



MINImill

ALTium TUTORIAL AND USEFUL TIPS

Developed by: Mattia Marchiori

Altium version: 16.0.5

Tutorial version: 1.0

Realization date: 09/03/2018

Summary

- 1. USEFUL TIPS 4
- 2. PRELIMINARY OPERATIONS..... 4
 - 2.1 Outline 4
 - 2.2 Origin 6
- 3. EXPORT GERBER FILES 6
- 4. MILLING 14

1. USEFUL TIPS

MINImill is specifically developed to work with 1.6mm thickness PCBs. If you want to work with thinner boards and you want to obtain good results, please put something underneath your board in such a way as to increase the overall thickness (a piece of cardboard is a good solution). Please also be sure that your board is rigid enough to avoid z-axis oscillations. This problem especially appears with thin boards. The best solution is to place some double-side tape between all the layers you have (for example mill base – tape – board or mill base – tape – cardboard – tape – board).

Other information is available on the website: www.minimill.at/en/pcb-milling-tutorial/

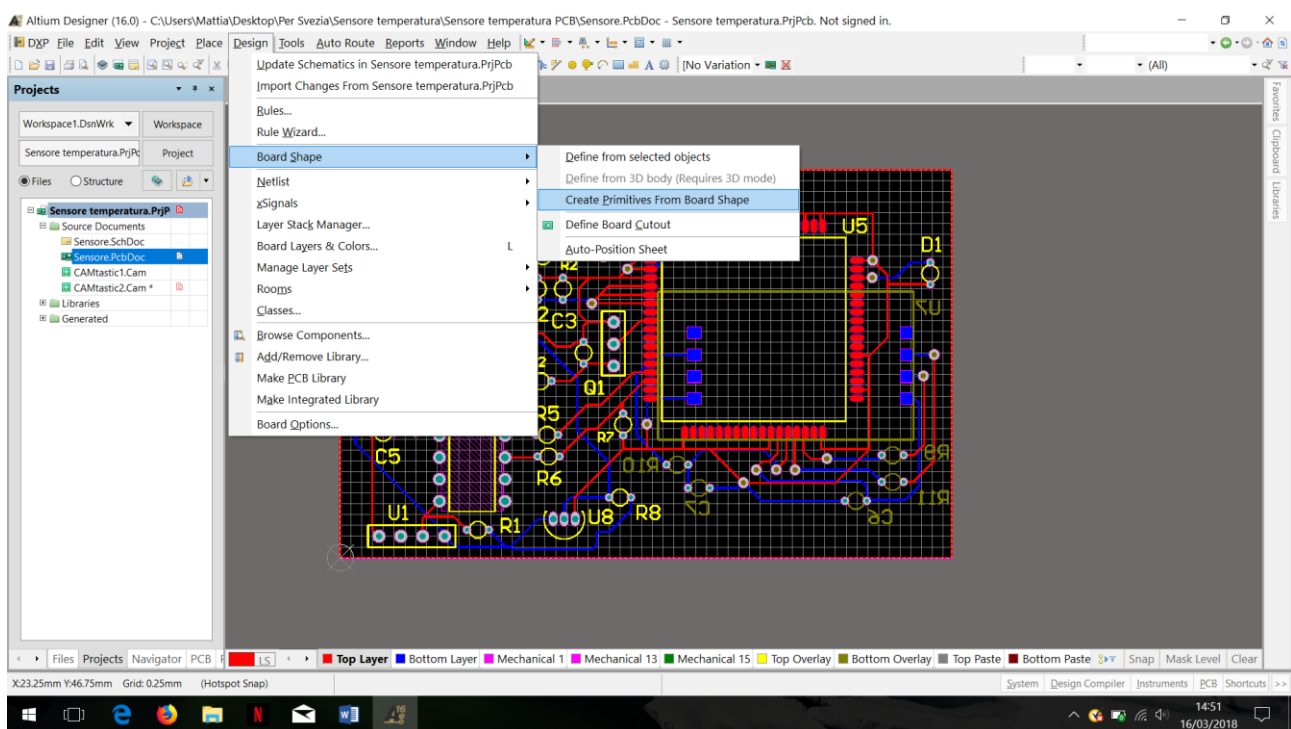
2. PRELIMINARY OPERATIONS

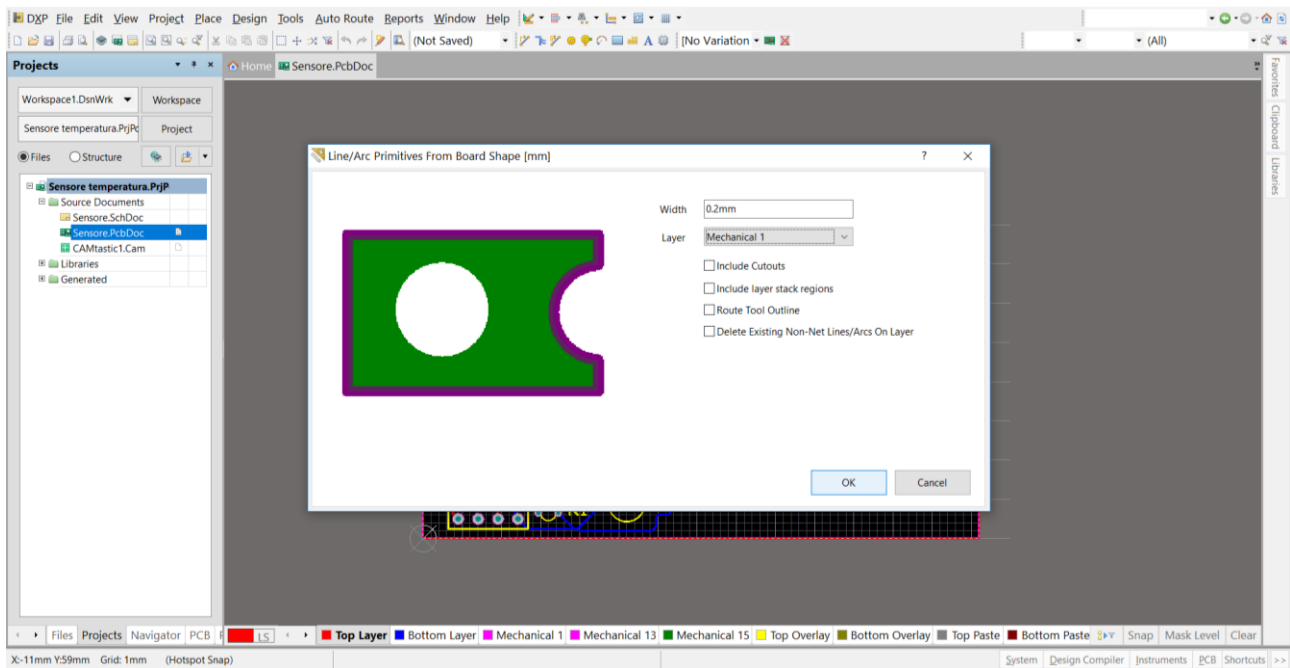
2.1 Outline

From your file *.PcbDoc* in *2D Layout Mode*, add the outline to your board by clicking on

