# Importing Gerbers and Drill into Altium Designer

## Introduction

This manual describes how to import Gerber and drill data into Altium Designer, review and correct the CAM setup, convert the data into a PCB document, and perform essential post-import cleanup steps commonly needed for semiconductor sensor/coil layouts and chip-interface PCBs.

## Contents



## Figures




© 2026 Renesas Electronics

# 1. Importing Gerbers and Drill into Altium Designer

Gerber files are the industry-standard format for PCB manufacturing data, containing information about copper layers, solder mask, silkscreen, and other PCB features.

To create a CAM (Computer-Aided Manufacturing) document in Altium Designer, configuring the necessary import settings, and loading Gerber and drill files, follow these steps:

1. Create a new CAM document by doing the following:
   a. Open Altium Designer.
   b. Go to File → New → CAM Document.

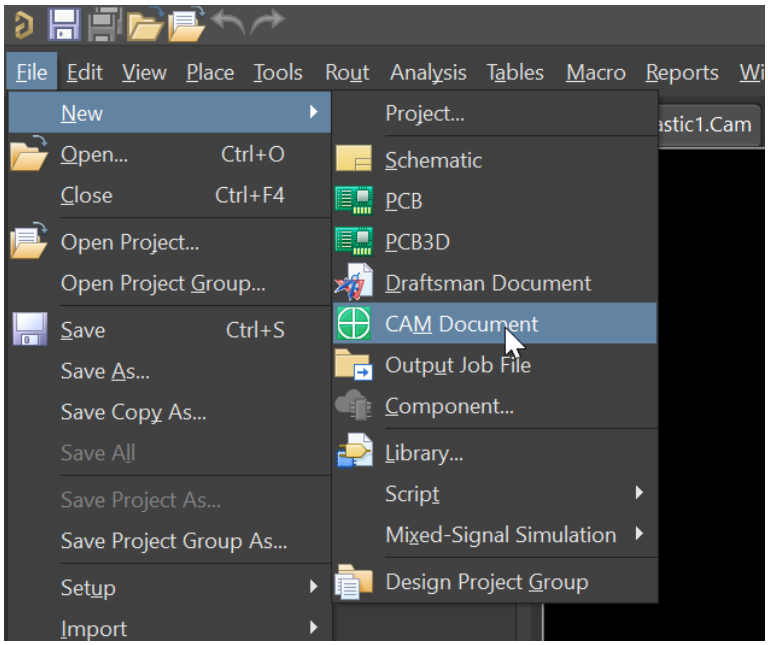


**Figure 1. Creating a New CAM Document**

2. Set up your import rules by doing the following:
   a. Go to File → Setup → Import/Export.
   b. Confirm that the standard import rules are known before loading files.

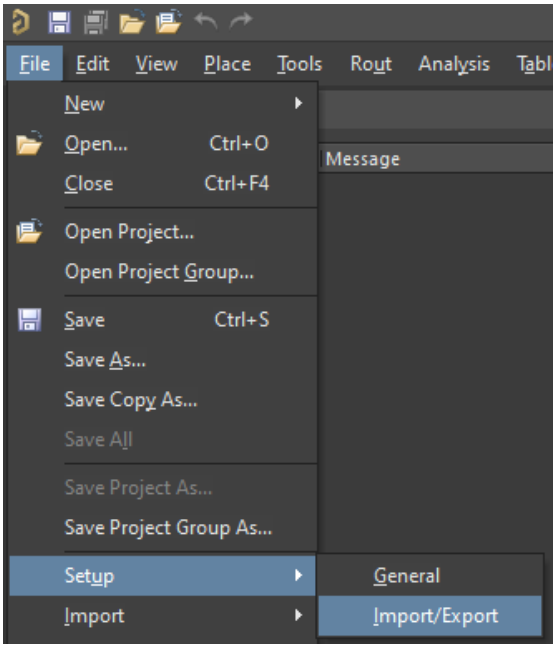


**Figure 2. Accessing Import/Export Setup**

3. Select the Gerber import settings by choosing Gerber Import (Default) or the company template if available by clicking Advanced Options.

