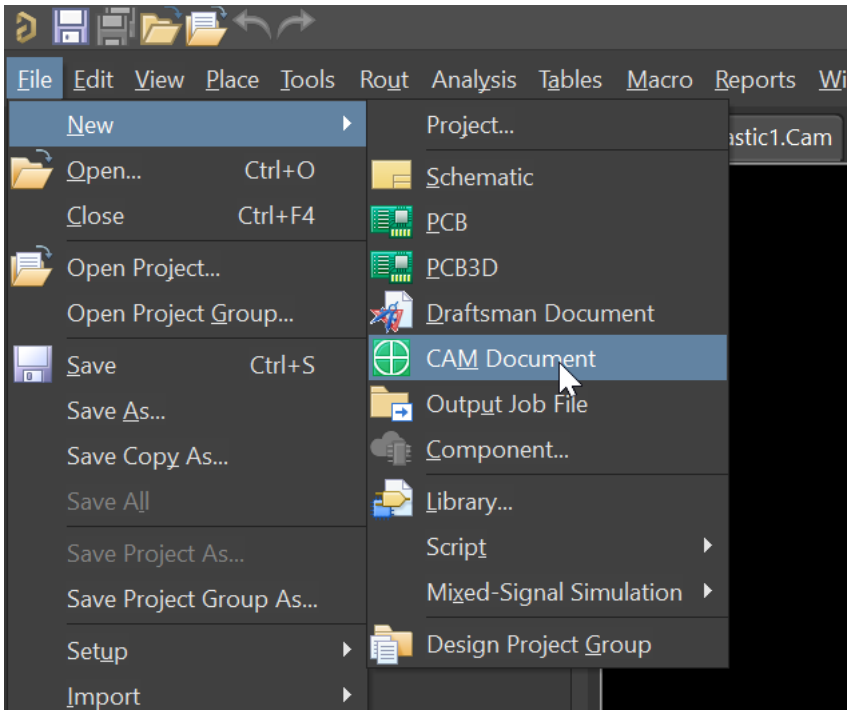
1. Process of importing Gerbers and Drill to Altium Designer 2
1.1 Importing Gerbers and Drill 2
1.1.1. Create new CAM Document in Altium Designer 2
1.1.2. Set up your import rules 2
1.1.3. Select Import Setting in the Gerber Import (Default) section 3
1.1.4. Set the exact same values then press OK 3
1.1.5. Import Gerber and Drill Files with Quick Load (hot keys are F I Q) 4
1.1.6. Select Import Directory by clicking the folder icon at the right side of the path field 5
1.1.7. Find the path and Select Folder 5
1.1.8. Select Gerbers and Drill then press OK 6
1.1.9. Press OK, the units are already set 6
1.1.10. Warnings and Errors should be 0, then close this log file 7
1.1.11. Gerber Files and Drill(s) are successfully imported 7
2. Adjusting the CAM File 8
2.1 Setting the Layers 8
2.1.1. Set the correct layers then press OK 8
2.1.2. Set the physical layer order 9
2.2 Extract the Netlist 9
2.3 Export to PCB 10
3. Adjusting the PCB 11
3.1 Converting Pads to Vias 11
3.1.1. Pad properties 11
3.1.2. In Properties menu only select the Pad as selection filter 12
3.1.3. Select the entire imported sensor to mark all pads 12
3.1.4. Convert selected free pads to vias (hot keys are T V P) 13
3.2 Adjusting vias 14
3.2.1. Select Vias in the Selection Filter under Properties 14
3.2.2. Select all the vias 14
3.2.3. Adjust Via parameters under Properties: 15
3.3 Adjust the rest 15
3.4 Don´t forget to Save the PCB! 15
4. Revision History 16

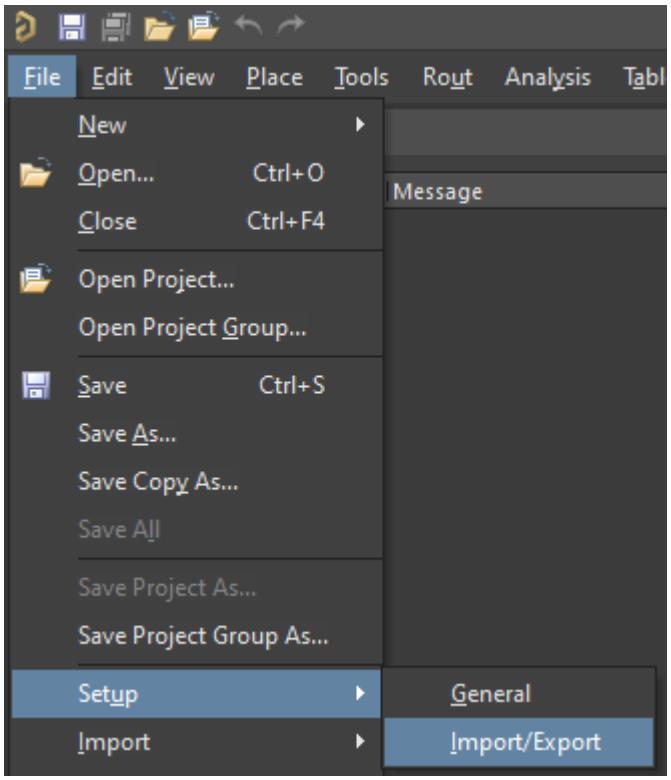
1. Process of importing Gerbers and Drill to Altium Designer

1.1 Importing Gerbers and Drill

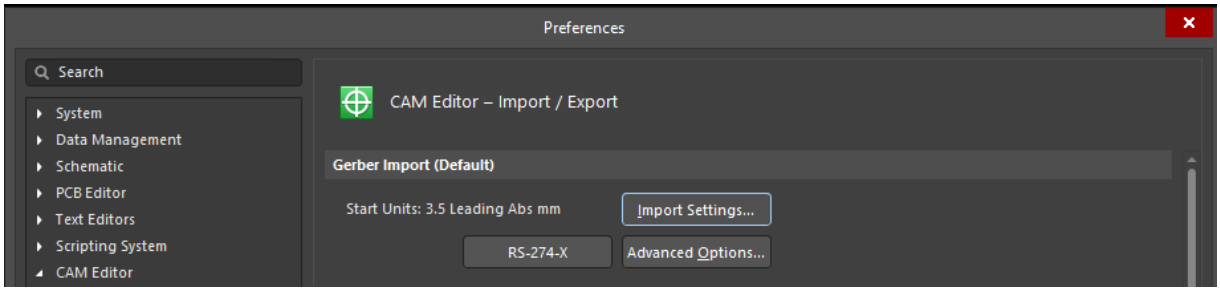
1.1.1. Create new CAM Document in Altium Designer