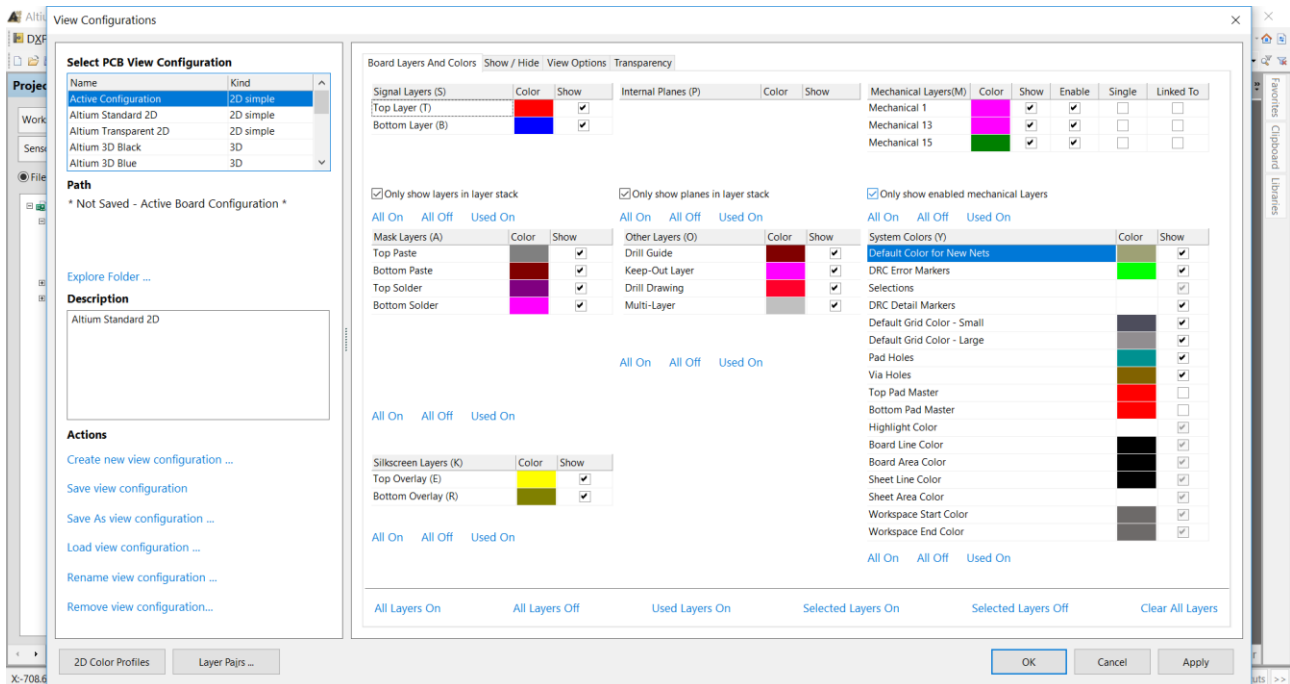
Design -> Board Shape -> Create Primitives from Board Shape

On the new window, you can choose the outline width and the layer where you want to create the outline. Choose *Mechanical 1* or another mechanical layer (be sure that the layer you choose is empty). MINImill will cut your board in the middle of the outline, so please choose a small width (for example 0.2mm).





If you want to add more mechanical layers, go back to your *.PcbDoc*, press the letter *L* of your keyboard, deselect *Only show enables mechanical layers* and then select *enable* for another layer.

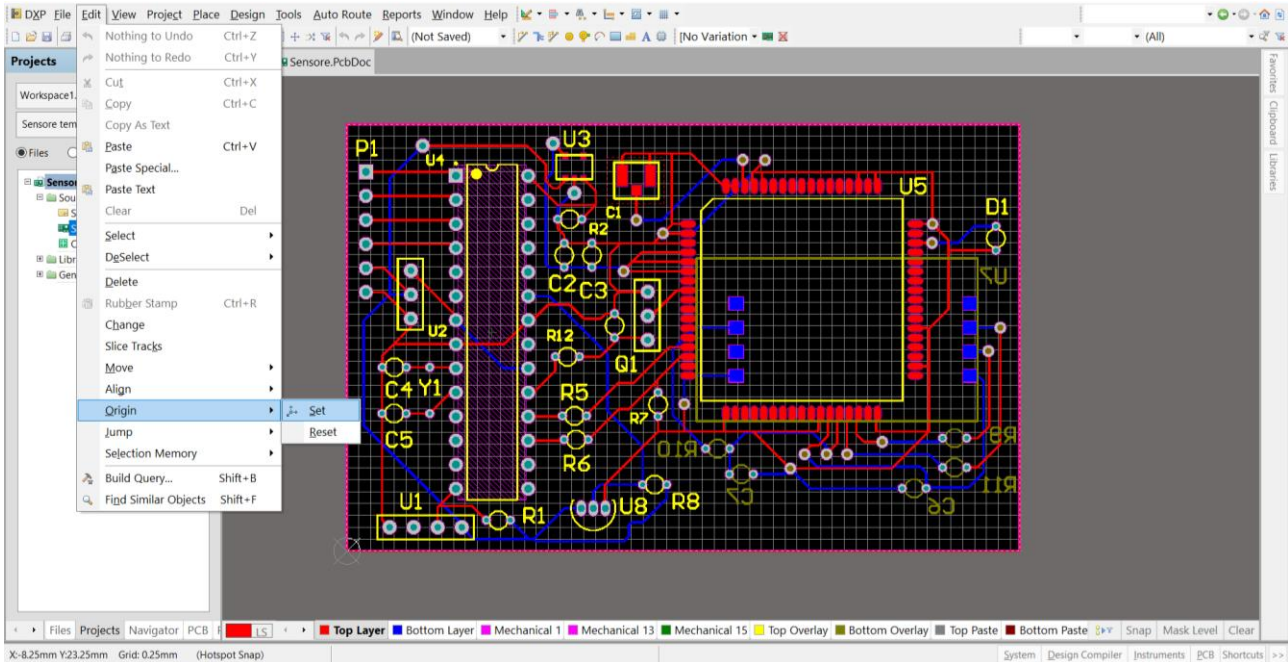


2.2 Origin

If you have not already done, set the origin in one of the corners of your PCB. Go to

