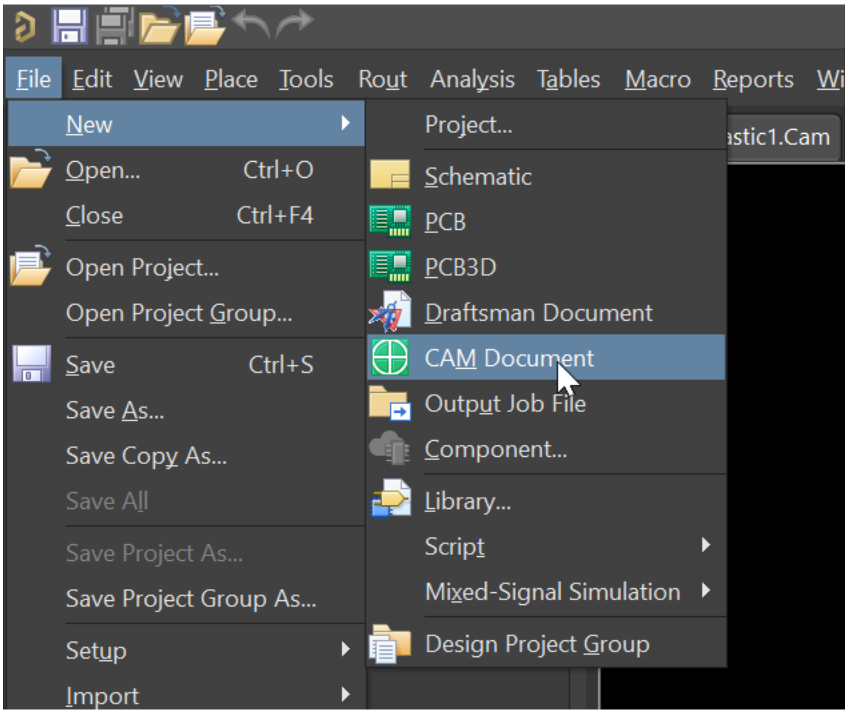
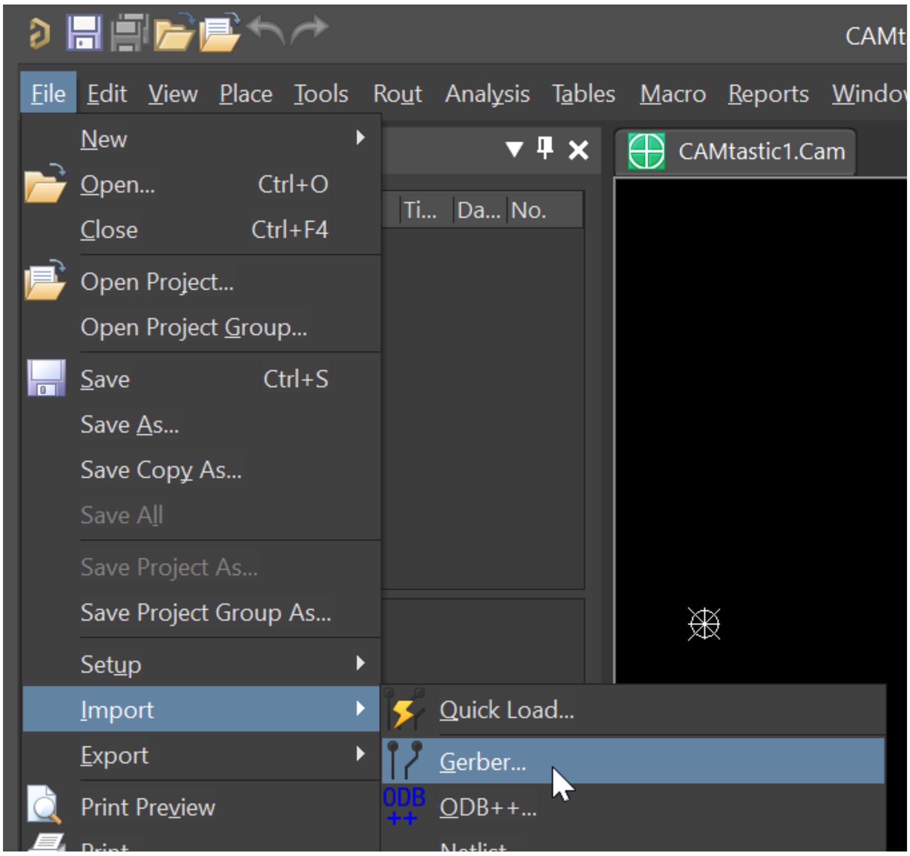
1. Process of importing Gerbers to Altium Designer

1.1 Importing Gerbers

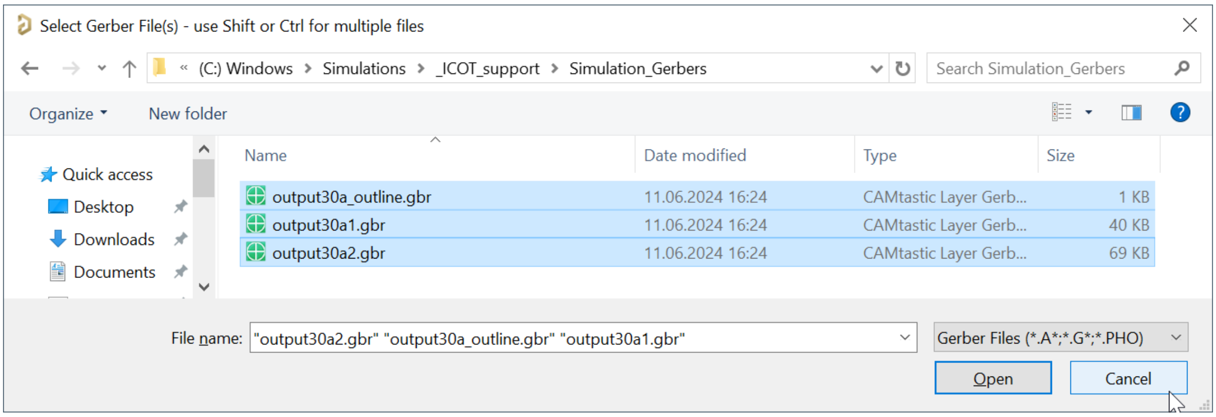
1.1.1. Create new CAM Document in Altium Designer