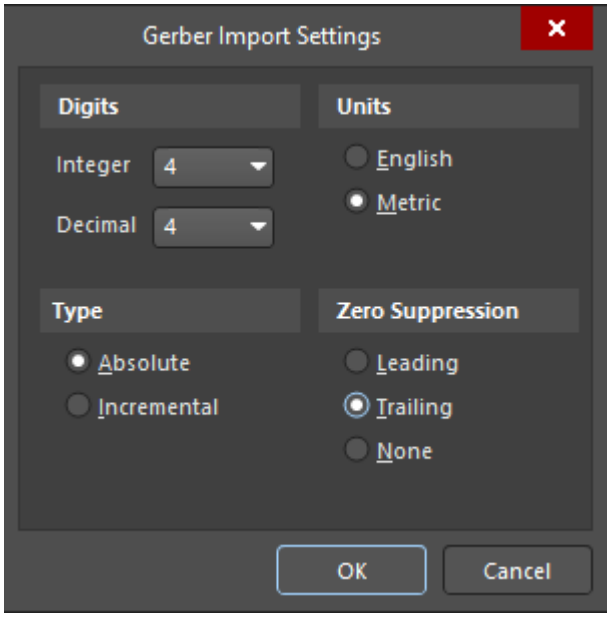
1.1.2. Set up your import rules



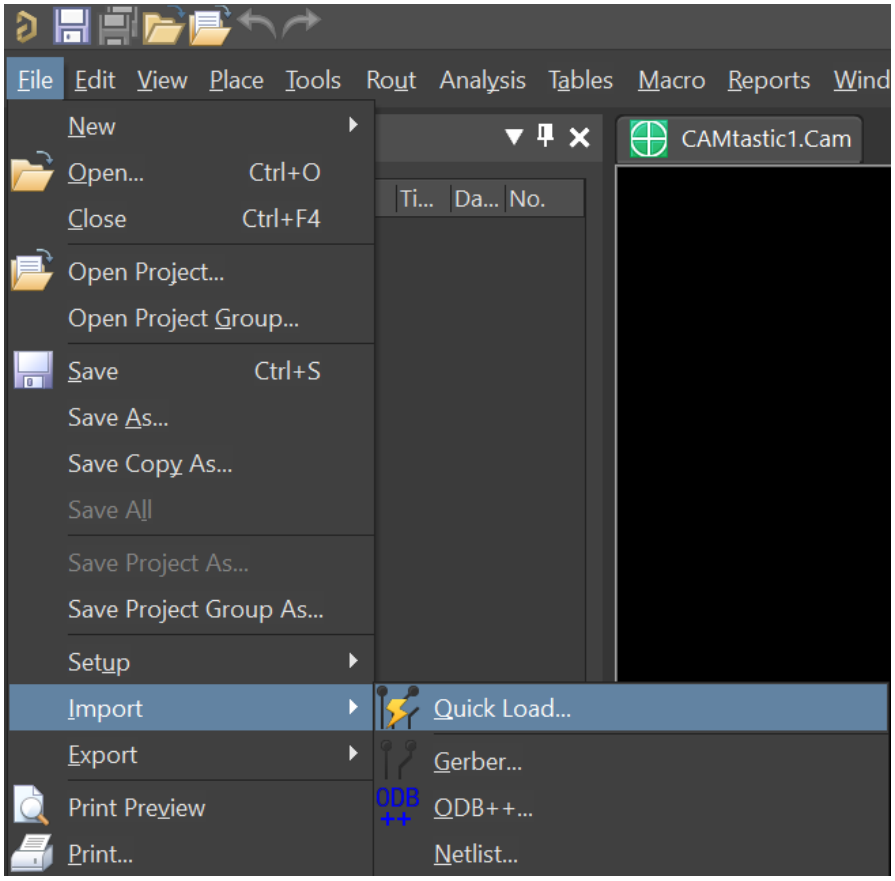
1.1.3. Select Import Setting in the Gerber Import (Default) section



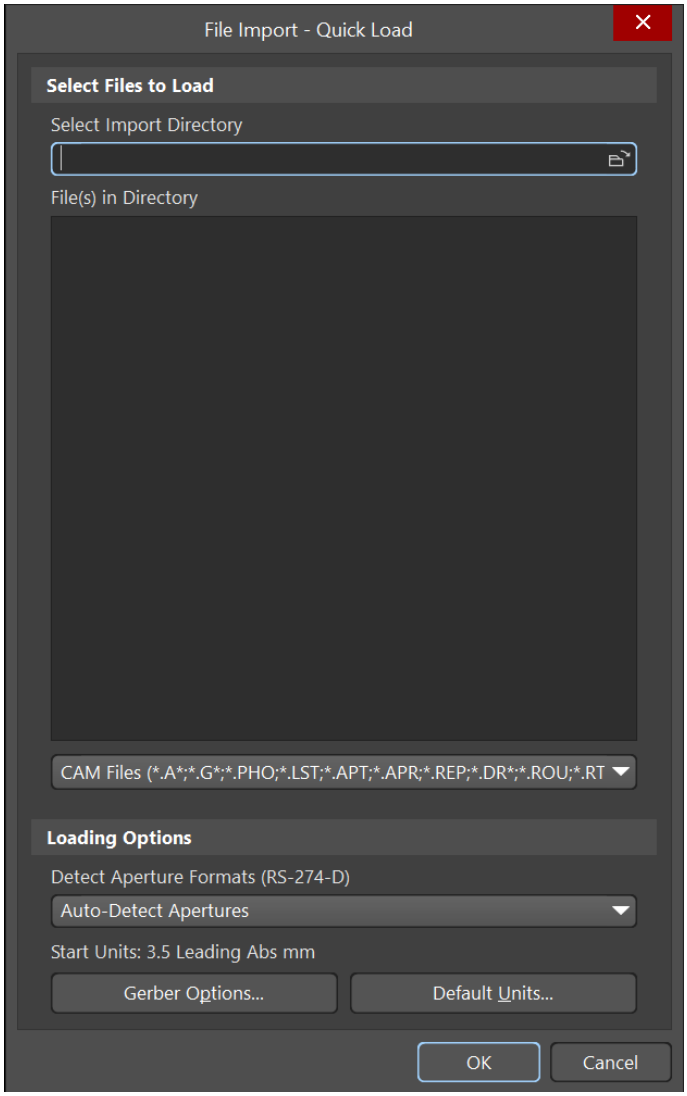
1.1.4. Set the exact same values then press OK



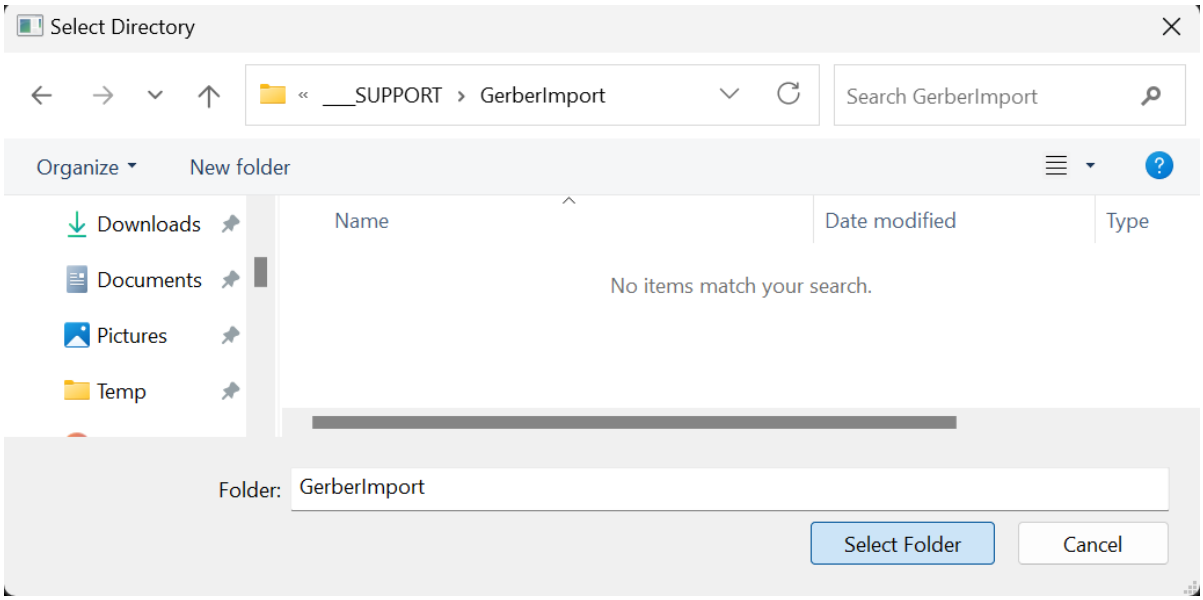
1.1.5. Import Gerber and Drill Files with Quick Load (hot keys are F I Q)