Edit -> Origin -> Set

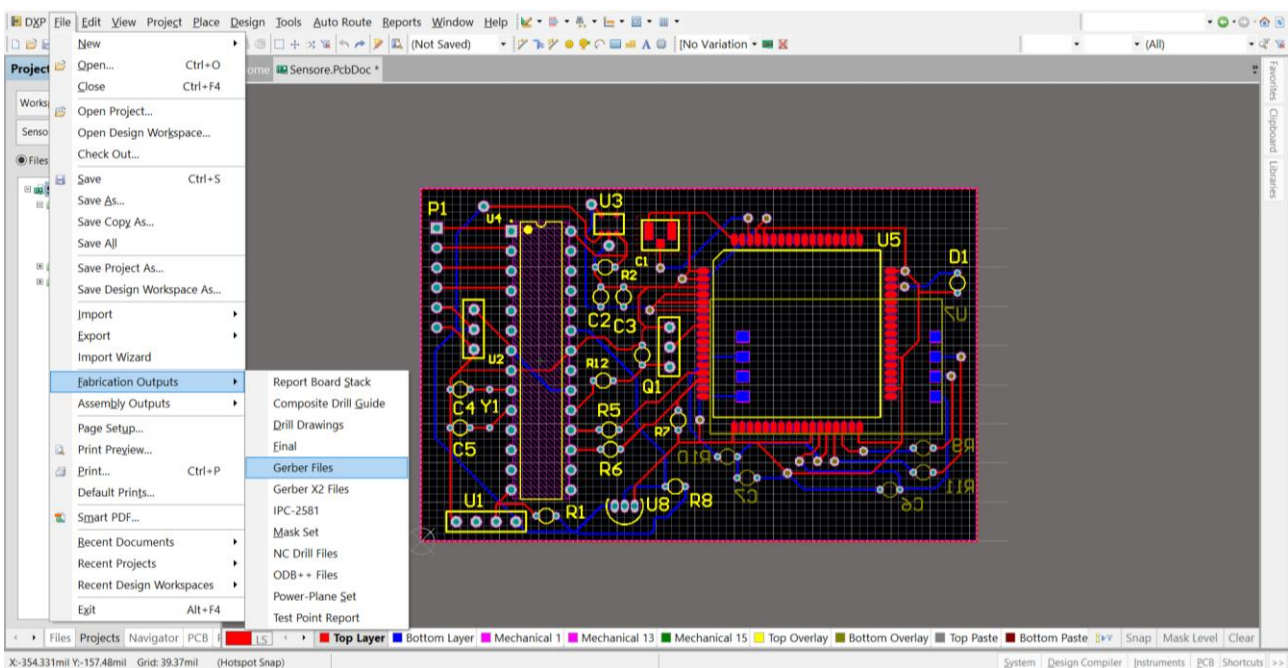
and select one corner.



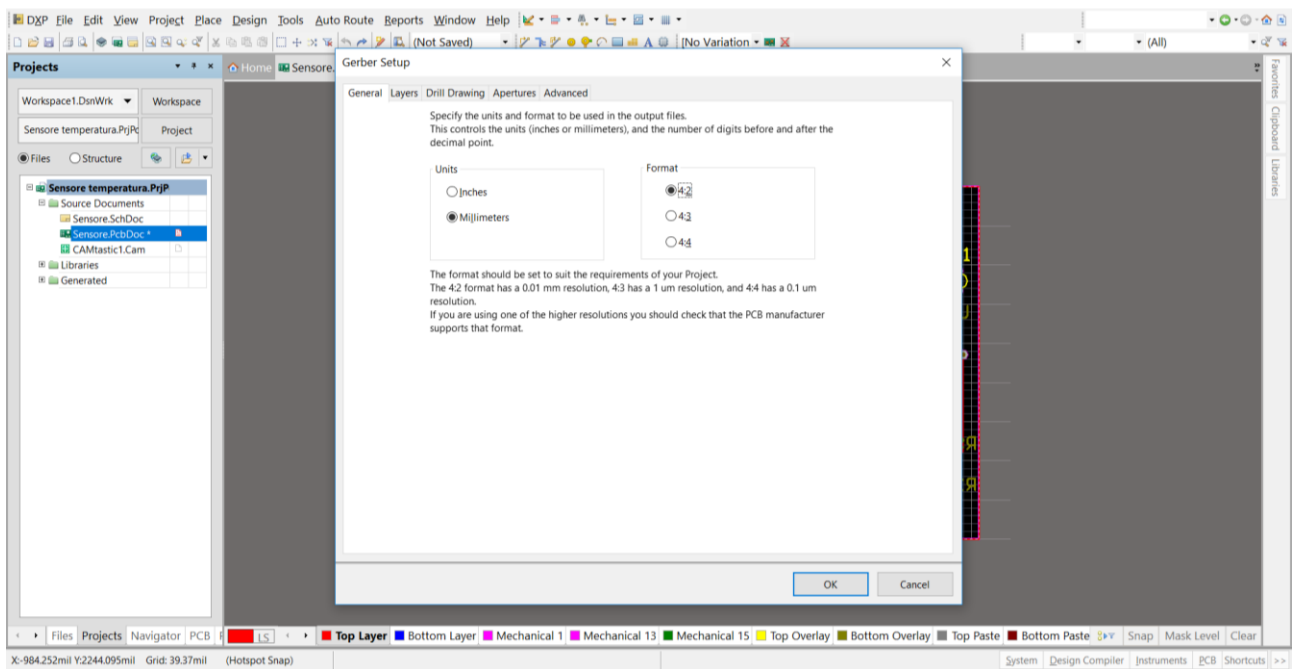
3. EXPORT GERBER FILES

You can now export your gerber files, bearing in mind that MINImill supports the extended version (RS-274-X). Go to

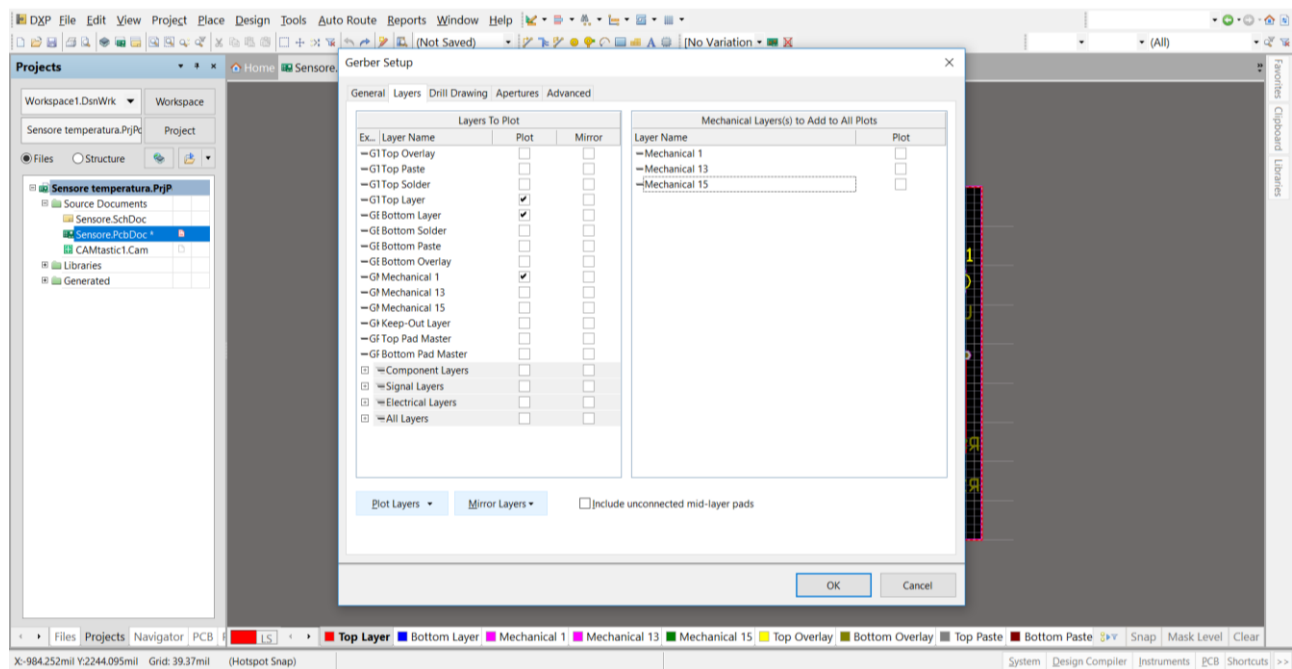
File -> Fabrication Outputs -> Gerber Files