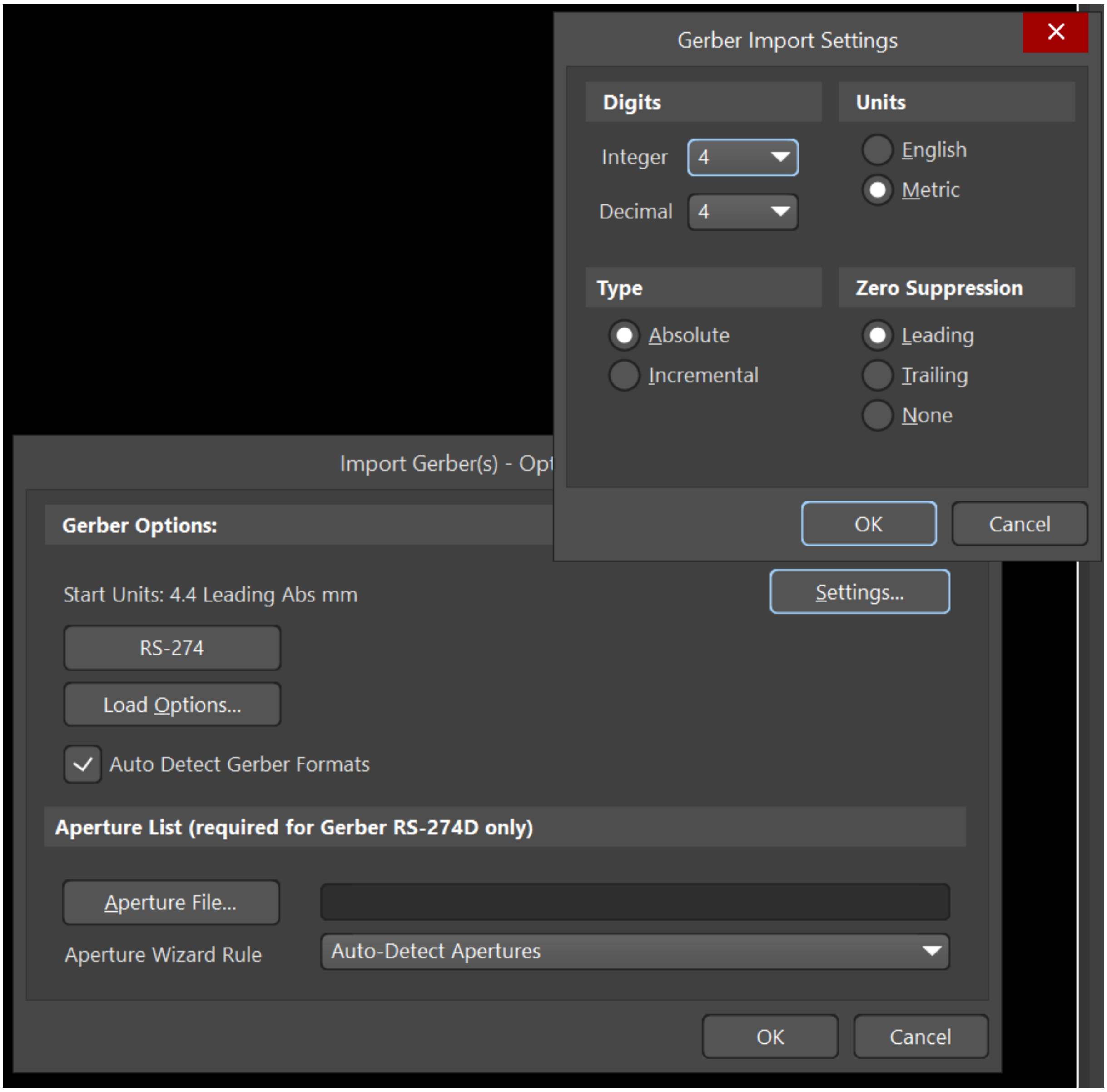
1.1.2. Import Gerber Files



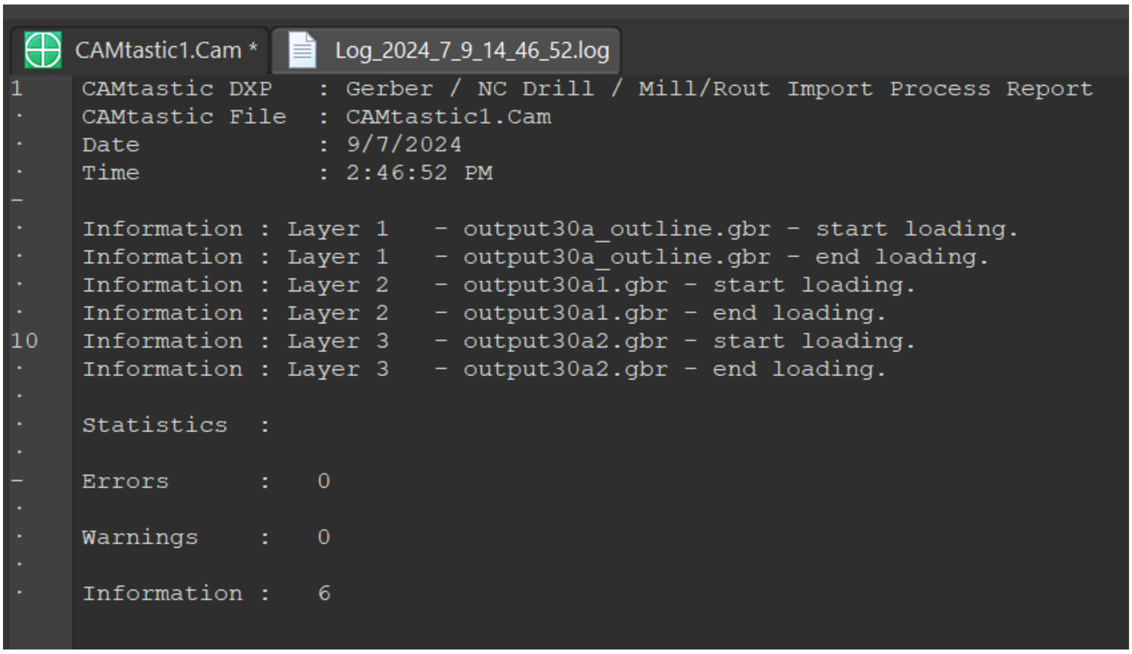
1.1.3. Select all the Gerber data in the right location, then click open



1.1.4. Change Gerber Import settings, and/or press OK

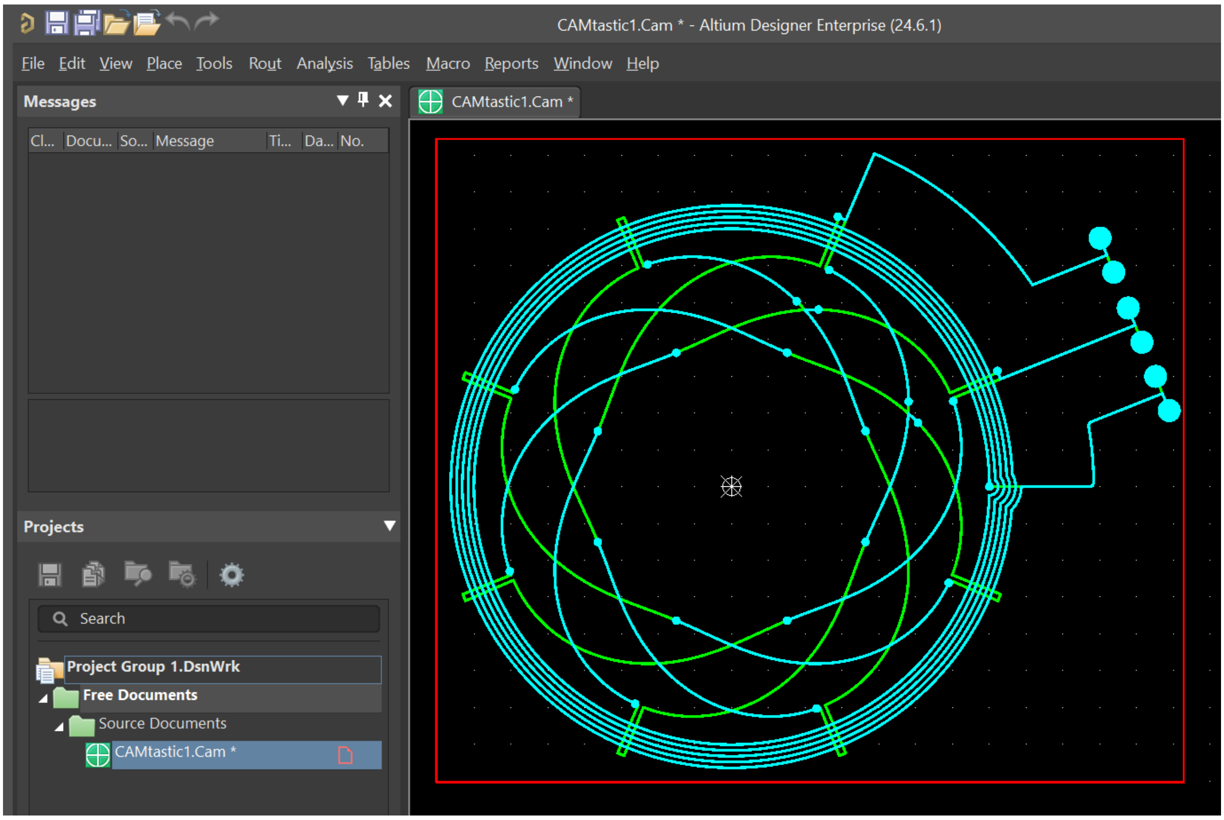


1.1.5. Warnings and Errors should be 0, then close this log file



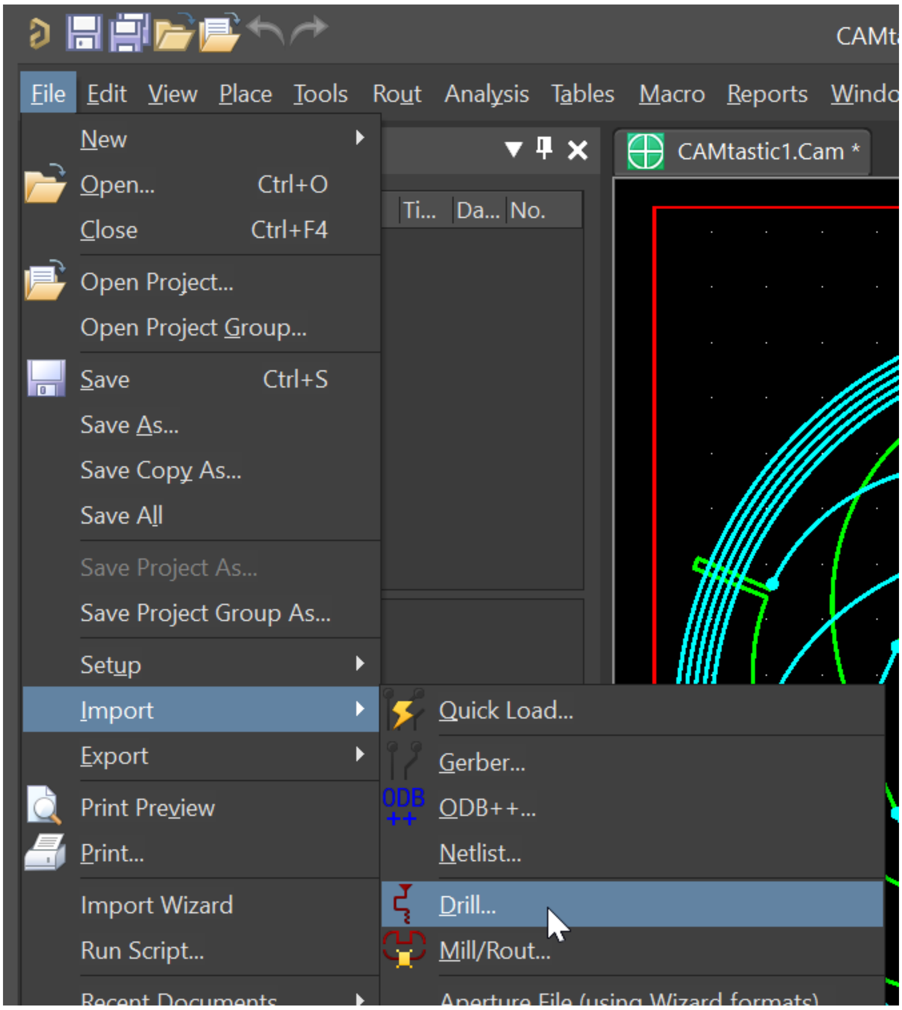

```
CAMtastic DXP    : Gerber / NC Drill / Mill/Rout Import Process Report
CAMtastic File   : CAMtastic1.Cam
Date             : 9/7/2024
Time             : 2:46:52 PM

Information : Layer 1  - output30a_outline.gbr - start loading.
Information : Layer 1  - output30a_outline.gbr - end loading.
Information : Layer 2  - output30a1.gbr - start loading.
Information : Layer 2  - output30a1.gbr - end loading.
Information : Layer 3  - output30a2.gbr - start loading.
Information : Layer 3  - output30a2.gbr - end loading.

Statistics  :

Errors      :   0

Warnings    :   0

Information :   6
```