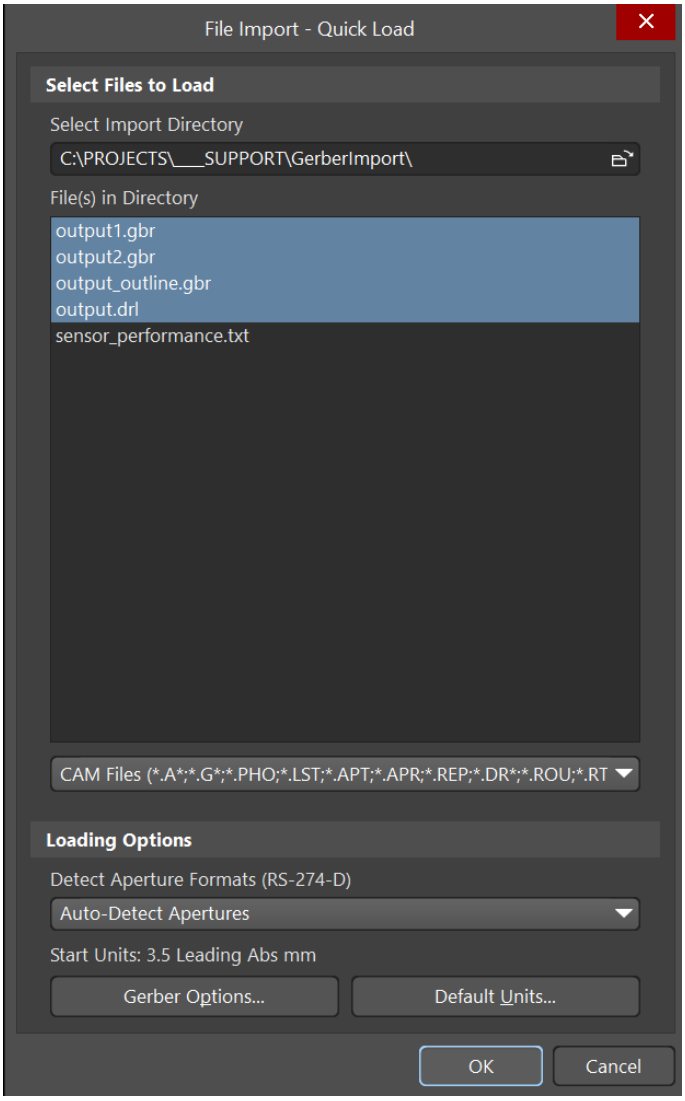
1.1.6. Select Import Directory by clicking the folder icon at the right side of the path field



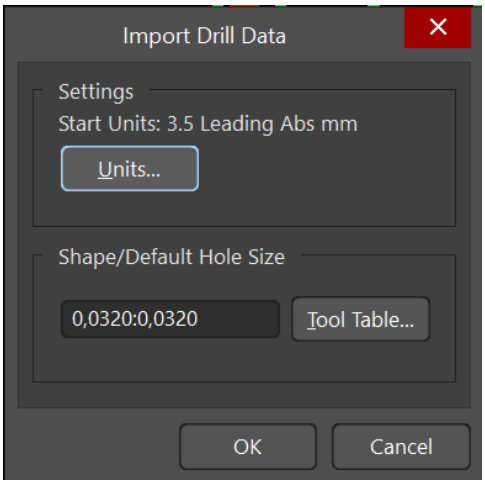
1.1.7. Find the path and Select Folder



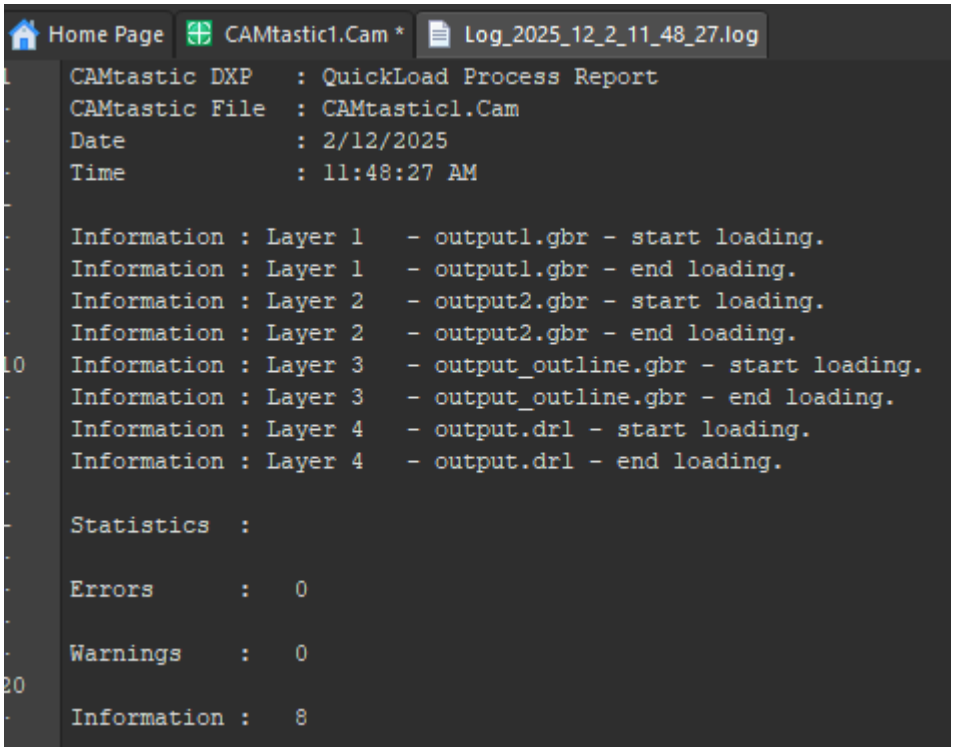
1.1.8. Select Gerbers and Drill then press OK