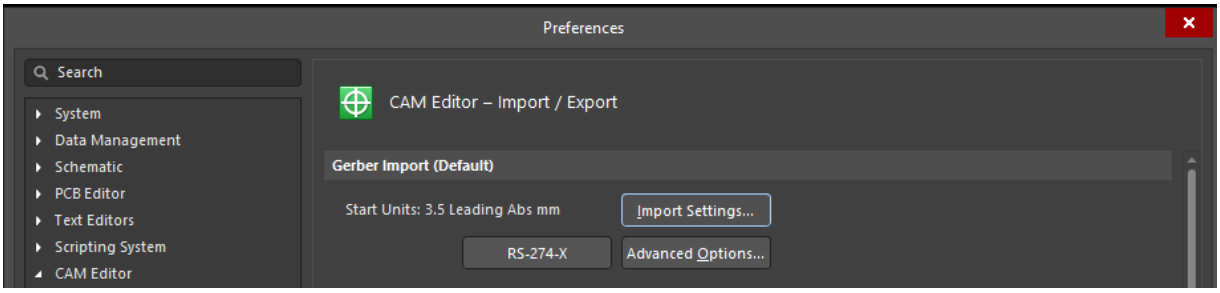


**Figure 3. Selecting Gerber Import Configuration**

4. Match settings exactly by doing the following:
   a. Enter the exact same import values as specified by your fabrication/export settings (digits, units, type, and zero suppression must match the Gerber generator).
   b. Click OK.