1.1.6. Gerber Files are successfully imported

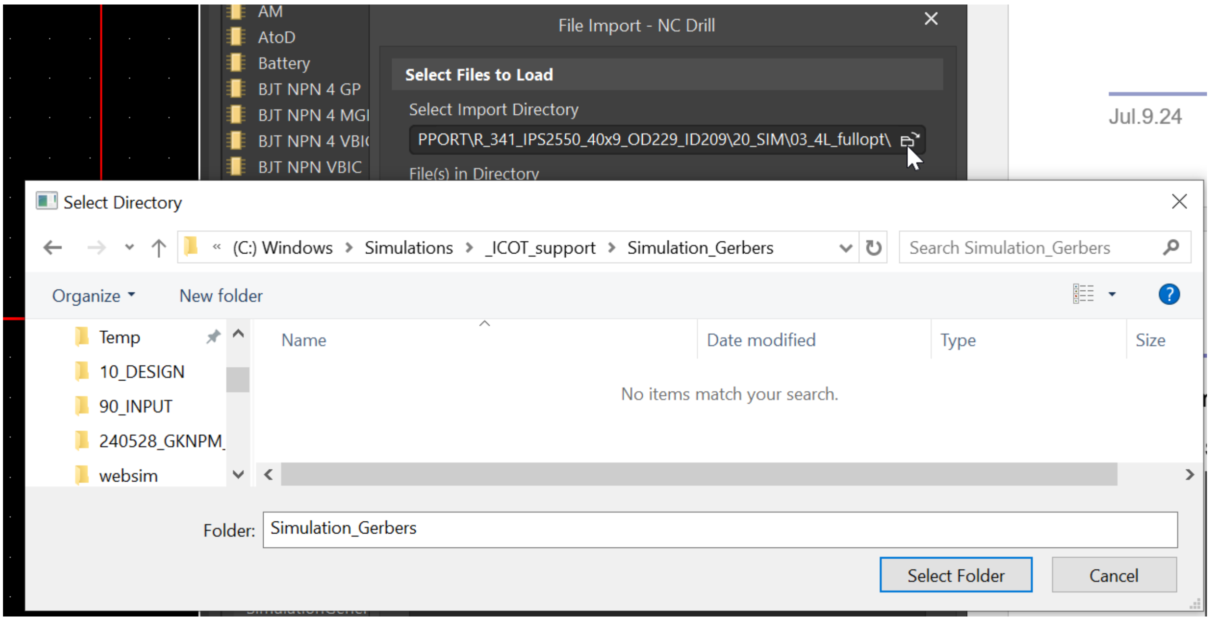


1.2 Import Drill File

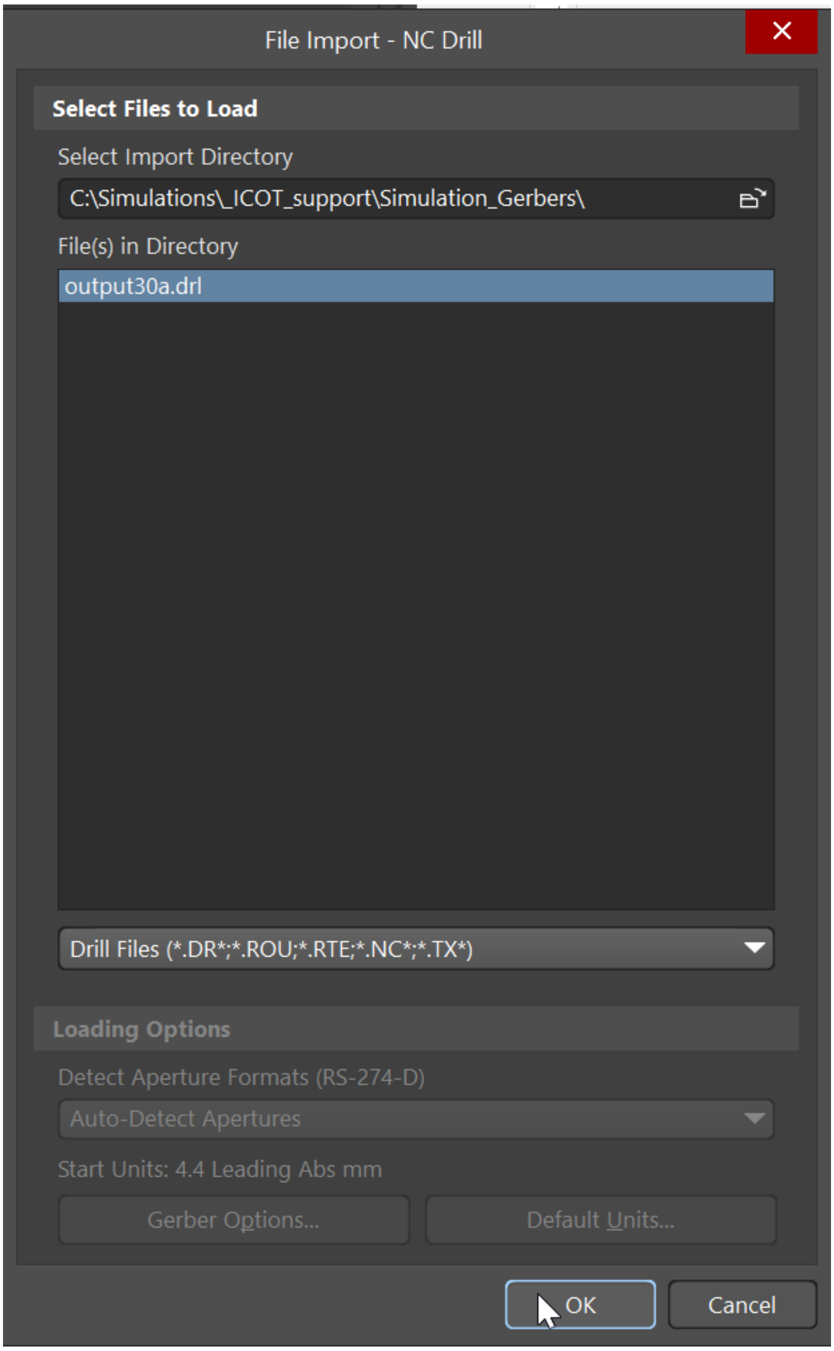
1.2.1. Select the Drill importer



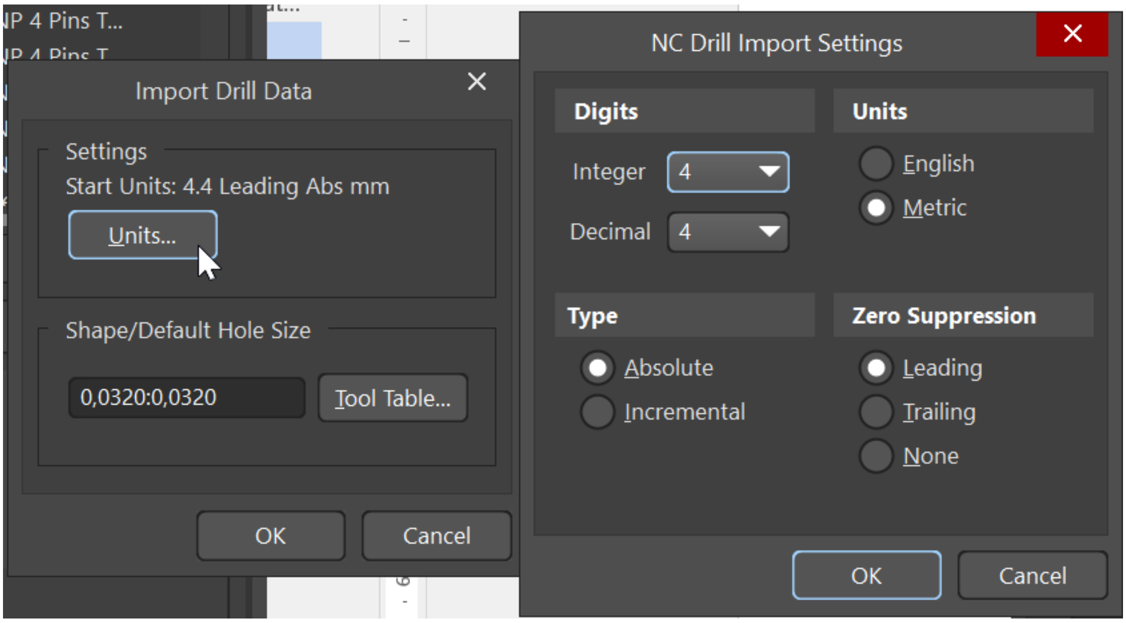
1.2.2. Select the right folder with the Drill that belongs to the Gerbers