1.1.9. Press OK, the units are already set

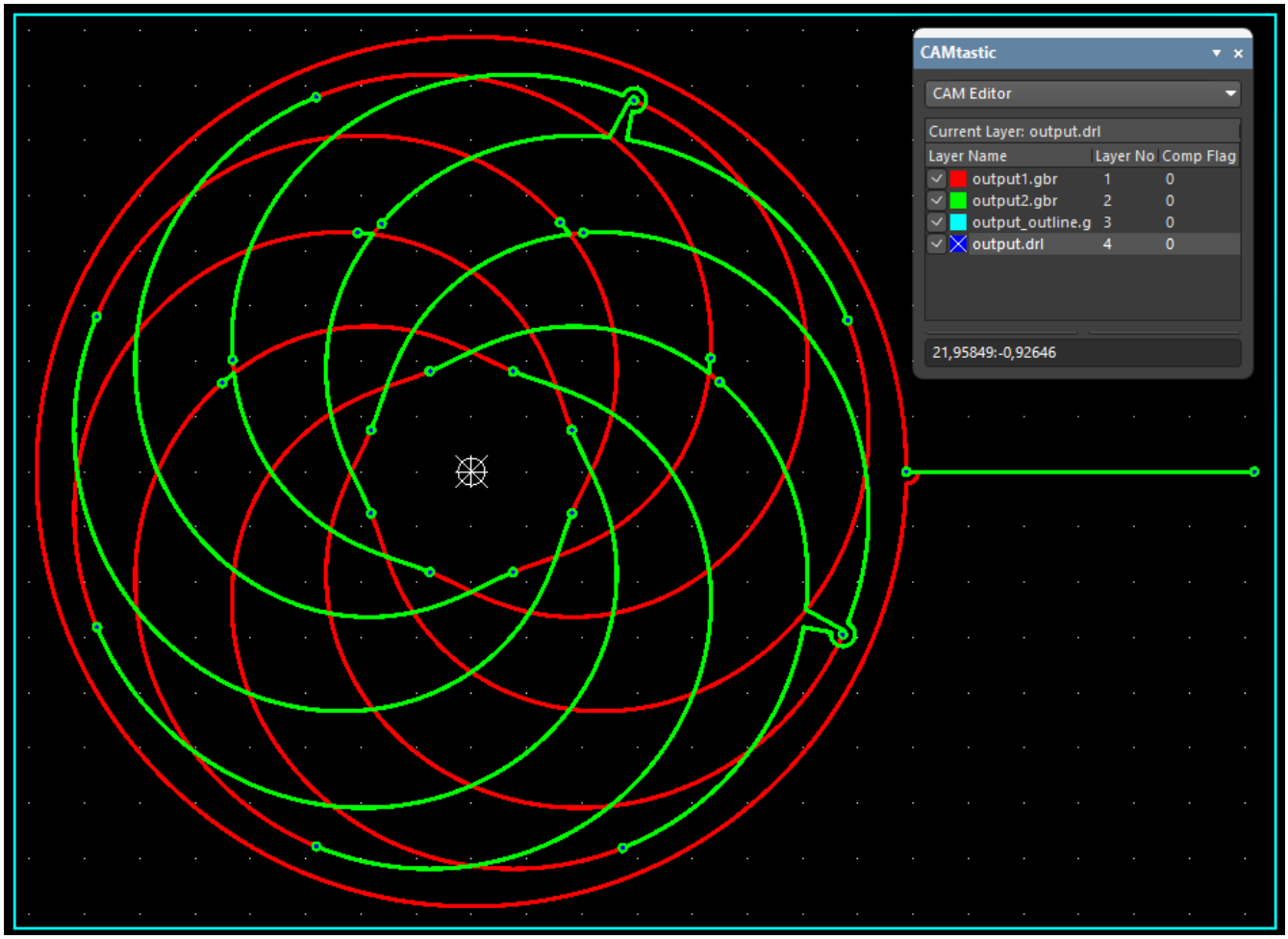


1.1.10. Warnings and Errors should be 0, then close this log file



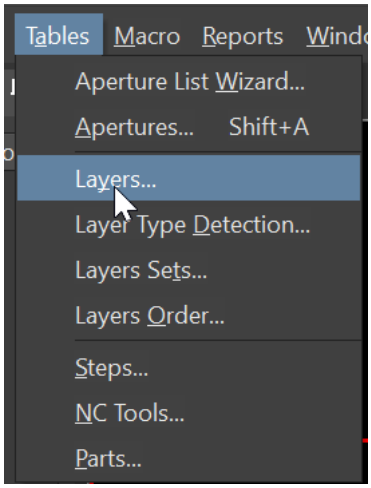
```
Home Page   CAMtastic1.Cam *   Log_2025_12_2_11_48_27.log

CAMtastic DXP    : QuickLoad Process Report
CAMtastic File   : CAMtastic1.Cam
Date             : 2/12/2025
Time             : 11:48:27 AM

Information : Layer 1   - output1.gbr - start loading.
Information : Layer 1   - output1.gbr - end loading.
Information : Layer 2   - output2.gbr - start loading.
Information : Layer 2   - output2.gbr - end loading.
Information : Layer 3   - output_outline.gbr - start loading.
Information : Layer 3   - output_outline.gbr - end loading.
Information : Layer 4   - output.drl - start loading.
Information : Layer 4   - output.drl - end loading.

Statistics  :

Errors      :   0

Warnings    :   0

Information :   8
```

