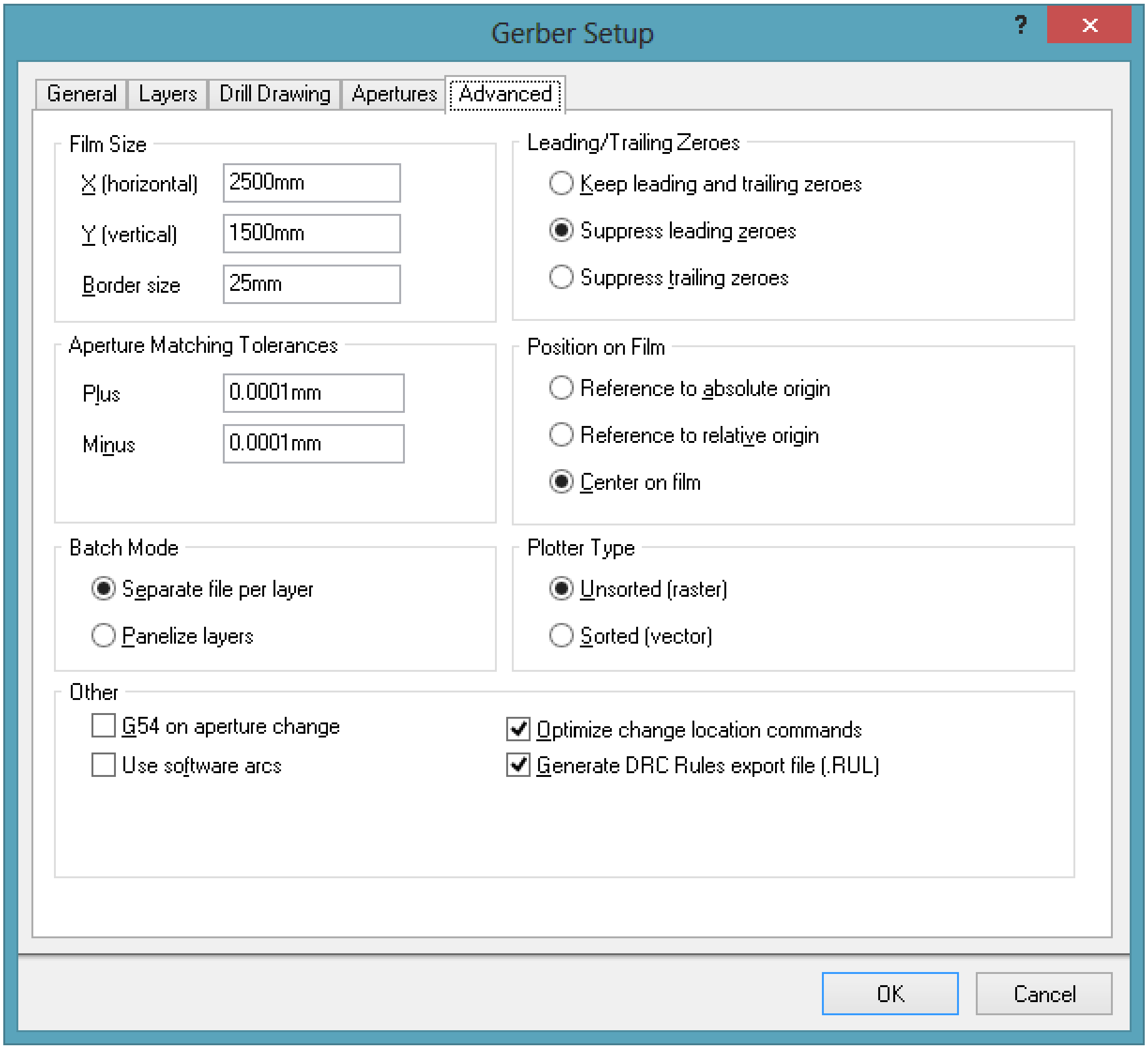
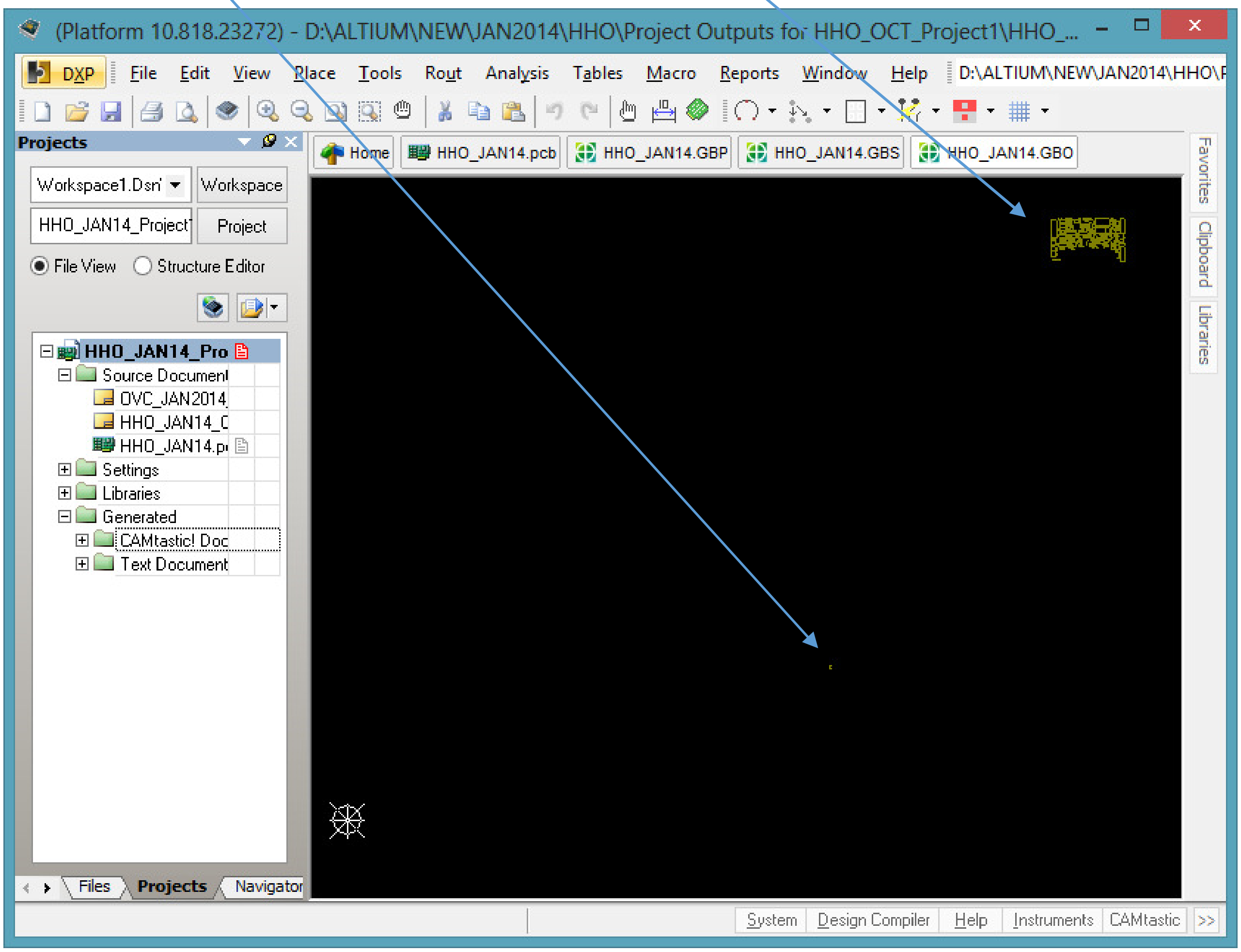
Problem: The film is too small for this pcb during generate Gerber File

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



Found an unknown componet 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 metods 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 ……..