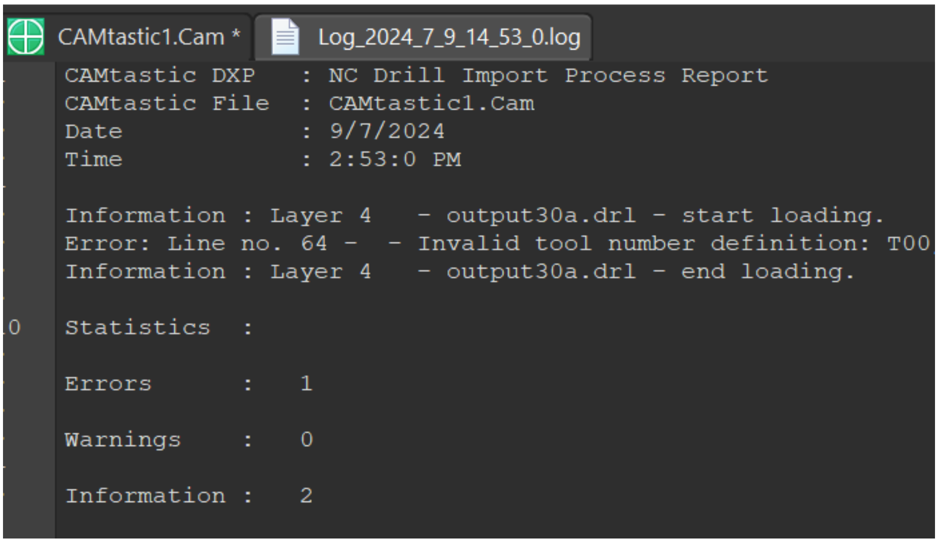
1.2.3. Select the Drill File, then press OK



1.2.4. Change Units if necessary, otherwise press OK

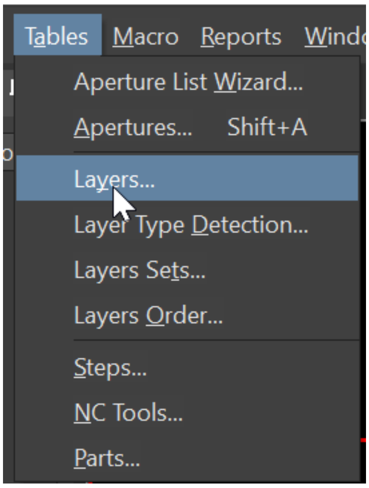


1.2.5. Errors and Warnings should be 0, then close the log file

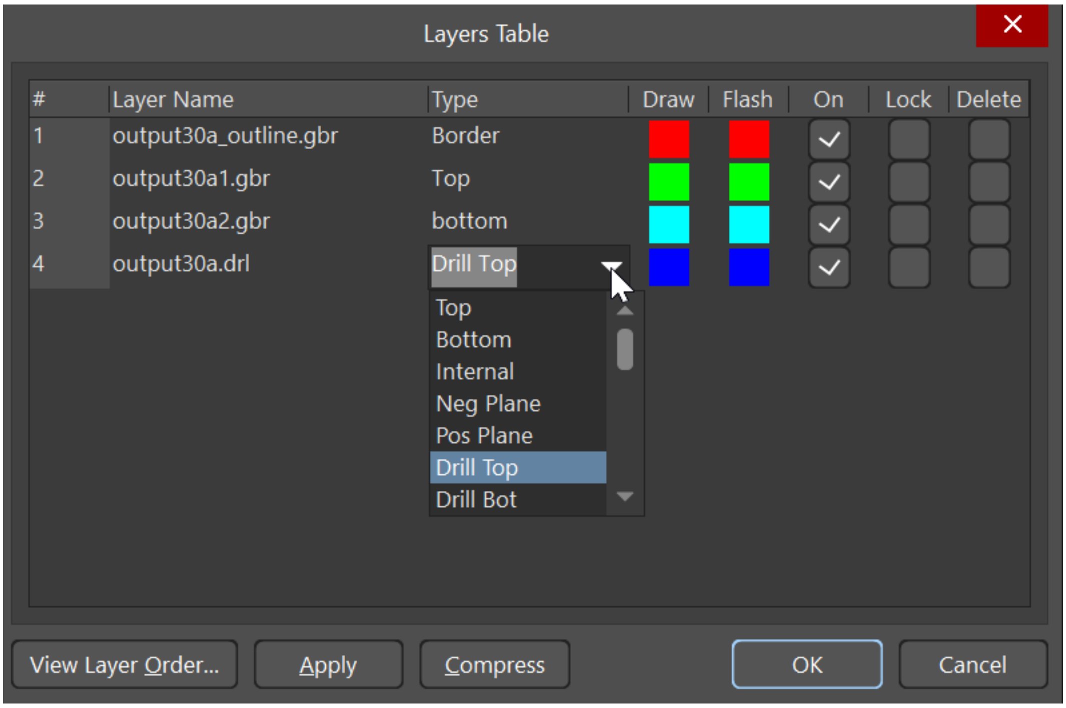


2. Adjusting the CAM File

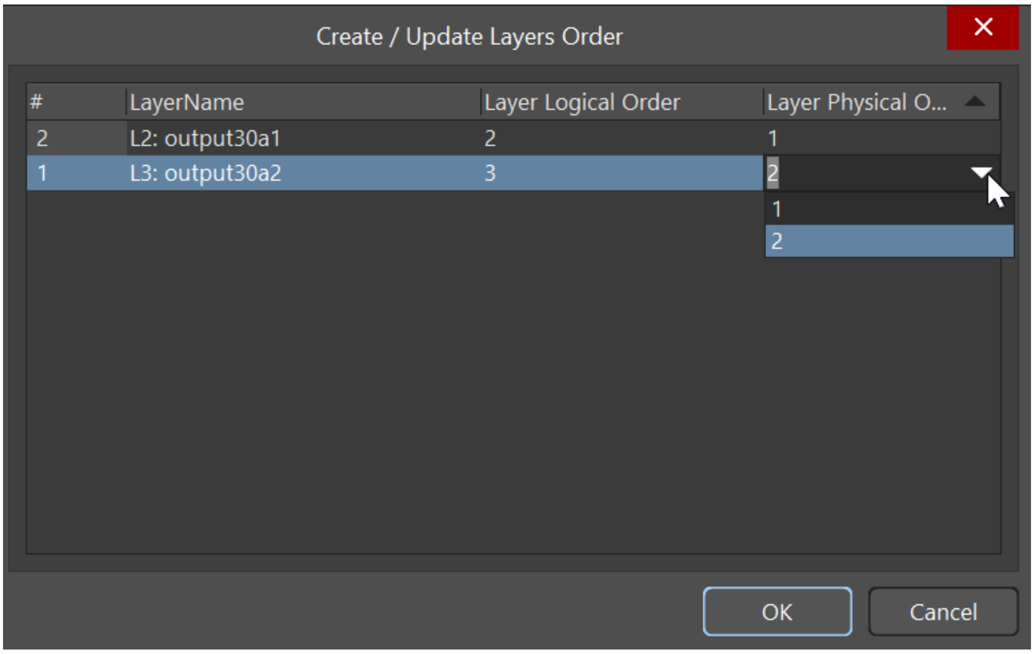
2.1 Setting the Layers



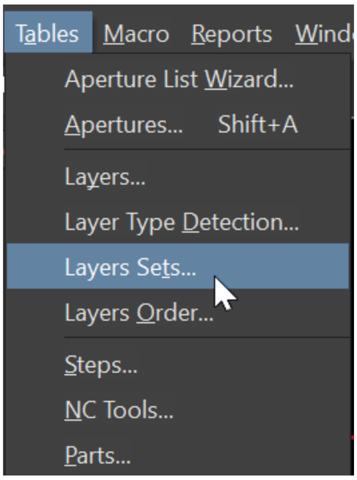
2.1.1. Set the correct layers then press OK



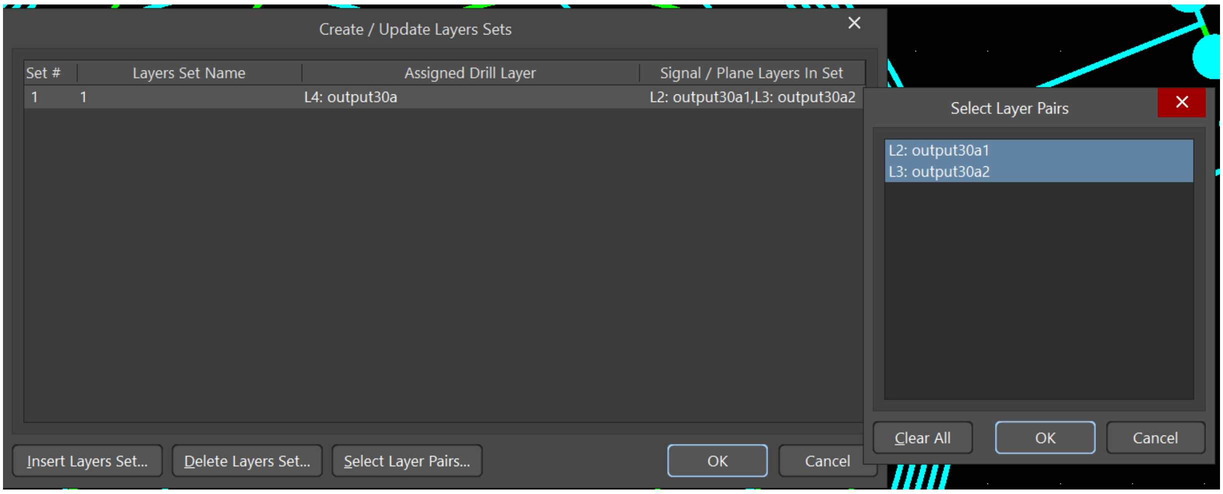
2.1.2. Set the correct layers order, then press OK