1.1.11. Gerber Files and Drill(s) are successfully imported

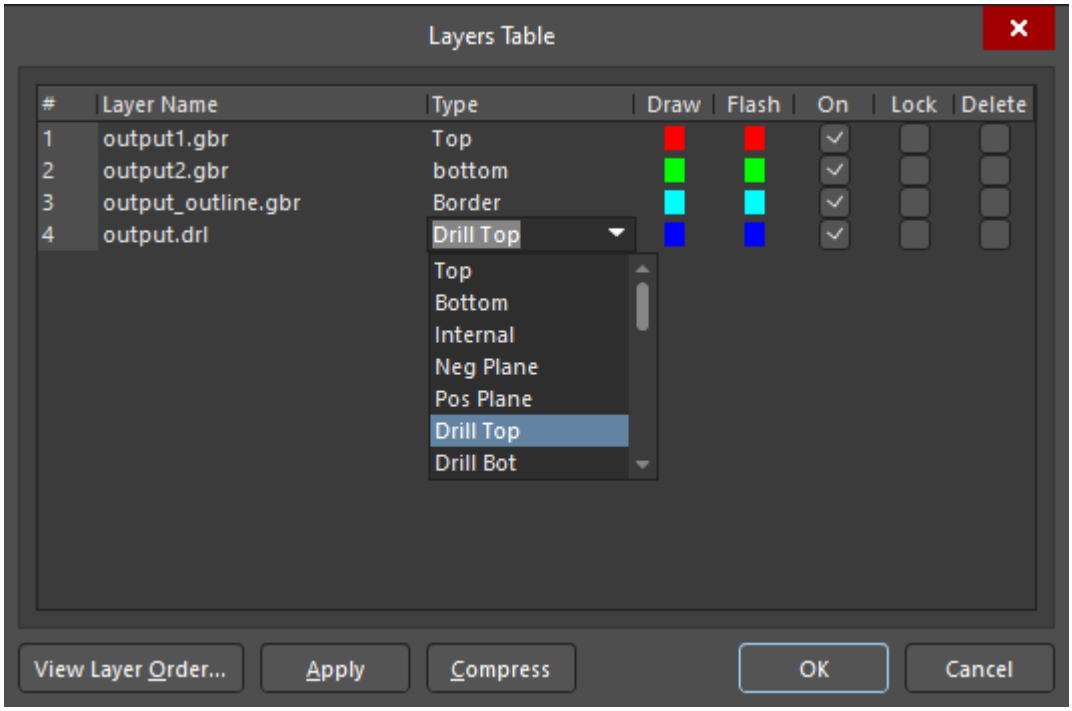


2. Adjusting the CAM File

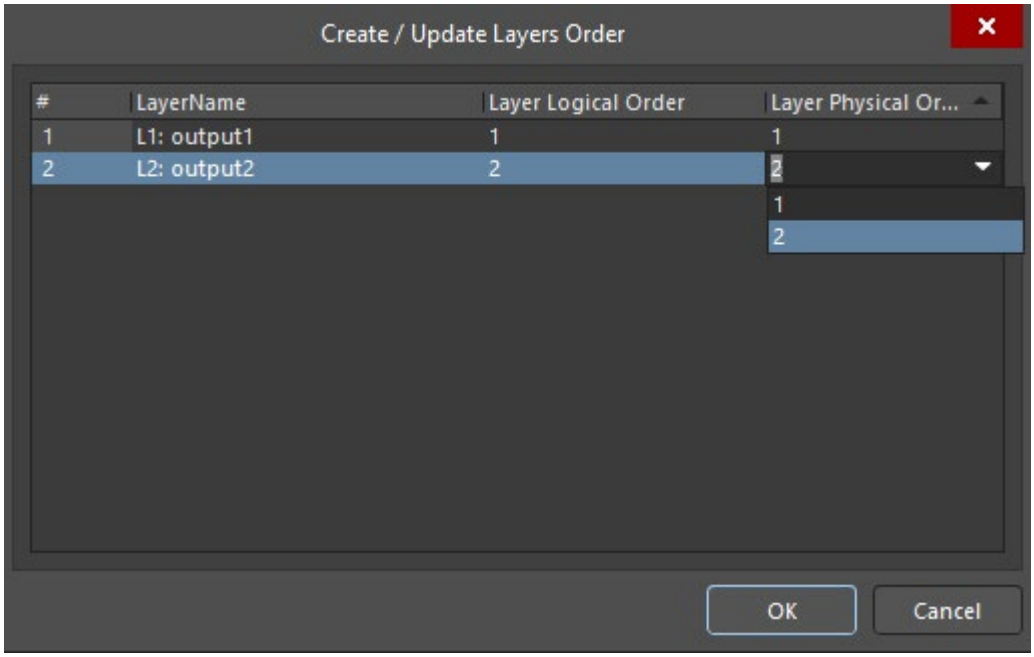
2.1 Setting the Layers



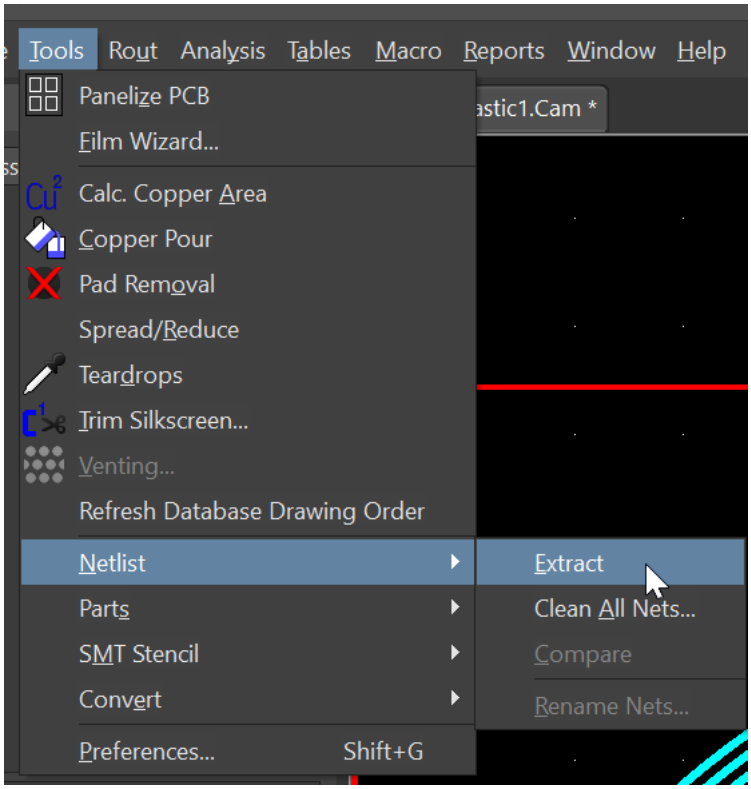
2.1.1. Set the correct layers then press OK