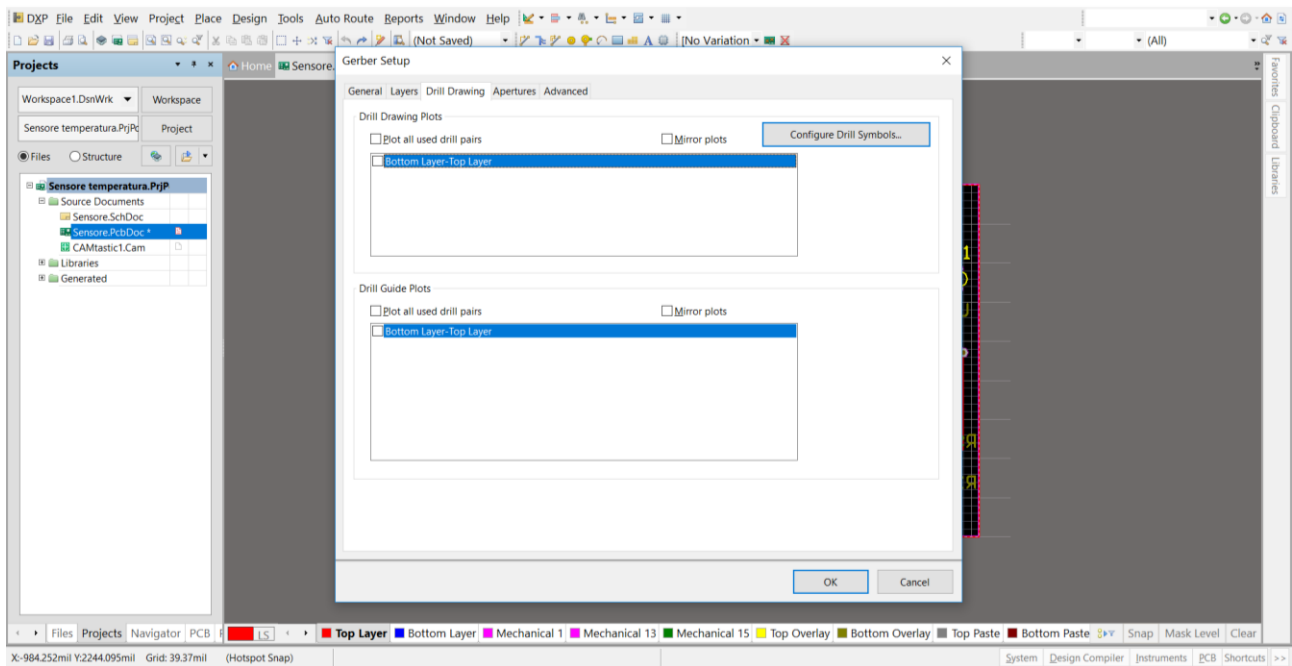
On the tab *General*, choose *Millimeters* and *4:2*.



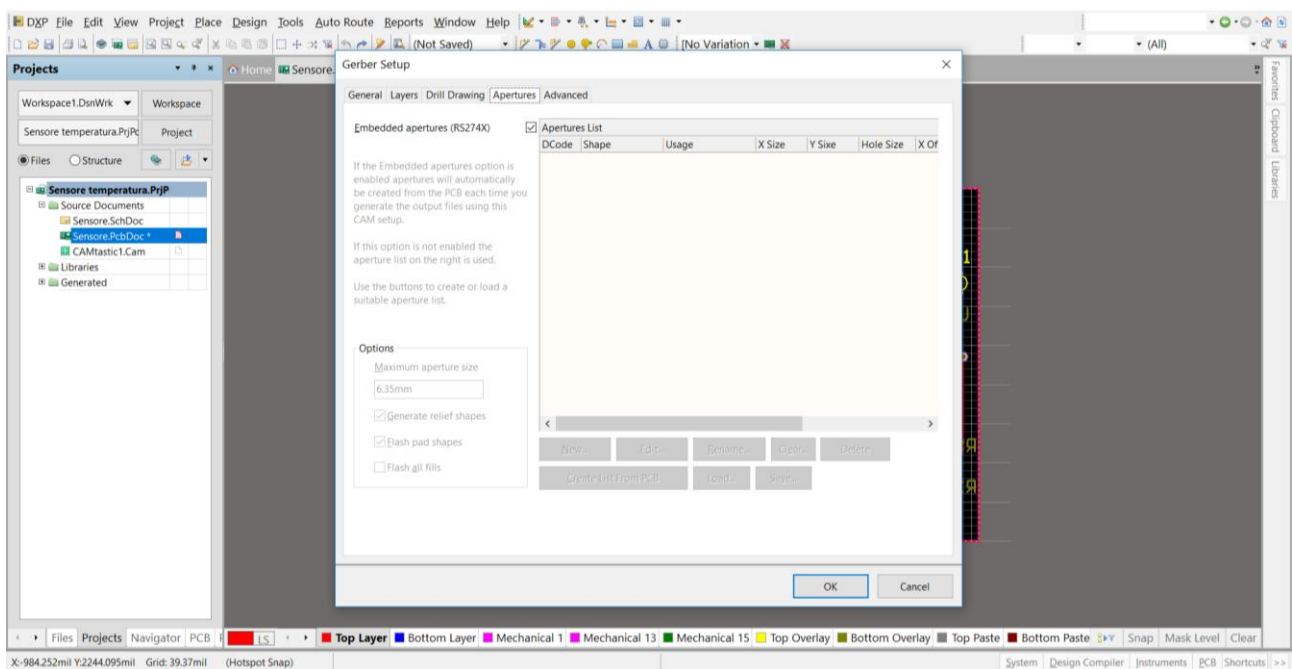
On *Layers*, choose only *Top Layer*, *Bottom layer* (if you have one) and *Mechanical 1* (or the other mechanical layer you previously chose for your outline).