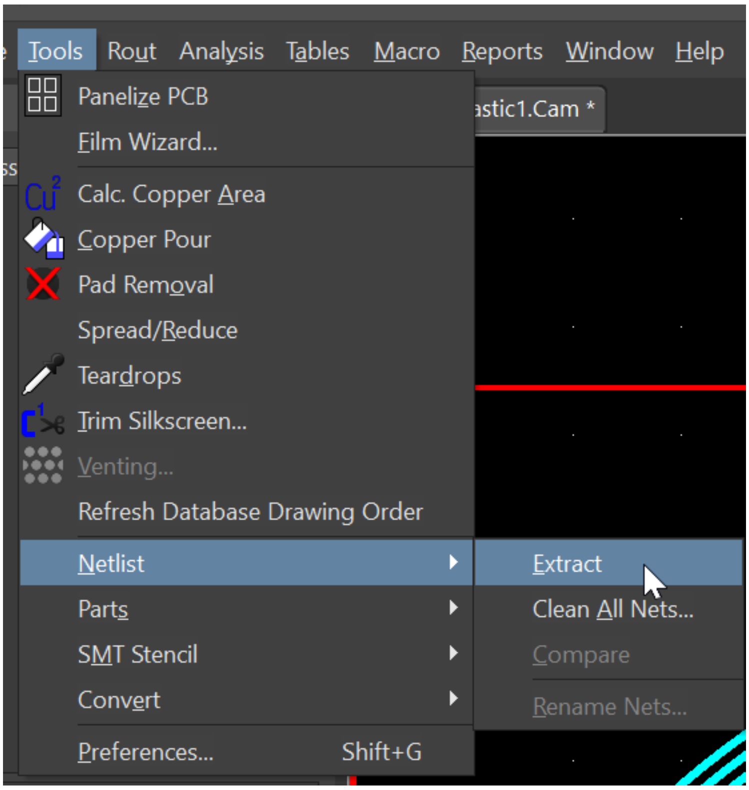
2.2 Setting the layer sets



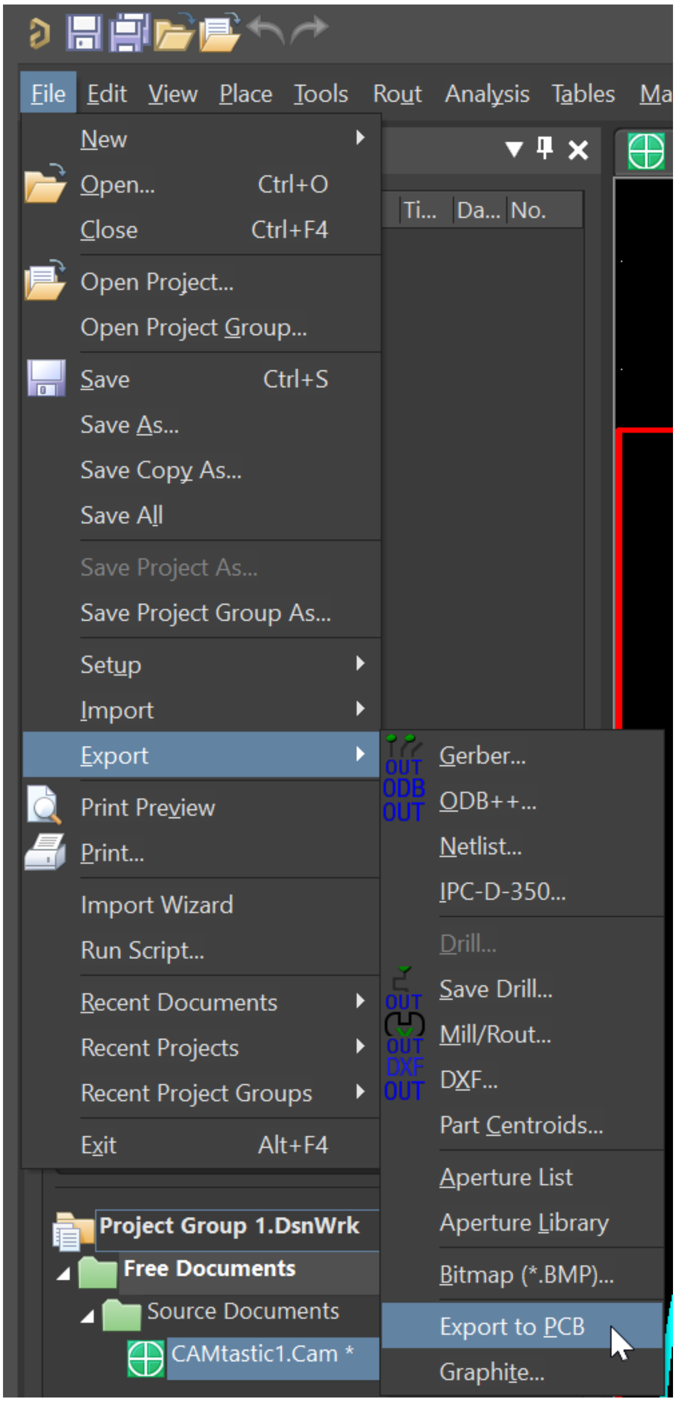
2.2.1. Add Drill and Plane Layers in set, then press OK



2.3 Extract the Netlist



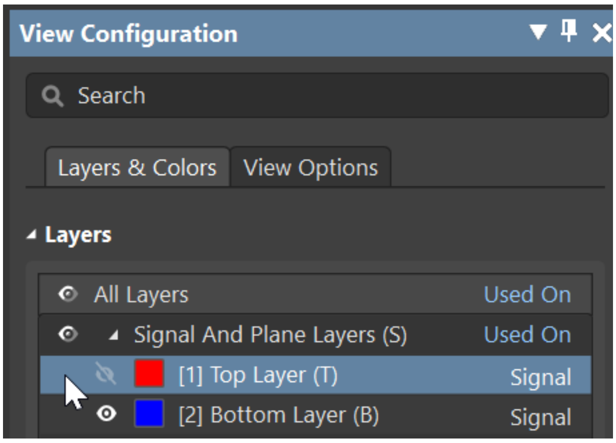
2.4 Export to PCB



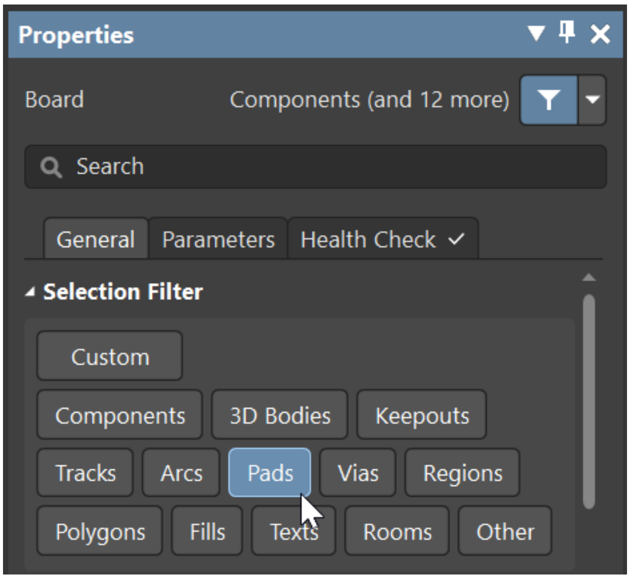
3. Adjusting the PCB

3.1 Deleting the excess pads that were imported by default

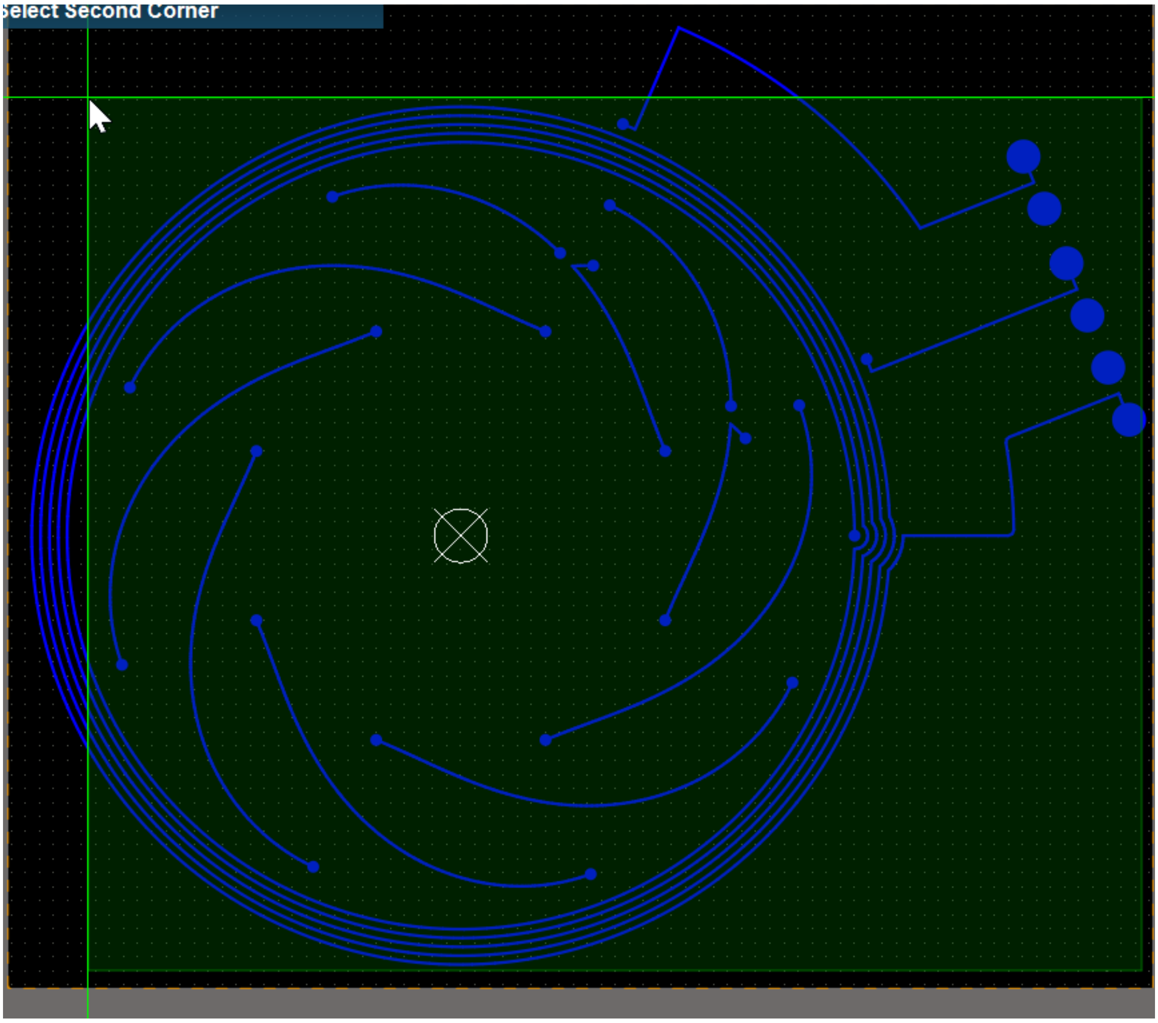
3.1.1. Keep only one layer on visible (Top Layer is turned off in the example)