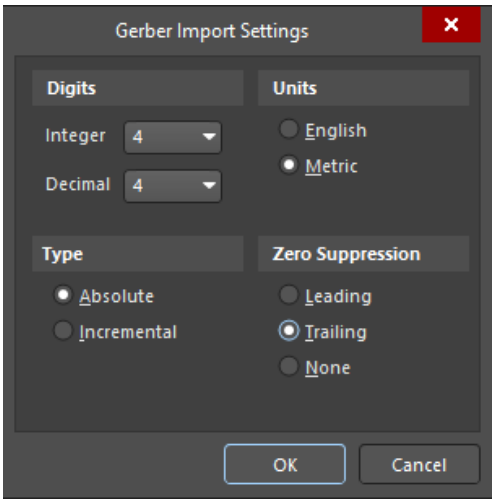


**Figure 4. Gerber Import Parameter Entry**

5. Import Gerber and drill files using Quick Load by going to File → Import → Quick Load.

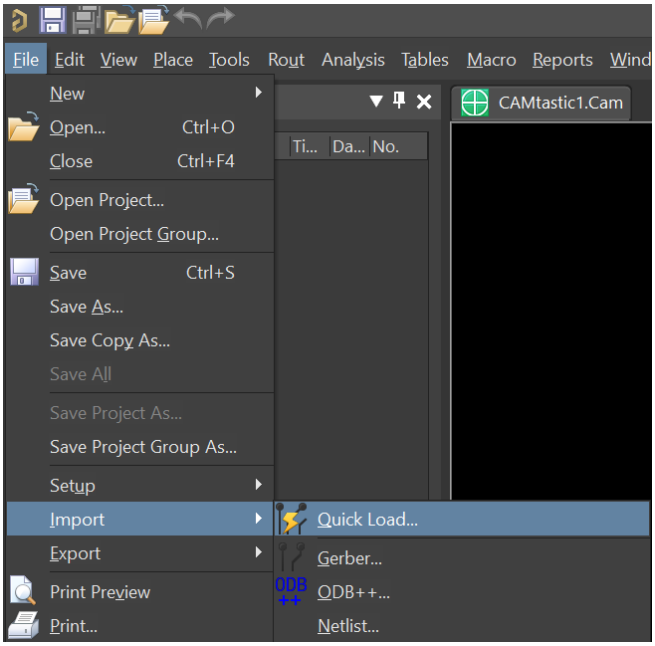


**Figure 5. Opening Quick Load for Gerber and Drill Import**

6. Select the import directory by clicking the folder icon to the right of the path field.

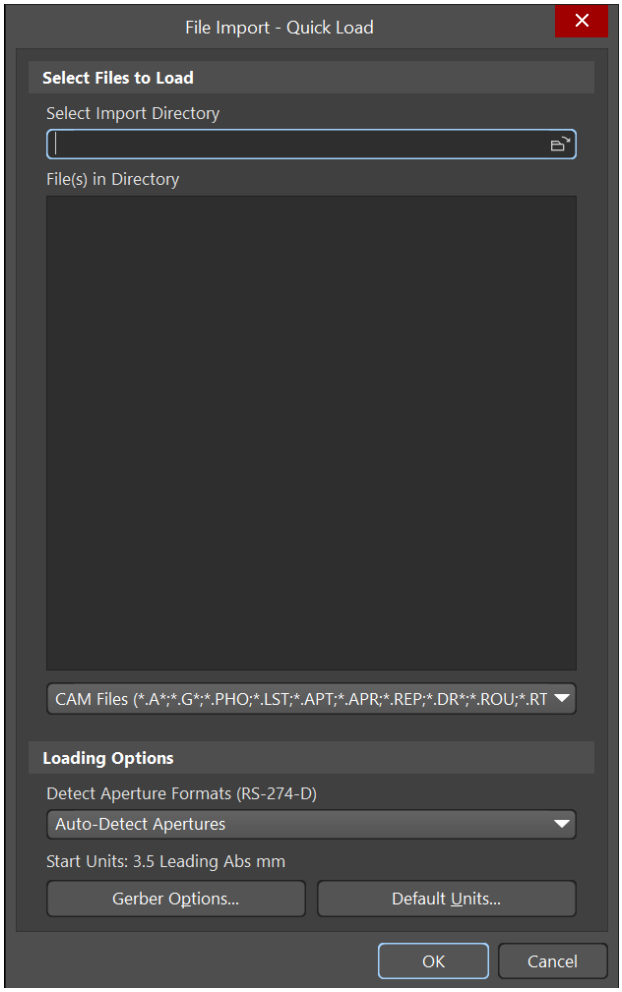


**Figure 6. Import Path Selection**

7. Locate the files by doing the following:
   a. Browse to the directory containing the exported fabrication package.
   b. Click Select Folder.