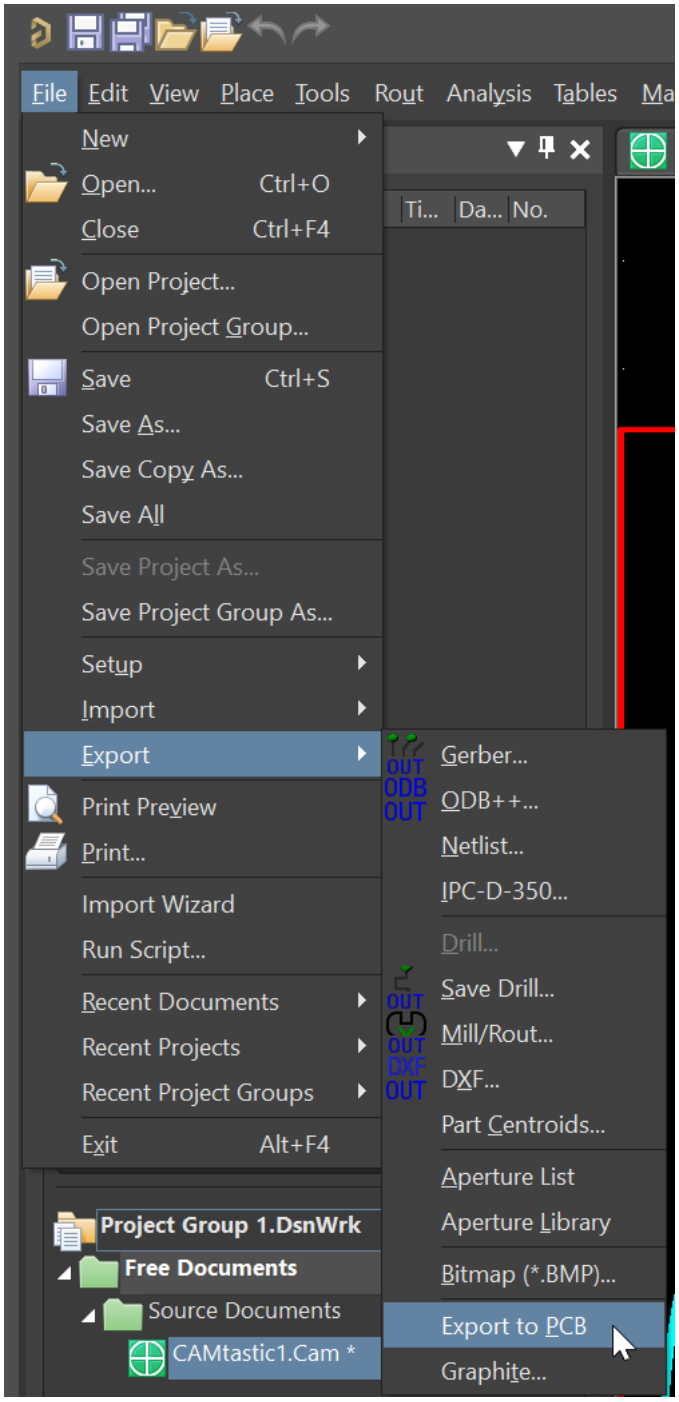
2.1.2. Set the physical layer order



2.2 Extract the Netlist



2.3 Export to PCB

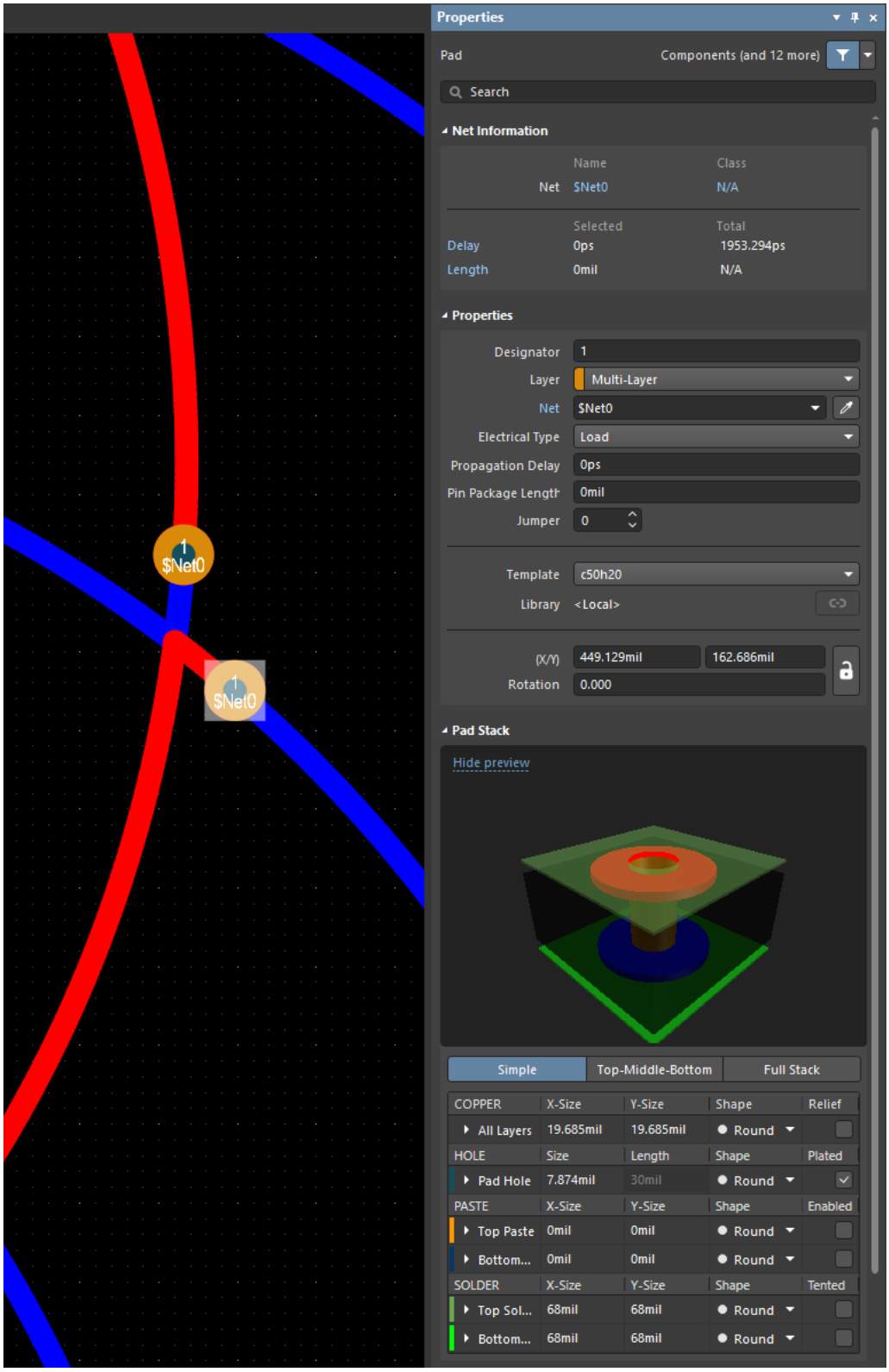


3. Adjusting the PCB

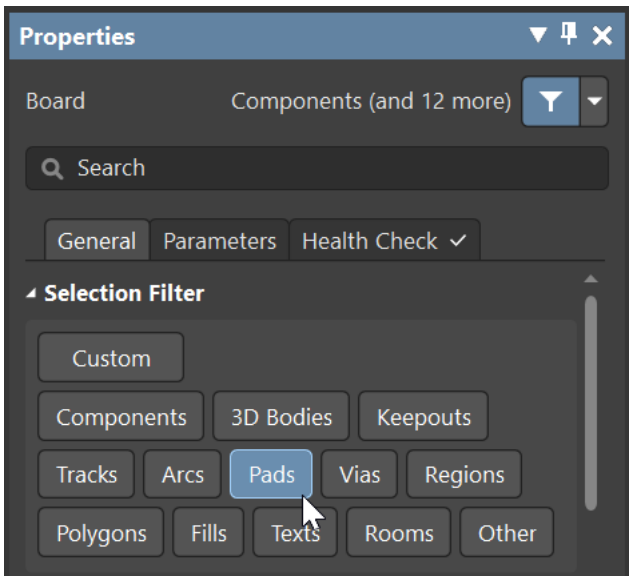
3.1 Converting Pads to Vias

3.1.1. Pad properties

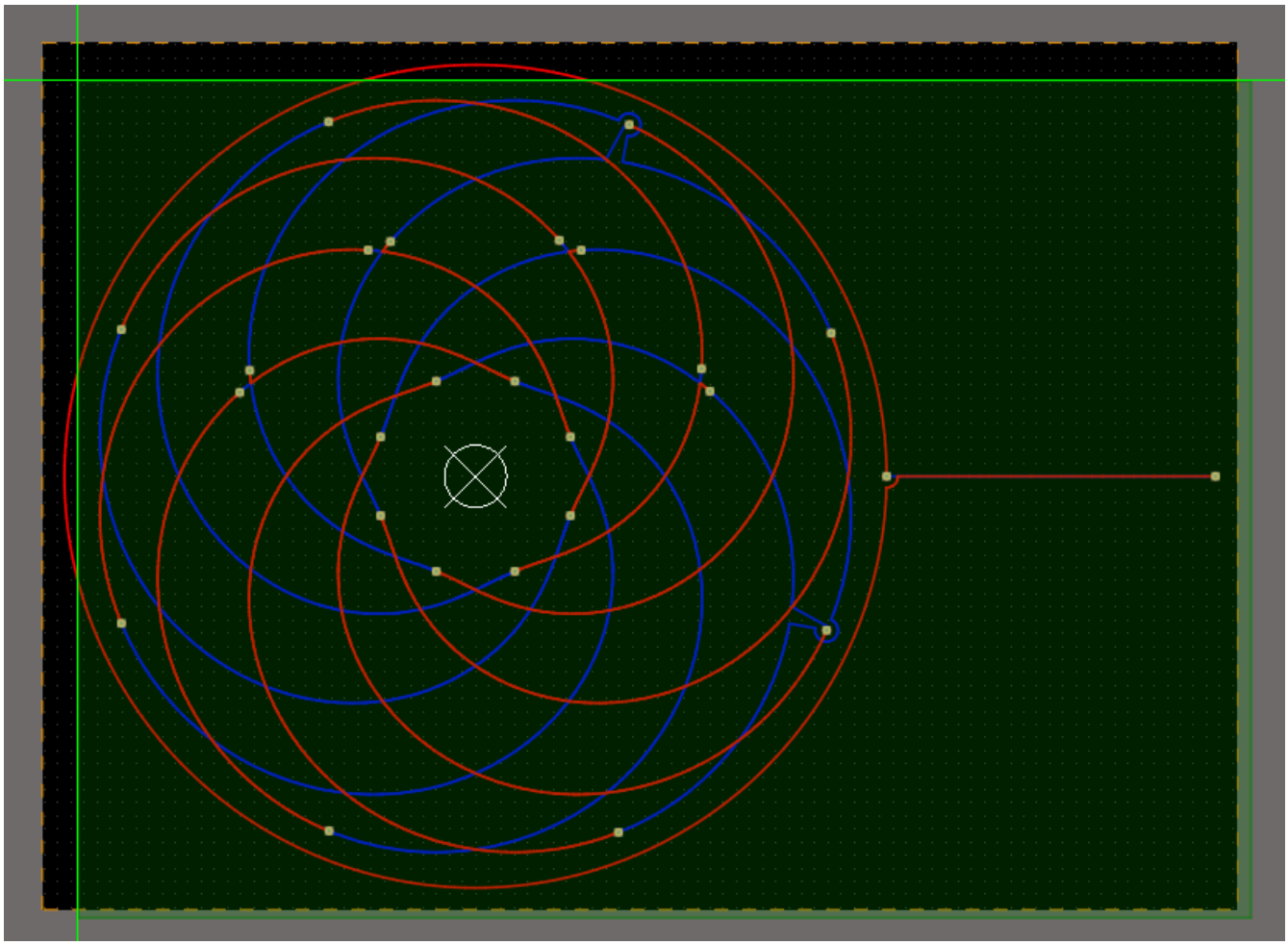
By default, via pads and via drills are imported combined as one “multilayer pad” with a plated drill hole. This can remain unchanged but, in this case, make sure that your design rules handle these “pads” correctly. We recommend converting them to “via” class.