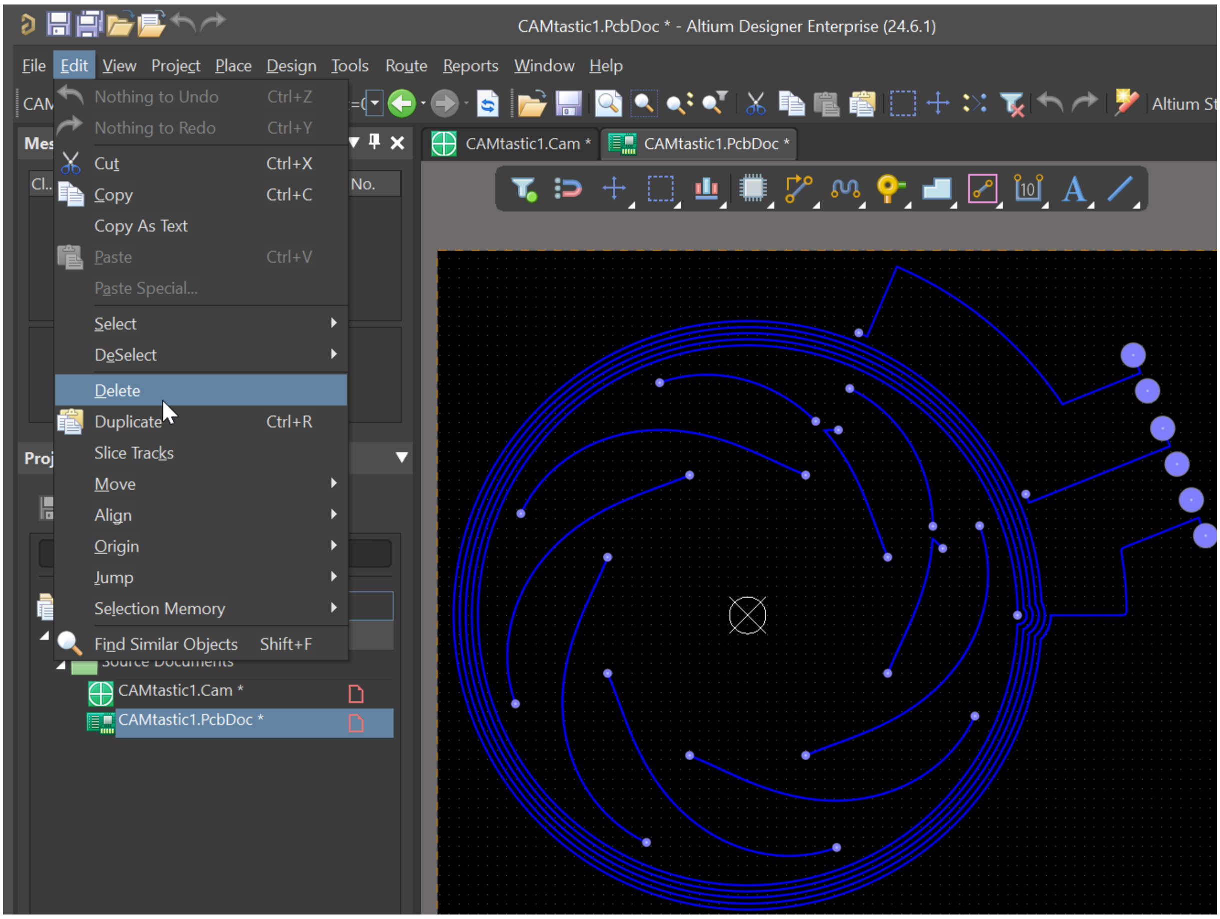
3.1.2. In Properties menu only select the Pad as selection filter



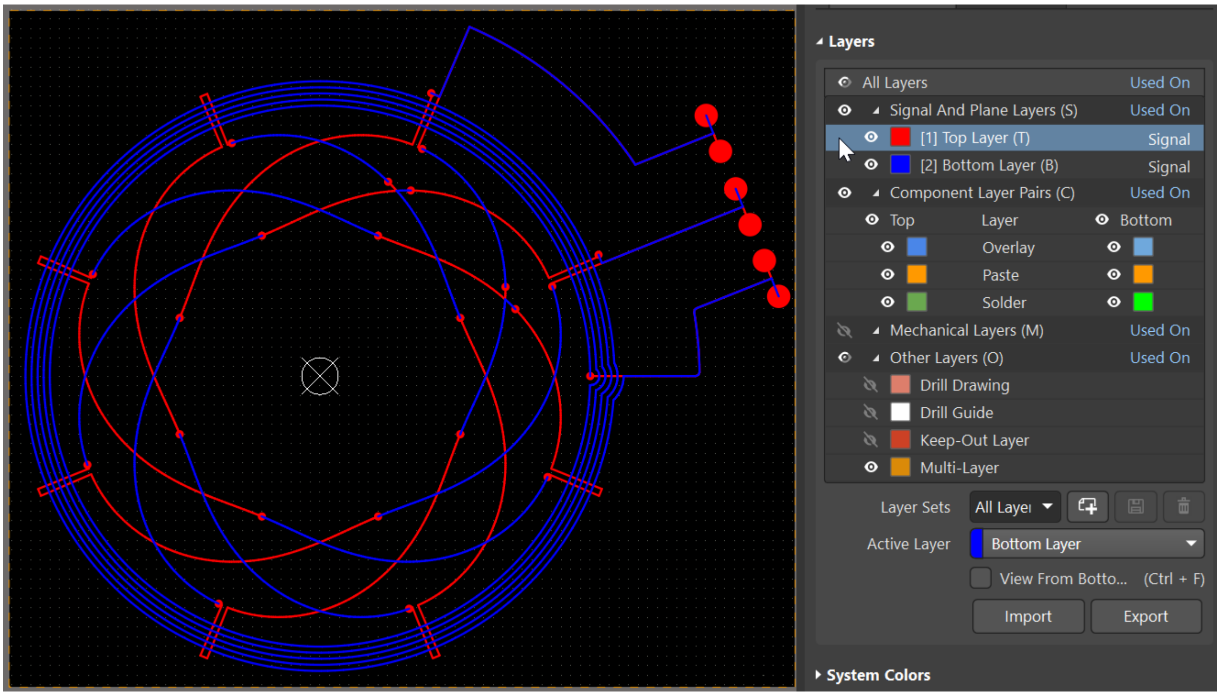
3.1.3. Select the entire imported sensor



3.1.4. Delete the selected pads

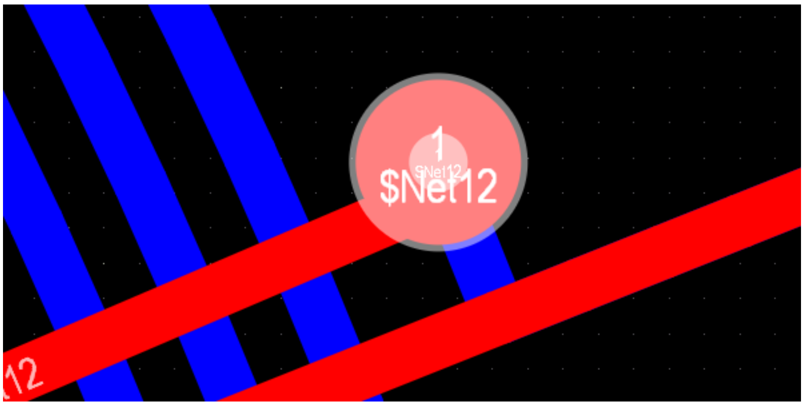


3.1.5. Turn the TOP layer back on

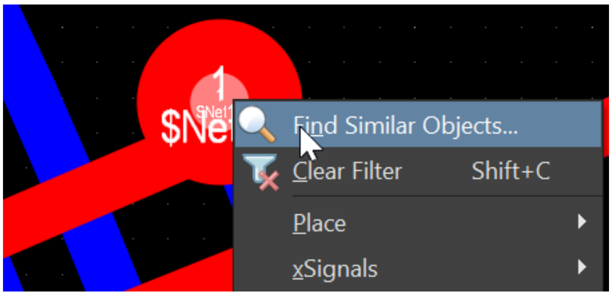


3.1.6. For some strange reason there are always 2 pads

imported on each layer. A correct sized, like in our case 0.6mm diameter and a smaller one with 0.2mm diameter. The latter needs to be deleted.