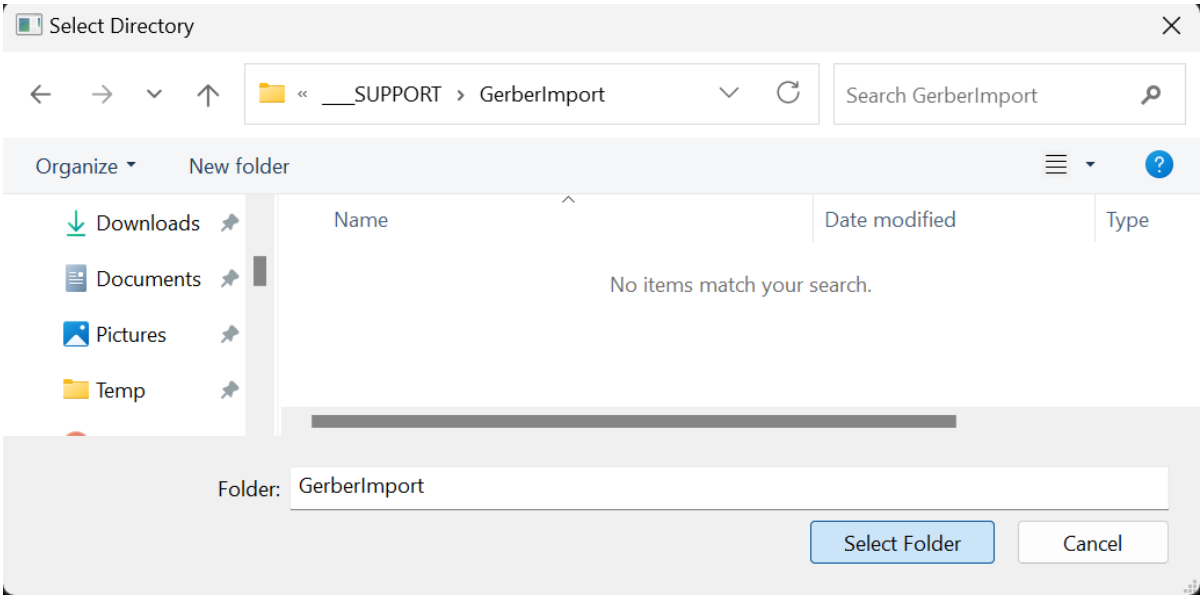


**Figure 7. Package Directory Selection**

8. Select file types to import by doing the following:
   a. Ensure Gerber and Drill are selected.
   b. Click OK.

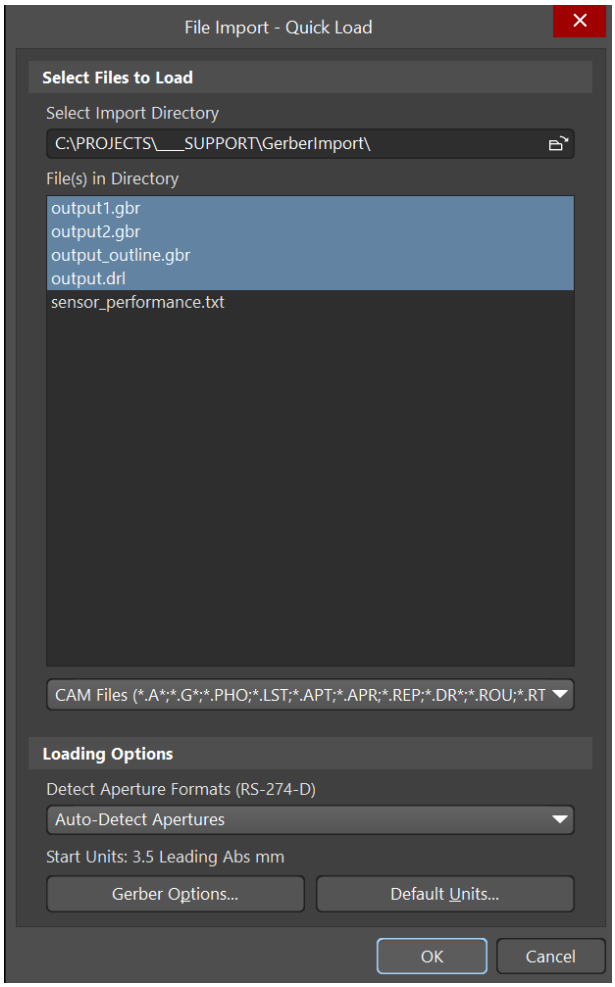


**Figure 8. Selecting Gerber and Drill File Types for Import**

9. Confirm units by doing the following:
   a. Click Units to proceed.
   b. Confirm units are correct.
   c. Click OK.

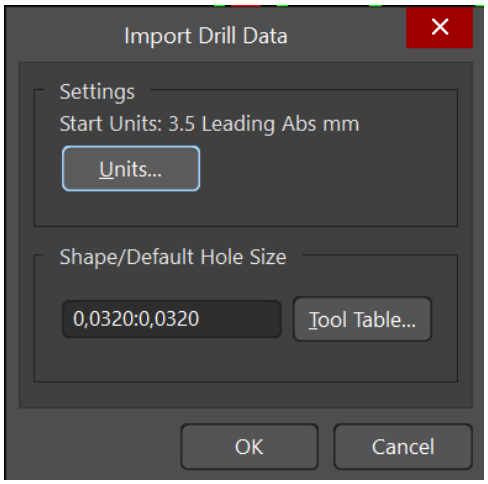


**Figure 9. Units Confirmation Dialog**

10. Check the import log by confirming:
    a. Errors: 0
    b. Warnings: 0
    c. Information: 0.

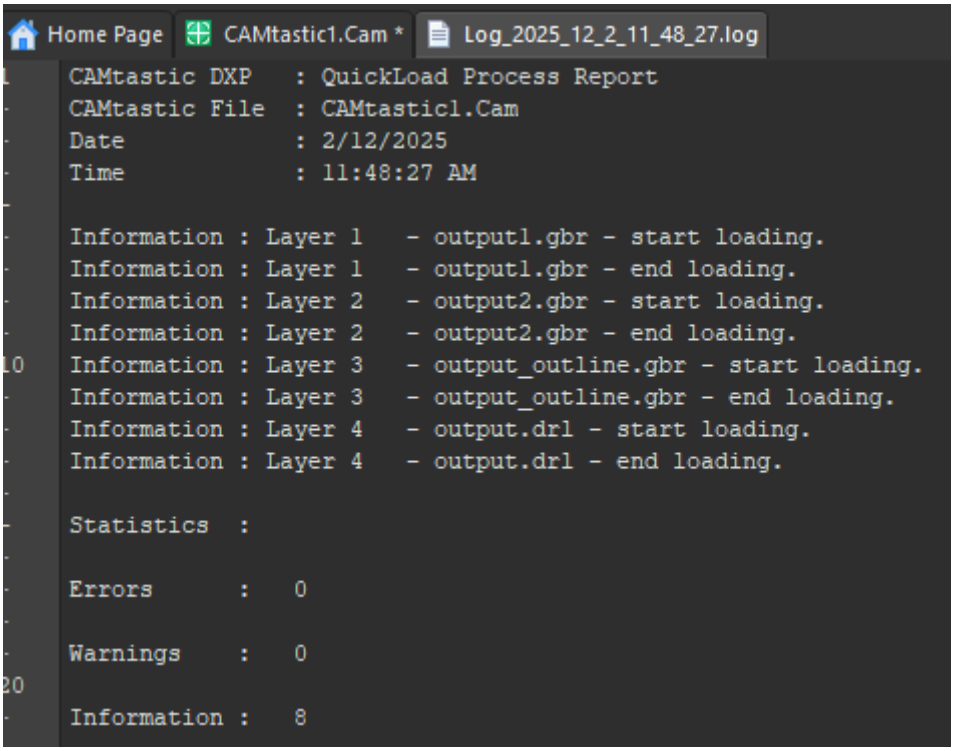


**Figure 10. Import Log Verification**

11. Verify successful import by doing the following:
    a. Confirm that all Gerber layers and drill files appear in the CAM document.
    b. Visually check that the board outline, copper layers, and drill hits look reasonable (no scaling/mirroring/offset).

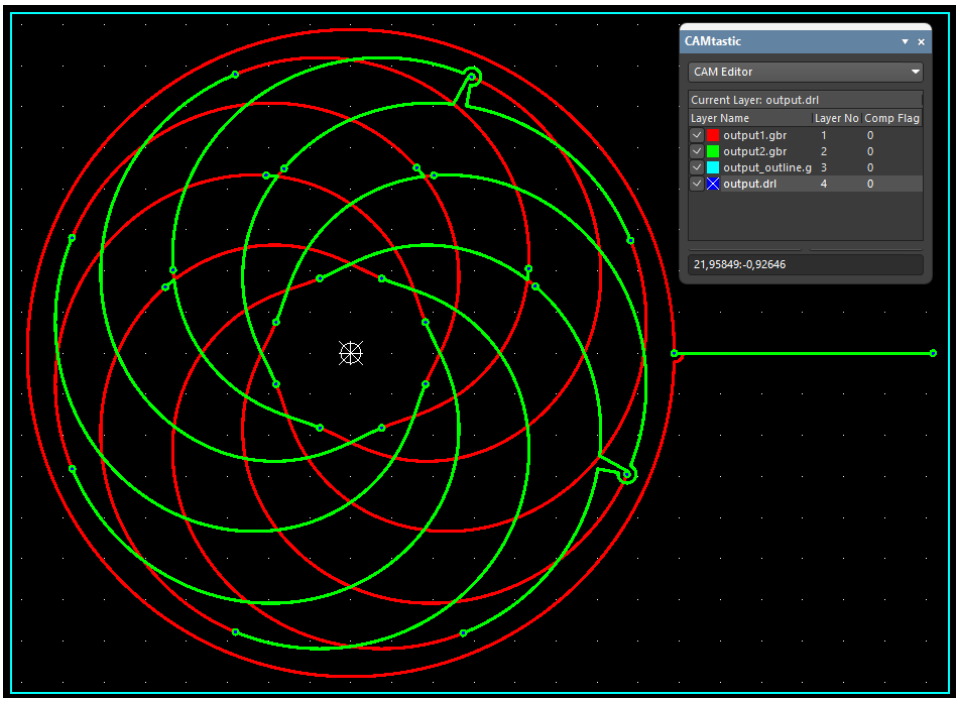


**Figure 11. Imported Gerber Layers and Drill Data**

# 2. Configuring the CAM Document

After import, the CAM document must be organized and verified before conversion to PCB. Correct layer mapping and physical layer order ensure that top/bottom and inner layers are interpreted properly and that drill/outline data aligns with copper features. Completing these checks in CAM reduces downstream rework during netlist extraction and PCB export.