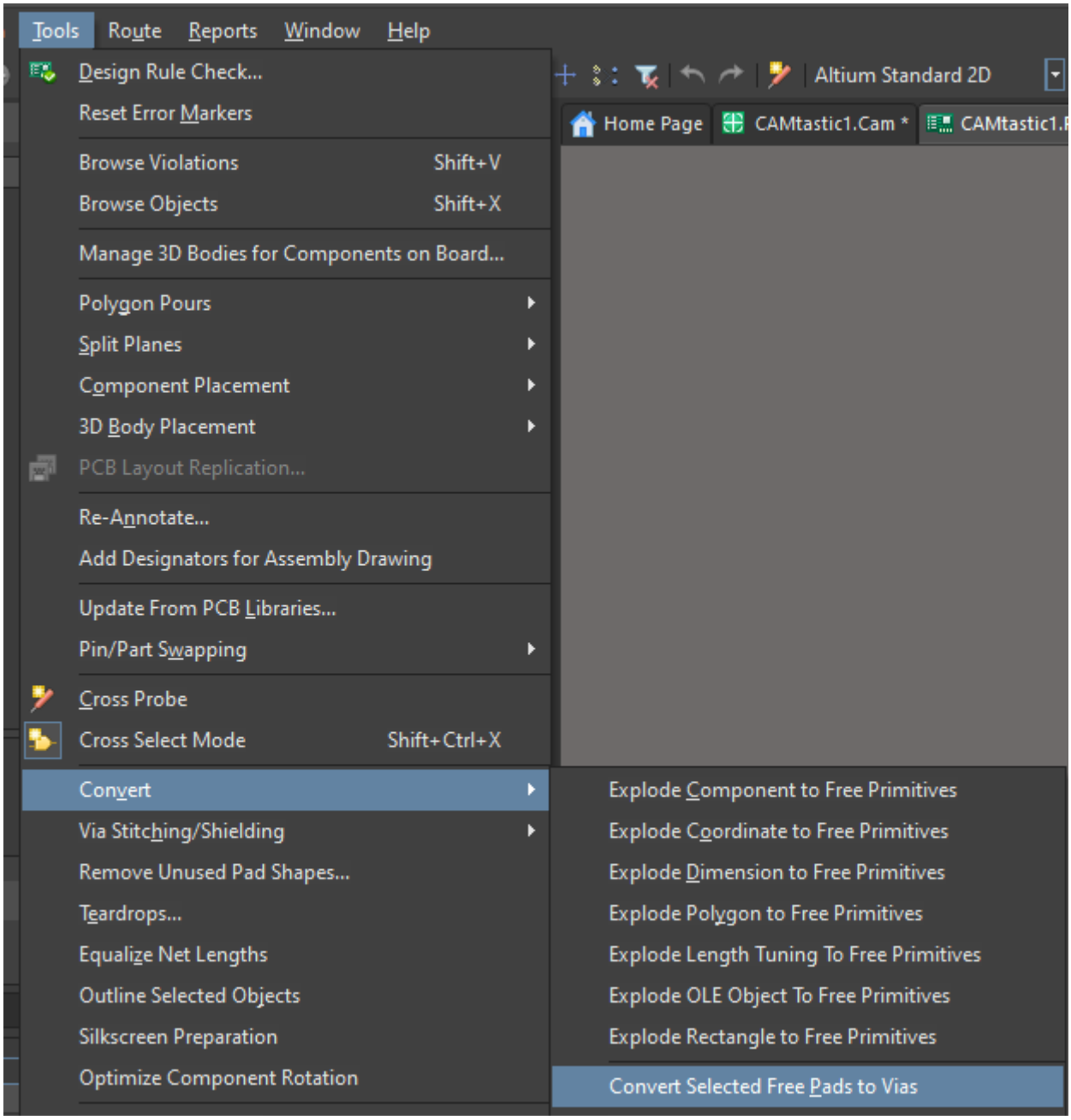
3.1.2. In Properties menu only select the Pad as selection filter