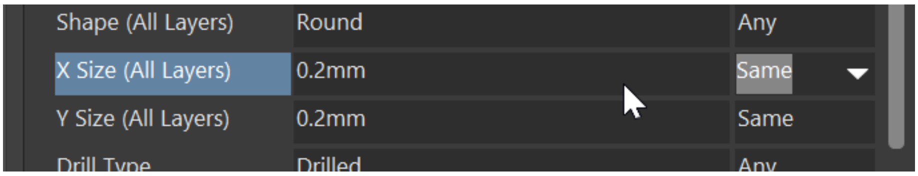


3.1.7. Select one smaller pad, then right click and select Find Similar Objects

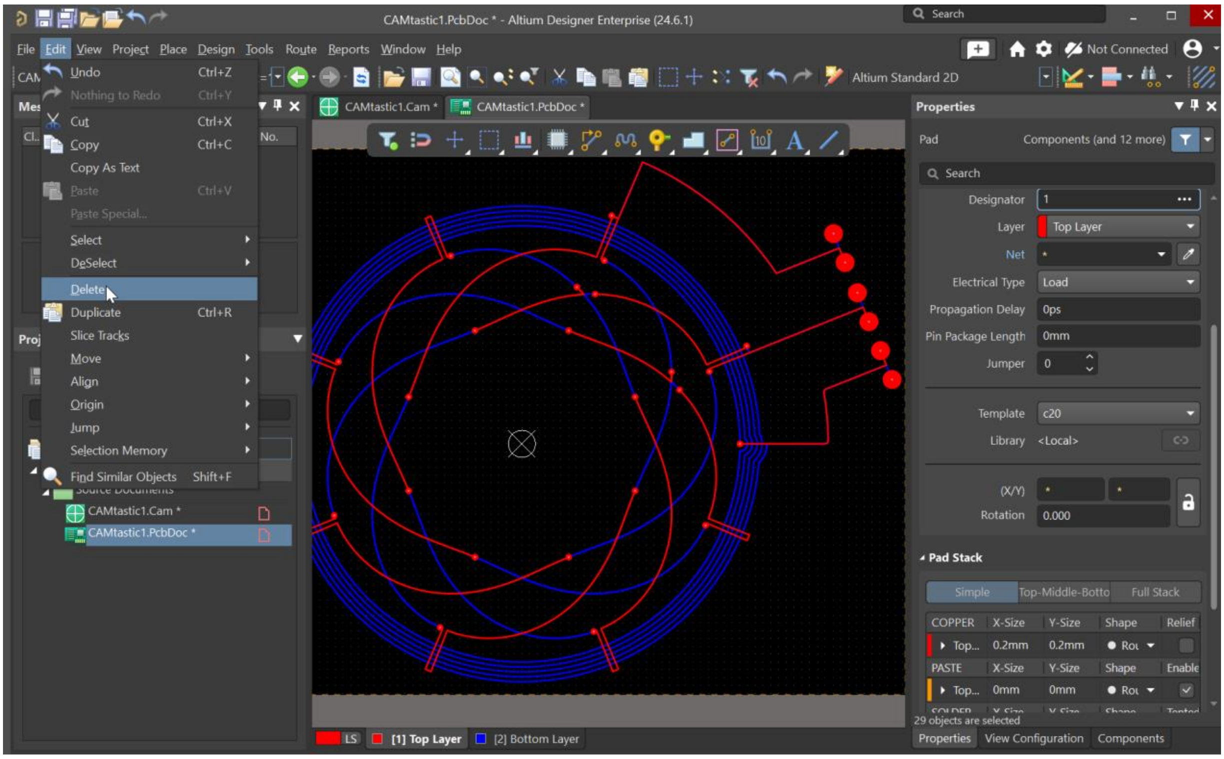


3.1.8. Set the X Size and Y Size

(or just one of them) to Same. Usually the rest of the settings should be unchanged, then press OK



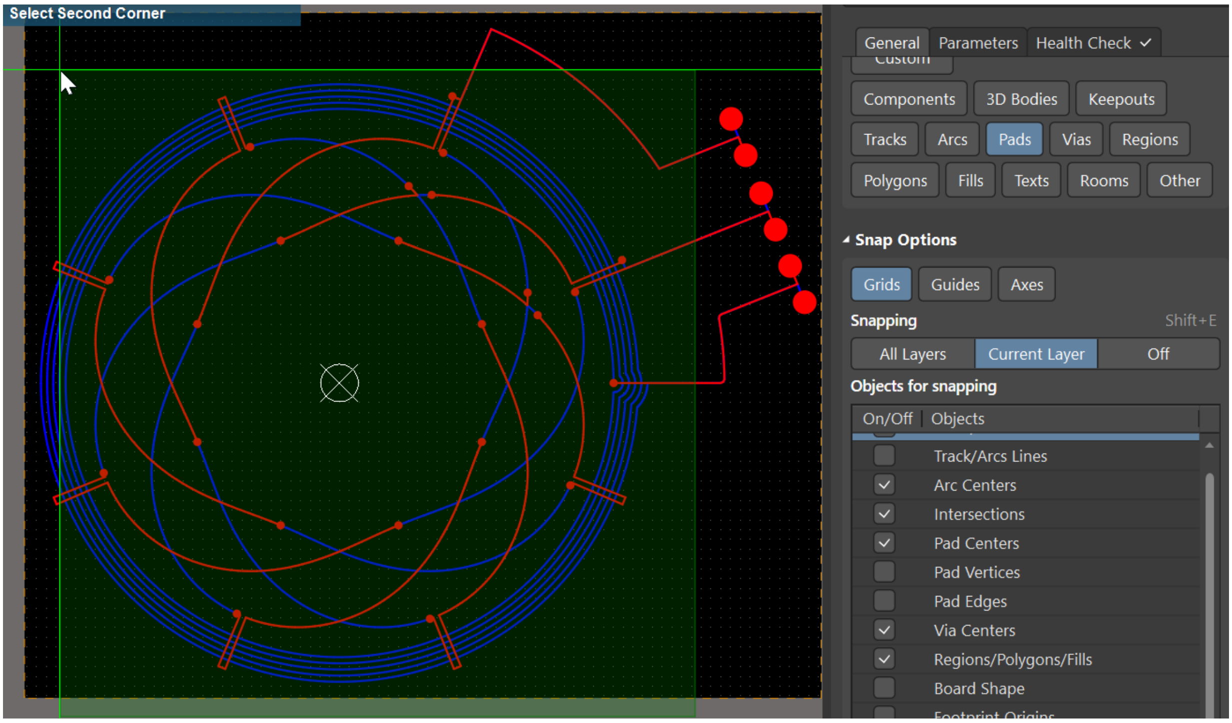
3.1.9. Now all 0.2mm pads are selected, press delete and eliminate them



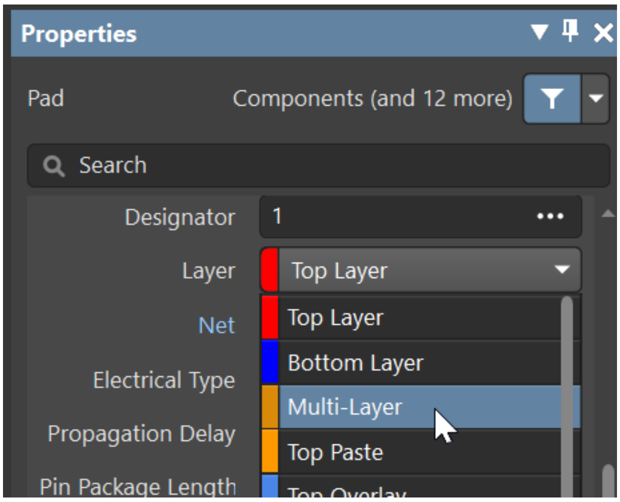
3.2 Creating vias and adjusting them

3.2.1. Select all the pads with the same size

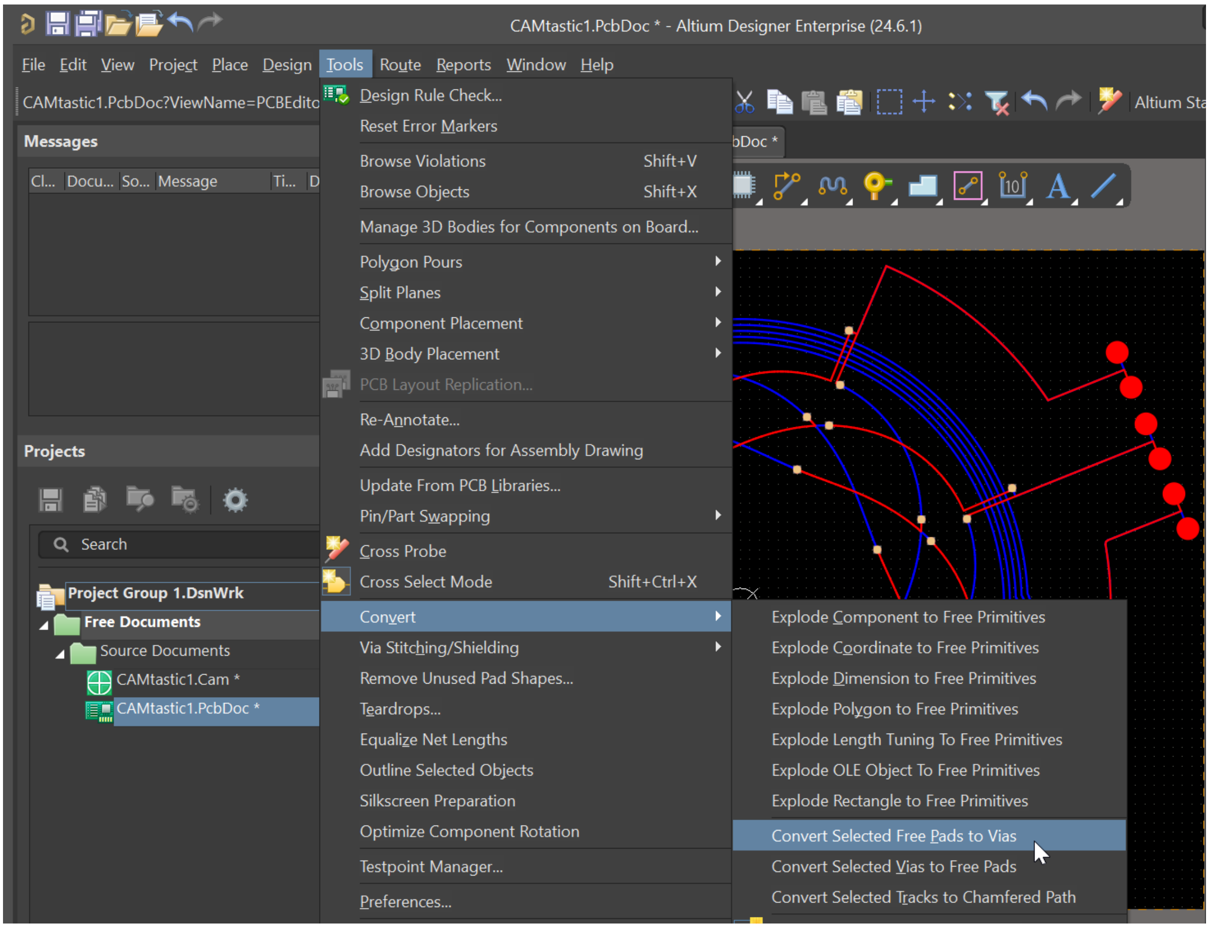
(exclude the header pads on the right)



