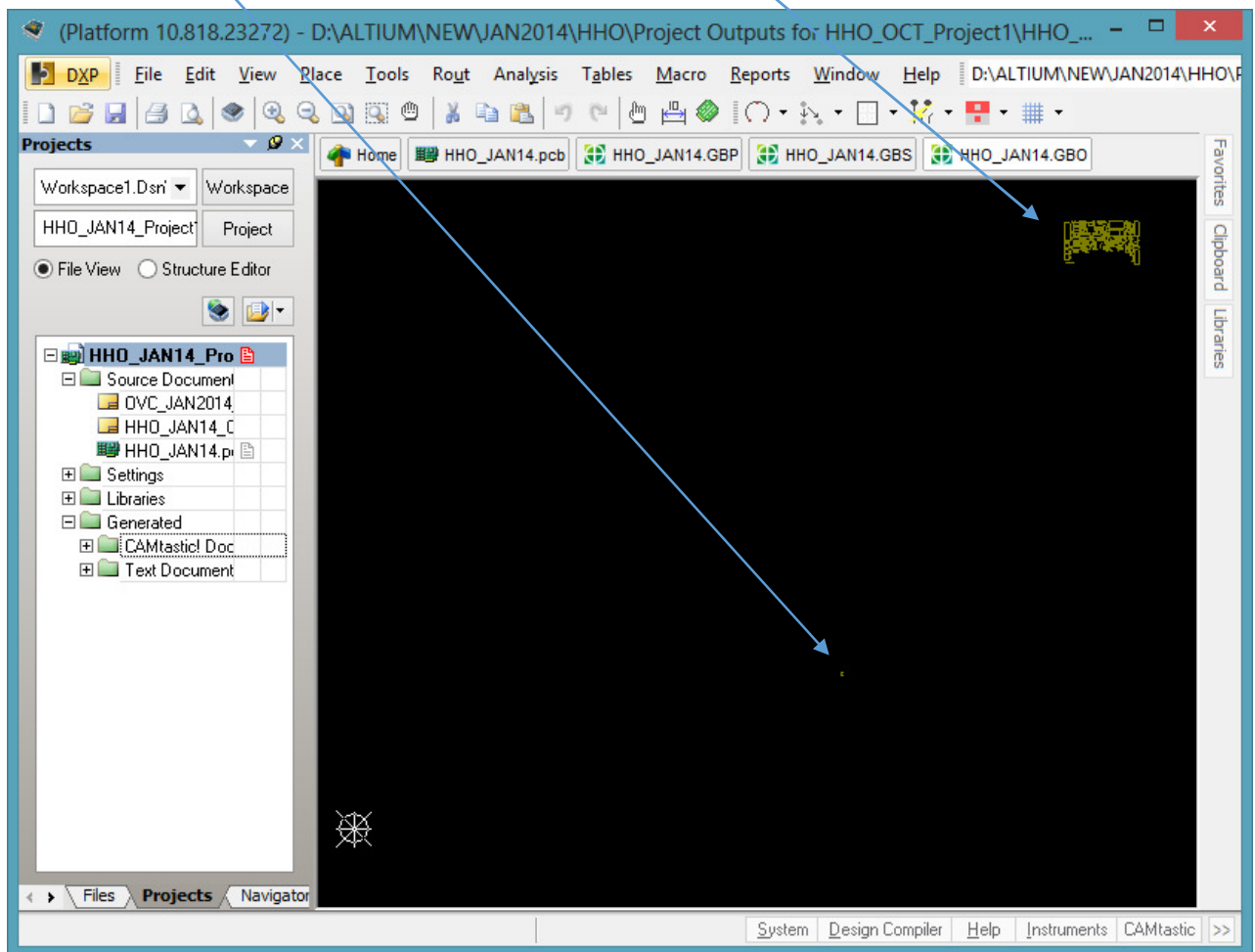
Other

- ☐ G54 on aperture change
- ☒ Optimize change location commands
- ☐ Use software arcs
- ☒ Generate DRC Rules export file (.RUL)

OK Cancel

Found an unknown component at here on GBO file:

My board



Are you able to advise how to remove this object from PCB editor on my project ?

Try below methods but not working !!

<http://altiumpcbdesigner.blogspot.com/2013/05/film-too-small.html>

<http://forum.sunstone.com/altium-designer-protel-film-is-too-small-for-this-pcb-error-t122.html>

Not really sure how to do it