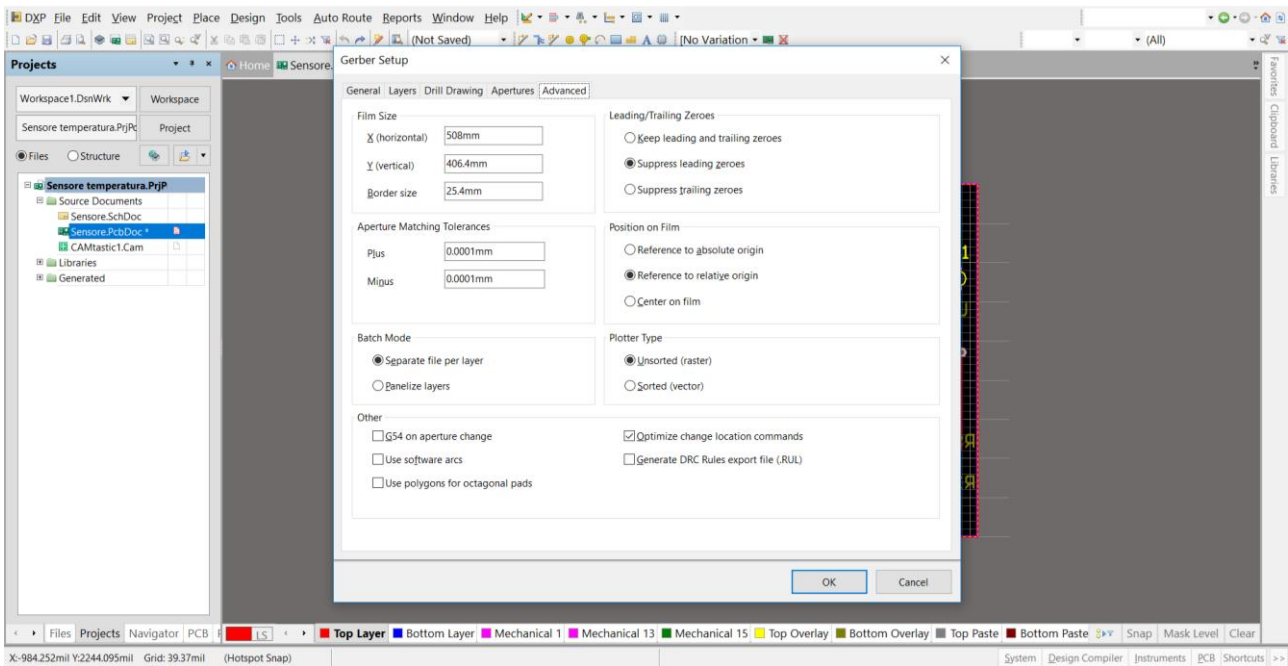
On *Drill Drawing*, be sure that nothing is selected.



On *Apertures* select *Apertures List*.

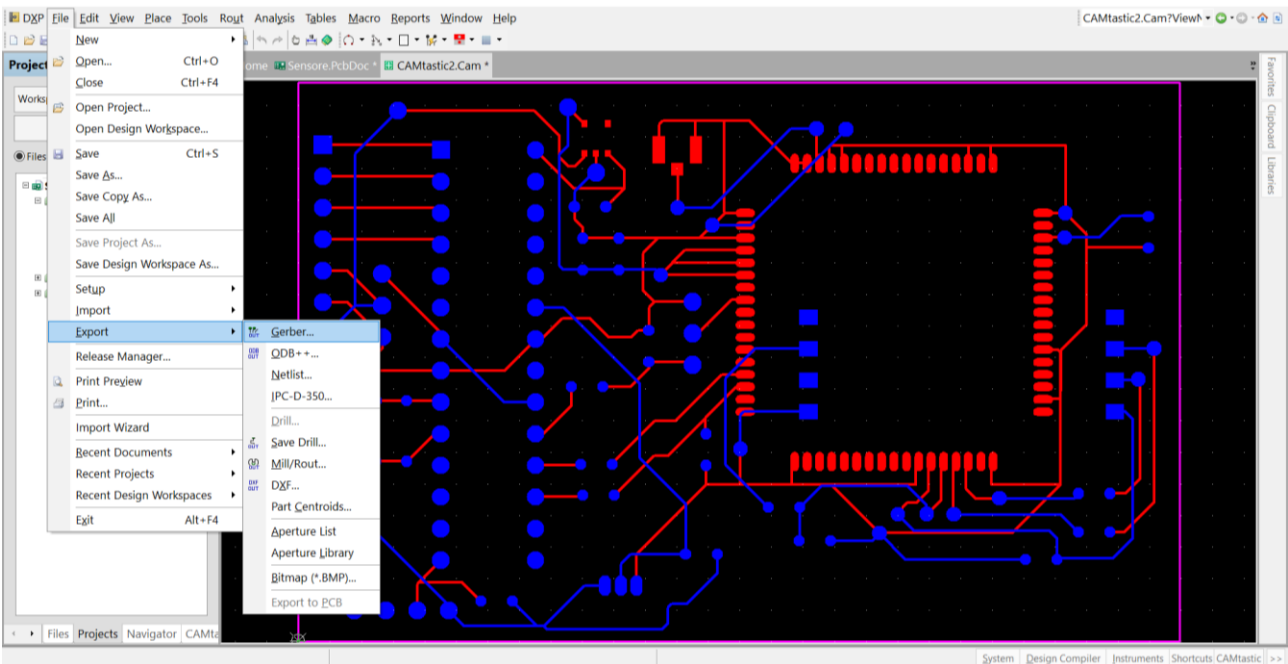


On *Advanced*, leave everything as default. Please only be sure that *Reference to relative origin* is selected.