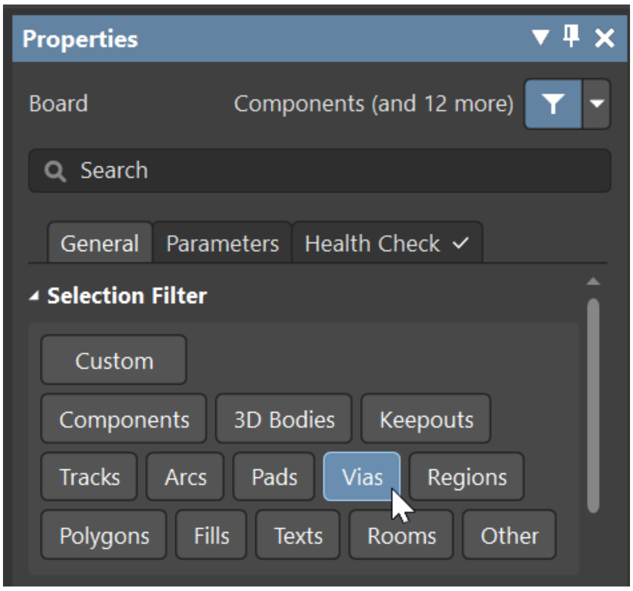
3.2.2. Set Multi-Layer instead of TOP Layer



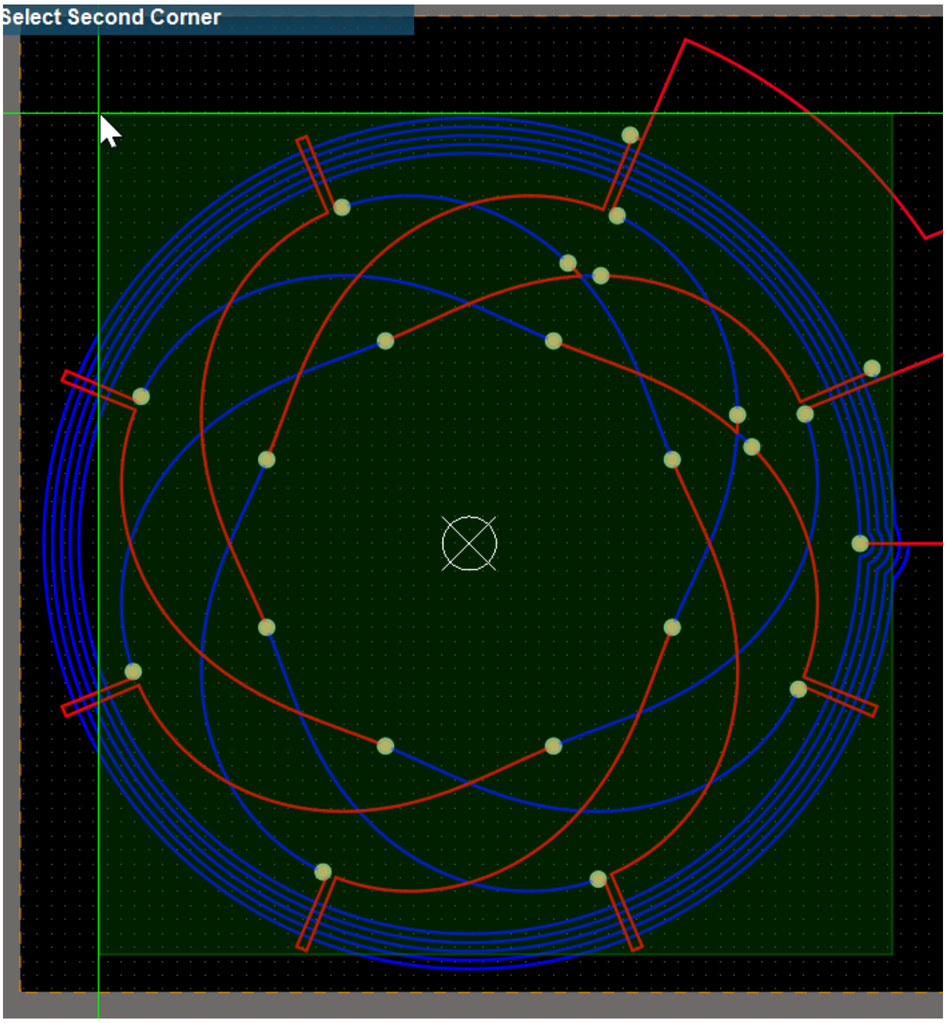
3.2.3. Convert Pads to Vias



3.2.4. Select Vias in the Selection Filter under Properties

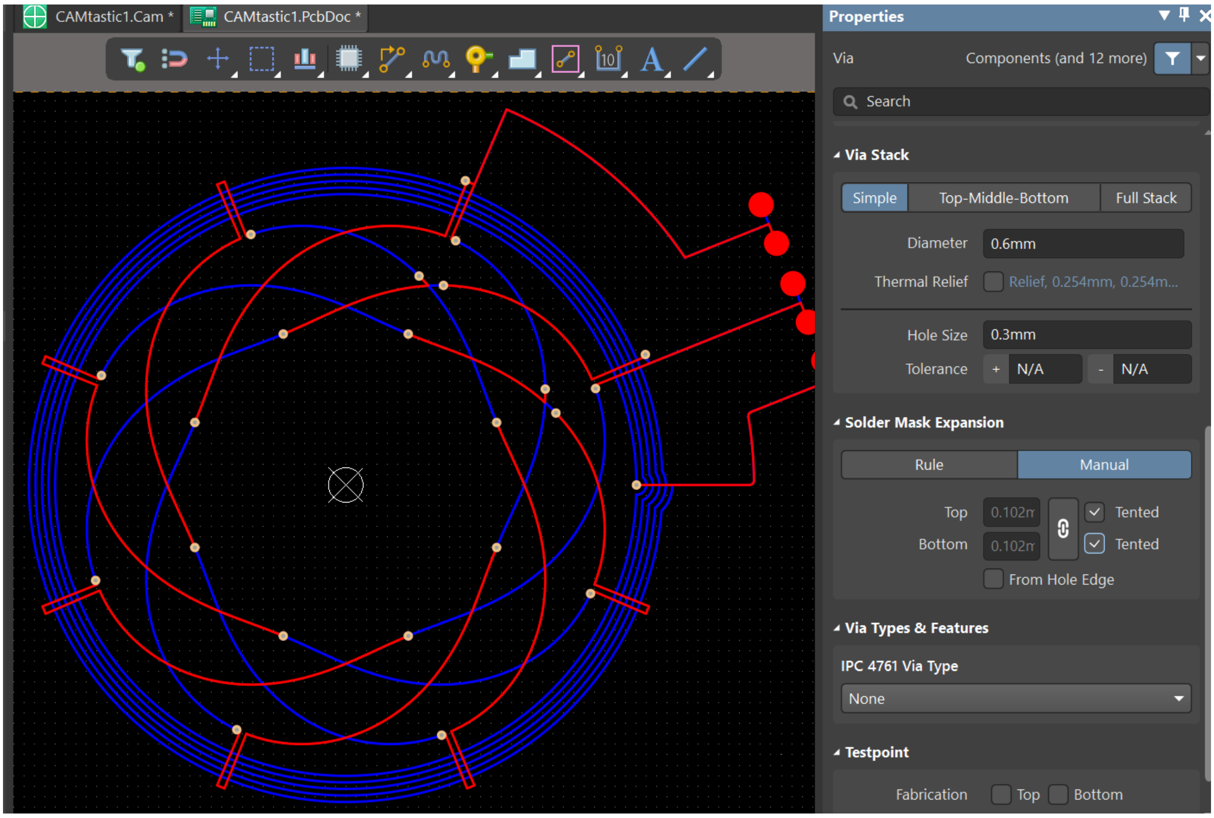


3.2.5. Select Vias



3.2.6. Adjust Via parameters under Properties:

Diameter should be correct as simulated, Hole Size should be adjusted to match PCB manufacturing requirements, also you can have tented or capped vias, it doesn´t matter in terms of inductive sensing performance



3.3 Adjust the rest:

Net Names, adjust the output headers, connect the sensing element to the IPS chip, finalize the layout, etc.

3.4 Don´t forget to Save the PCB! 😊

4. Revision History

Revision	Date	Description
1.0	Jul. 22, 24	Initial release.