3.1.3. Select the entire imported sensor to mark all pads

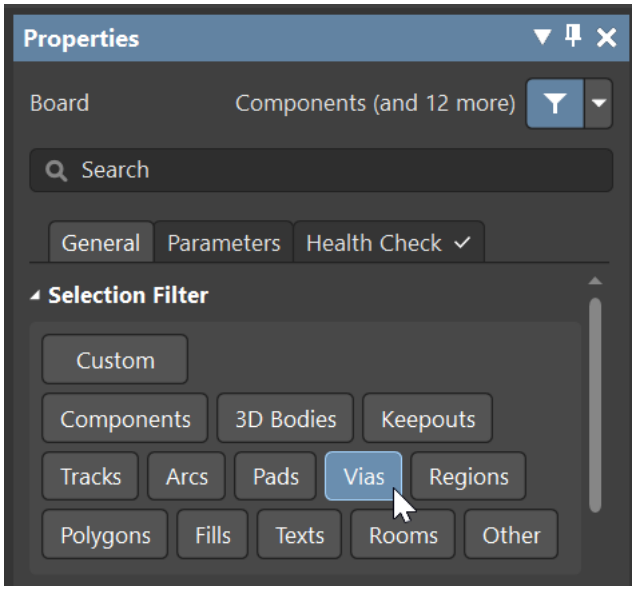


3.1.4. Convert selected free pads to vias (hot keys are T V P)

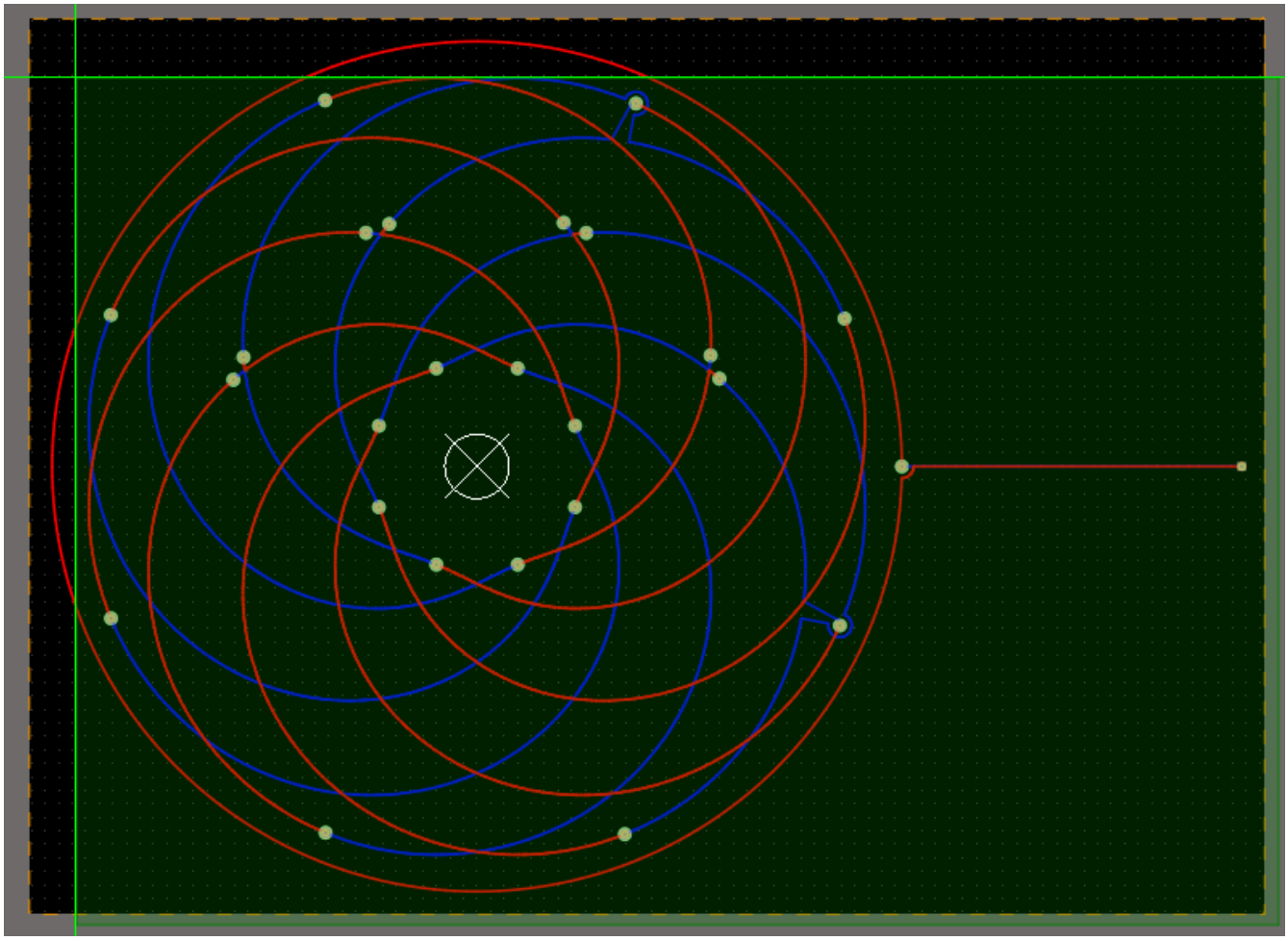


3.2 Adjusting vias

3.2.1. Select Vias in the Selection Filter under Properties

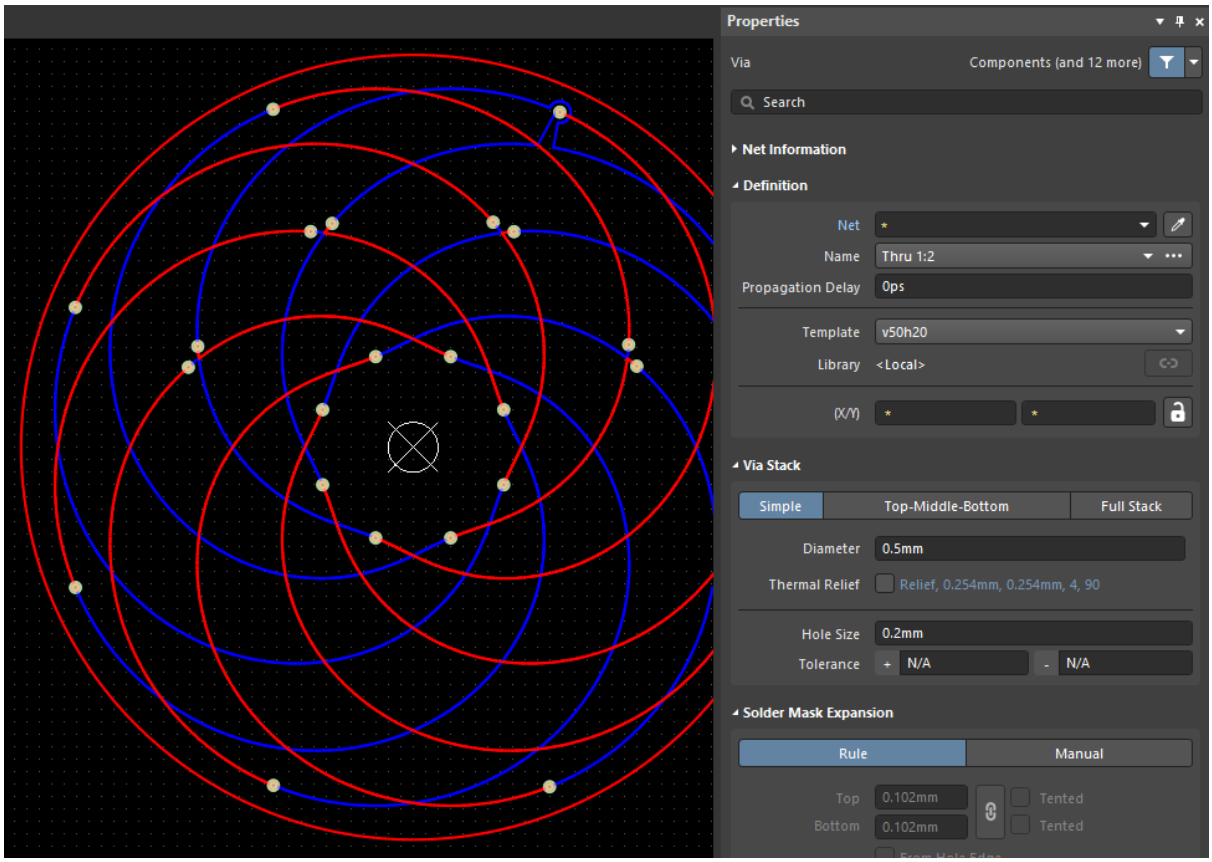


3.2.2. Select all the vias



3.2.3. Adjust Via parameters under Properties:

Via diameter and drill size should be correct, just as the simulated values. Feel free to adjust it to your needs: tenting, different sizes (then DRC must be applied to eliminate distance related issues)



3.3 Adjust the rest

Net Names, adjust the output headers, copy coils to a chip layout template, connect the sensing element to the IPS chip, finalize the layout, properly run a DRC (design rules check), etc.

3.4 Don´t forget to Save the PCB! 😊

4. Revision History

Revision	Date	Description
1.0	Jul. 22, 24	Initial release.
1.1	Dec. 2, 25	Update with Quick Load function
1.2	Dec. 2, 25	Update in Quick Load explanation due to bugfix in the IPSCO Gerber tool

IMPORTANT NOTICE AND DISCLAIMER

RENESAS ELECTRONICS CORPORATION AND ITS SUBSIDIARIES (“RENESAS”) PROVIDES TECHNICAL SPECIFICATIONS AND RELIABILITY DATA (INCLUDING DATASHEETS), DESIGN RESOURCES (INCLUDING REFERENCE DESIGNS), APPLICATION OR OTHER DESIGN ADVICE, WEB TOOLS, SAFETY INFORMATION, AND OTHER RESOURCES “AS IS” AND WITH ALL FAULTS, AND DISCLAIMS ALL WARRANTIES, EXPRESS OR IMPLIED, INCLUDING, WITHOUT LIMITATION, ANY IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, OR NON-INFRINGEMENT OF THIRD-PARTY INTELLECTUAL PROPERTY RIGHTS.

These resources are intended for developers who are designing with Renesas products. You are solely responsible for (1) selecting the appropriate products for your application, (2) designing, validating, and testing your application, and (3) ensuring your application meets applicable standards, and any other safety, security, or other requirements. These resources are subject to change without notice. Renesas grants you permission to use these resources only to develop an application that uses Renesas products. Other reproduction or use of these resources is strictly prohibited. No license is granted to any other Renesas intellectual property or to any third-party intellectual property. Renesas disclaims responsibility for, and you will fully indemnify Renesas and its representatives against, any claims, damages, costs, losses, or liabilities arising from your use of these resources. Renesas' products are provided only subject to Renesas' Terms and Conditions of Sale or other applicable terms agreed to in writing. No use of any Renesas resources expands or otherwise alters any applicable warranties or warranty disclaimers for these products.

(Disclaimer Rev.1.01)

Corporate Headquarters

TOYOSU FORESIA, 3-2-24 Toyosu,
Koto-ku, Tokyo 135-0061, Japan
www.renesas.com

Trademarks

Renesas and the Renesas logo are trademarks of Renesas Electronics Corporation. All trademarks and registered trademarks are the property of their respective owners.

Contact Information

For further information on a product, technology, the most up-to-date version of a document, or your nearest sales office, please visit www.renesas.com/contact-us/.

© 2025 Renesas Electronics Corporation. All rights reserved.