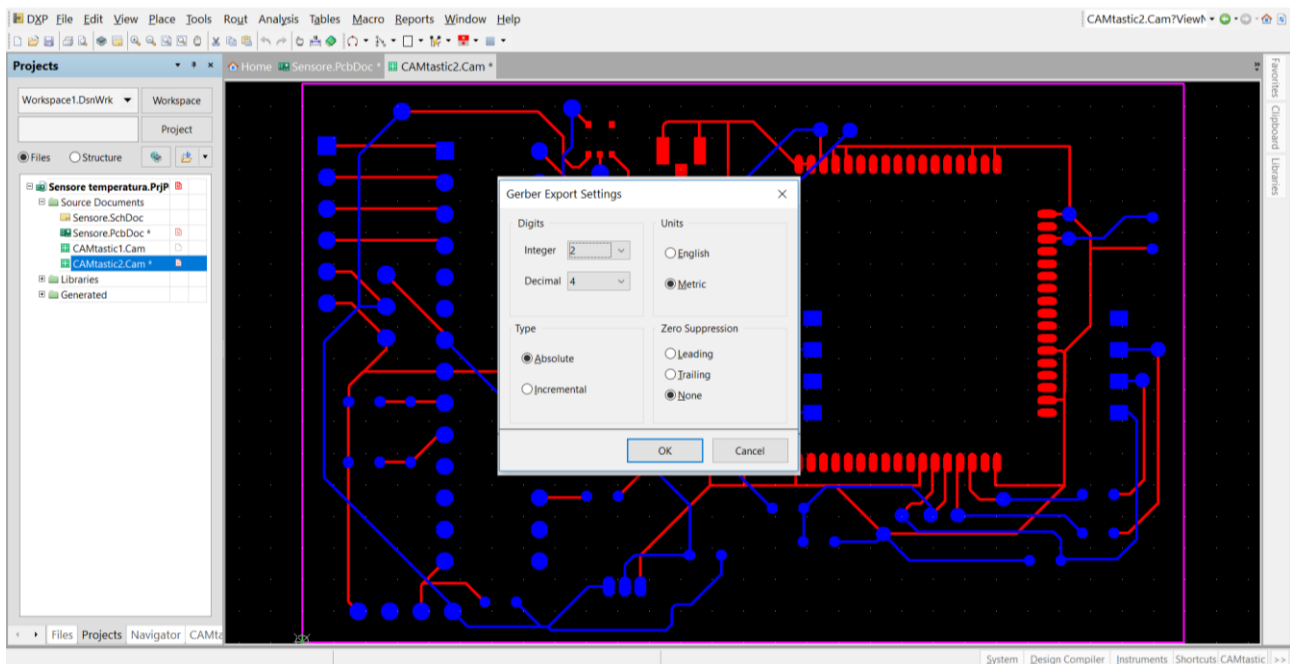
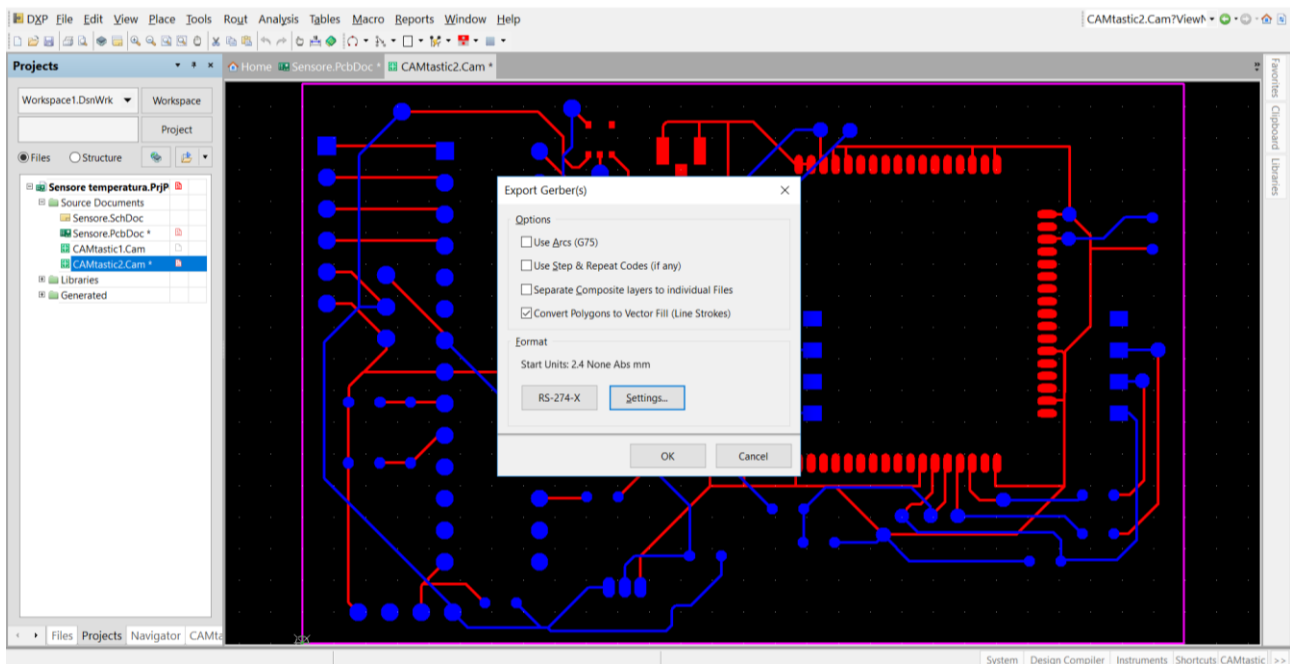


Then press OK. On the new file opened automatically (.Cam), go to

File -> Export -> Gerber



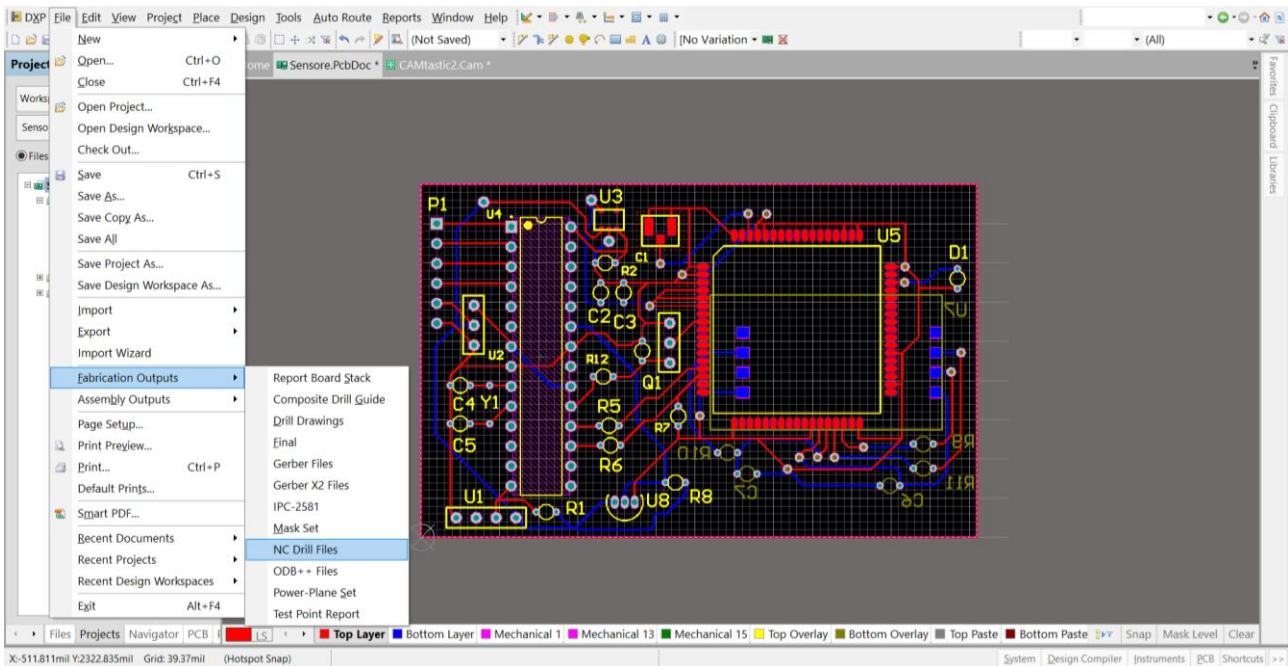
On the new window, select only *Convert Polygons to Vector Fill (Line Strokes)* and choose as *Format 2.4 None Abs mm* by setting the values on *Settings*...



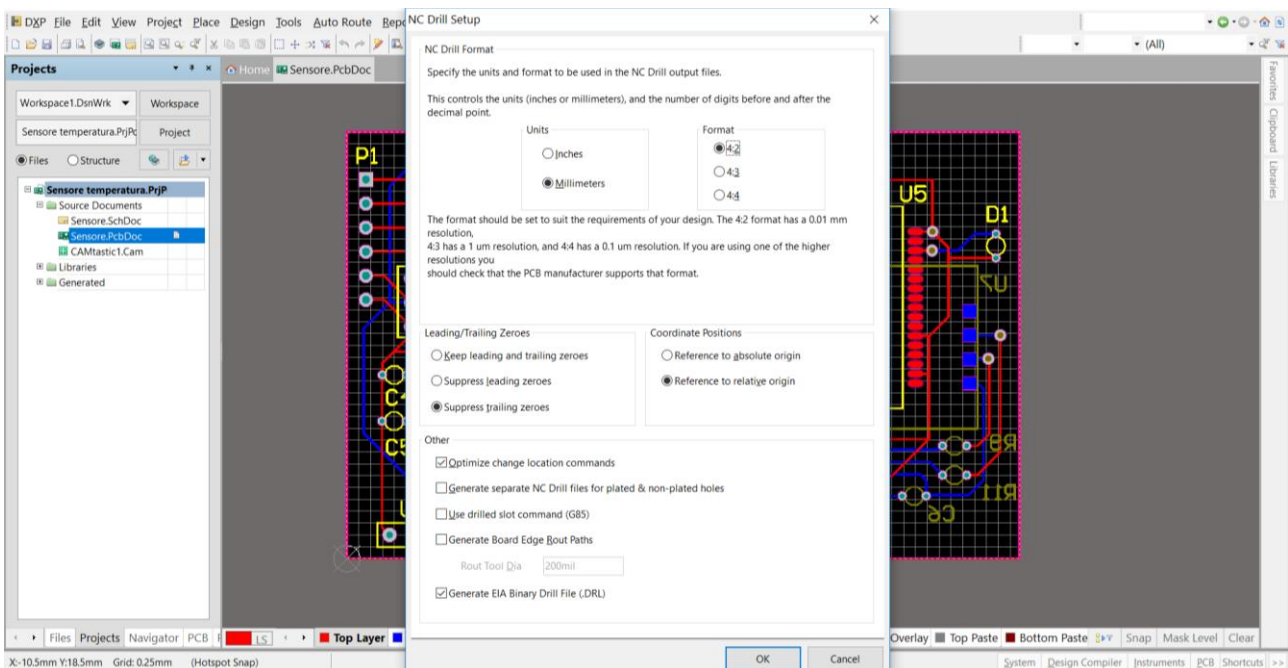
Press OK and choose the destination folder where you want to save your files. You should now have 3 different files *.gtl*, *.gbl* (if you have a bottom layer) and *.gm1* (or gm plus the number of your mechanical layer).

Now go back to your .PcbDoc and select

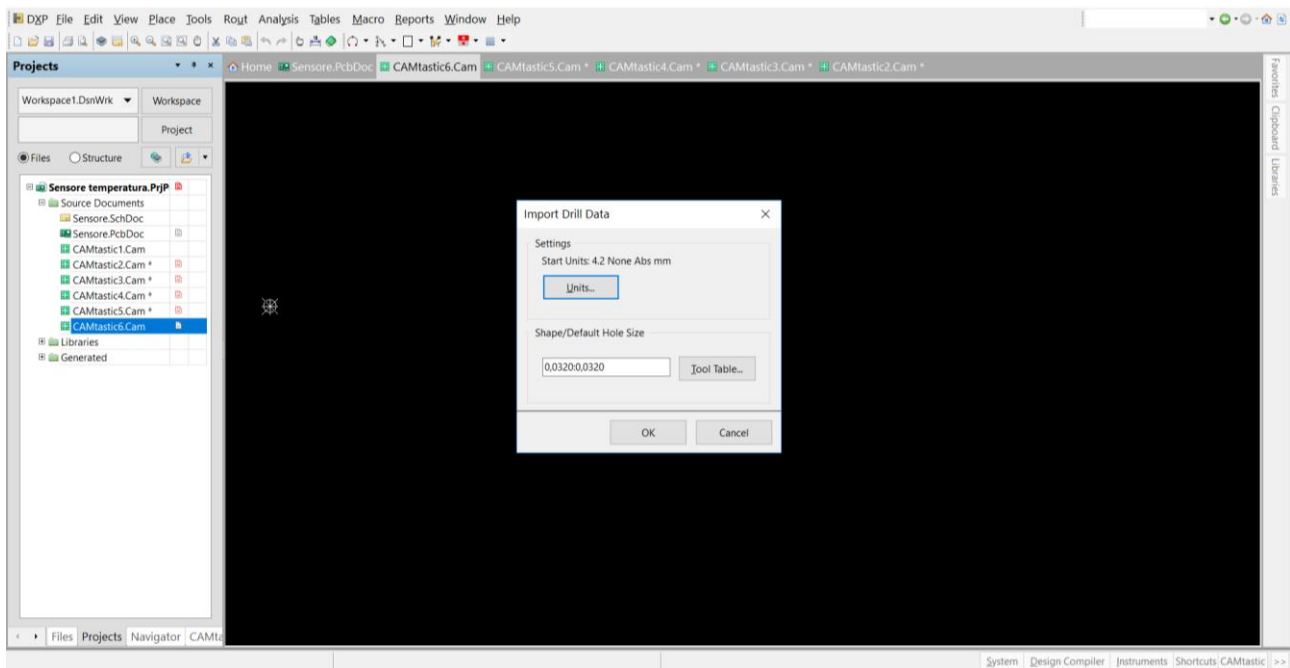
File -> Fabrication Outputs -> NC Drill Files



On the new window, select *Millimeters* and *4:2*. Choose *Reference to relative origin* and select *Generate EIA Binary Drill File (.DRL)*.

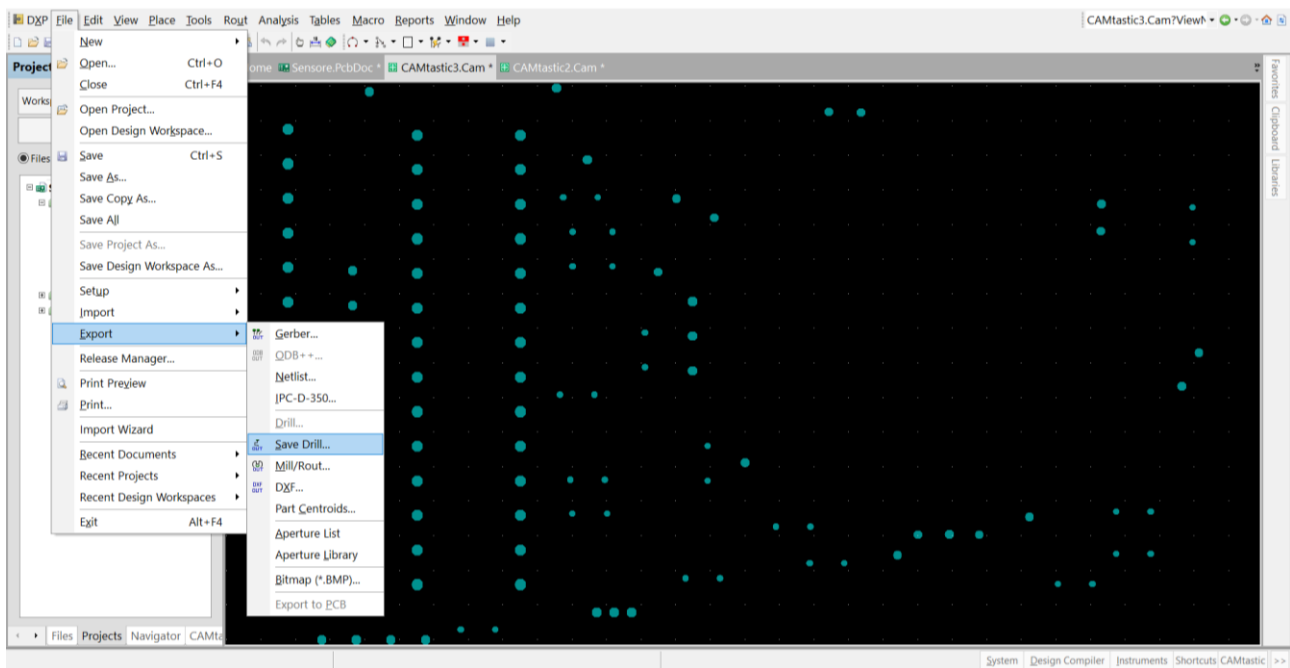


Press OK. On the new window, select *4.2 None Abs mm*, as you did previously. Leave *Shape/Default Hole Size* as default. Press OK.

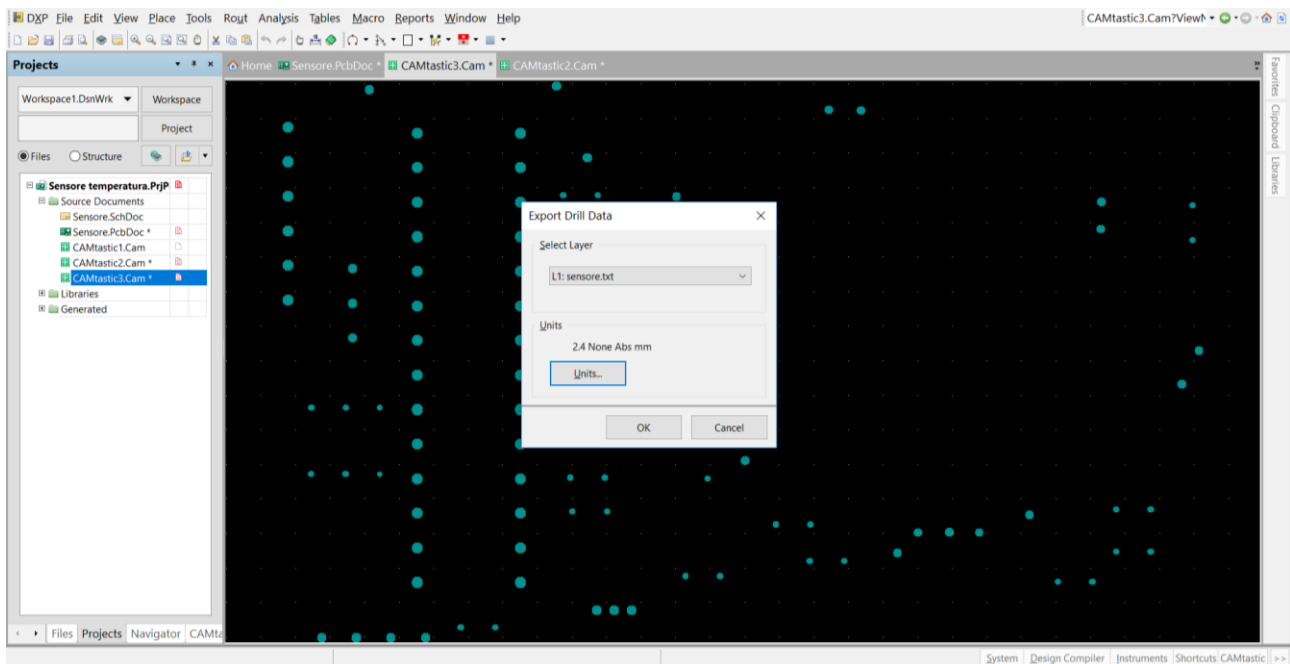


On the new file opened automatically, go to

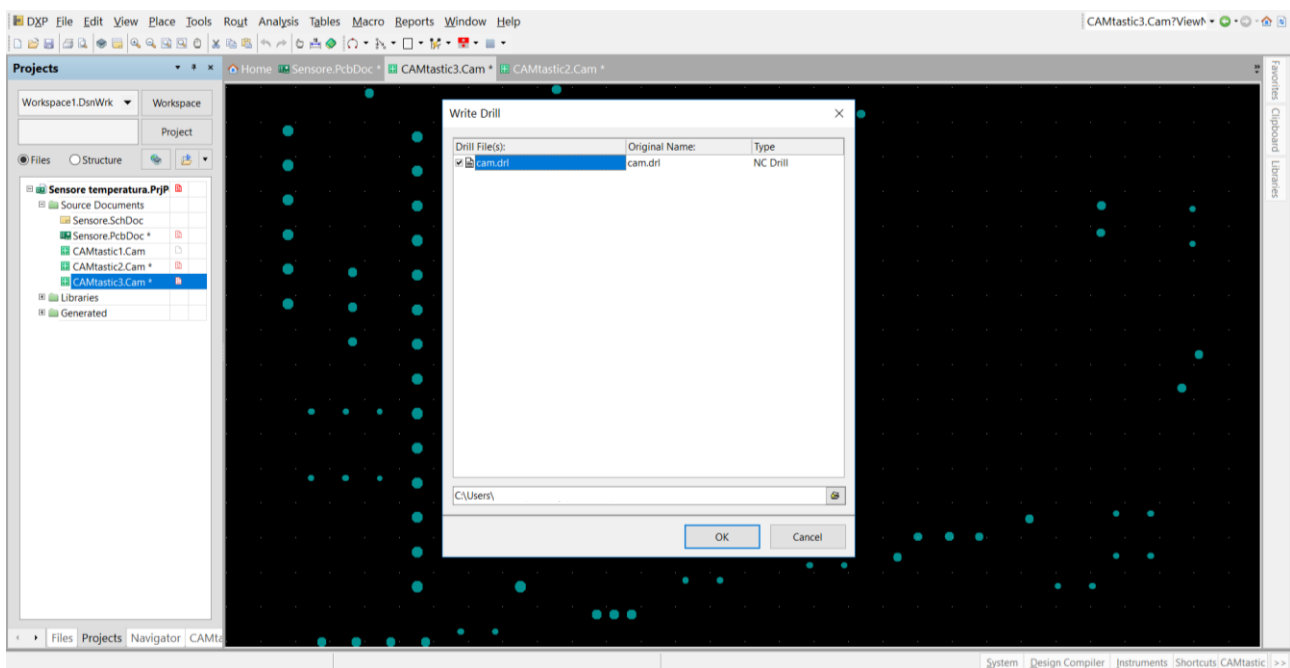
File -> Export -> Save Drill...



Select your layer and 2.4 *None Abs mm*. Then press OK.



Save now your file.



4. MILLING

Open the MINImill software, choose PCB on the bottom panel and load your files as follow:

.gtl as top layout

.gbl as bottom layout

.drl as drill file

.gm1 as outline.

Then start milling and enjoy your PCB!