Follow these steps to adjust the CAM file:

1. Set up layers by doing the following:

a. Go to Tables → Layers.

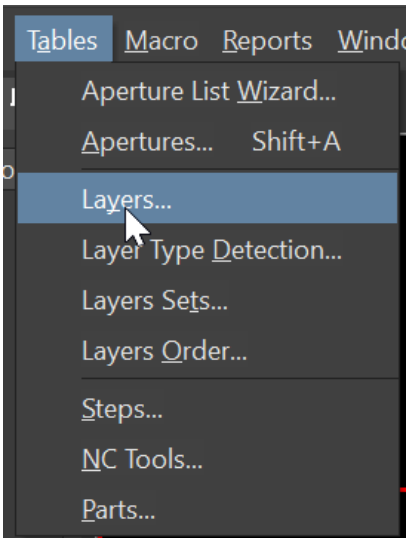


**Figure 12. Opening the CAM Layers Table**

b. Assign correct layers by:

i. Map each imported Gerber file to the correct CAM/PCB layer type (Top/Bottom Drill, Border, etc.).

ii. Click OK.

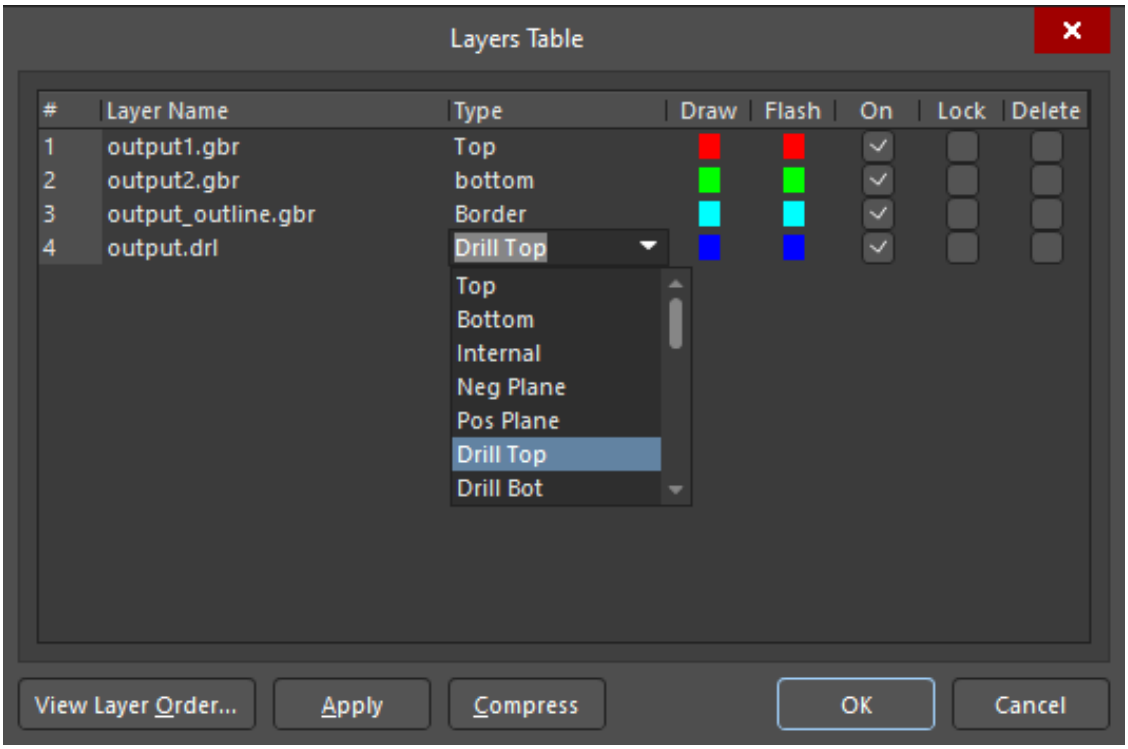


**Figure 13. Mapping Imported Gerber Files to Correct CAM/PCB Layer Types**

c. Set physical layer order by:
   i. Define the physical stack order.
   ii. Ensure the order matches the intended fabrication stackup.
   iii. Click OK.

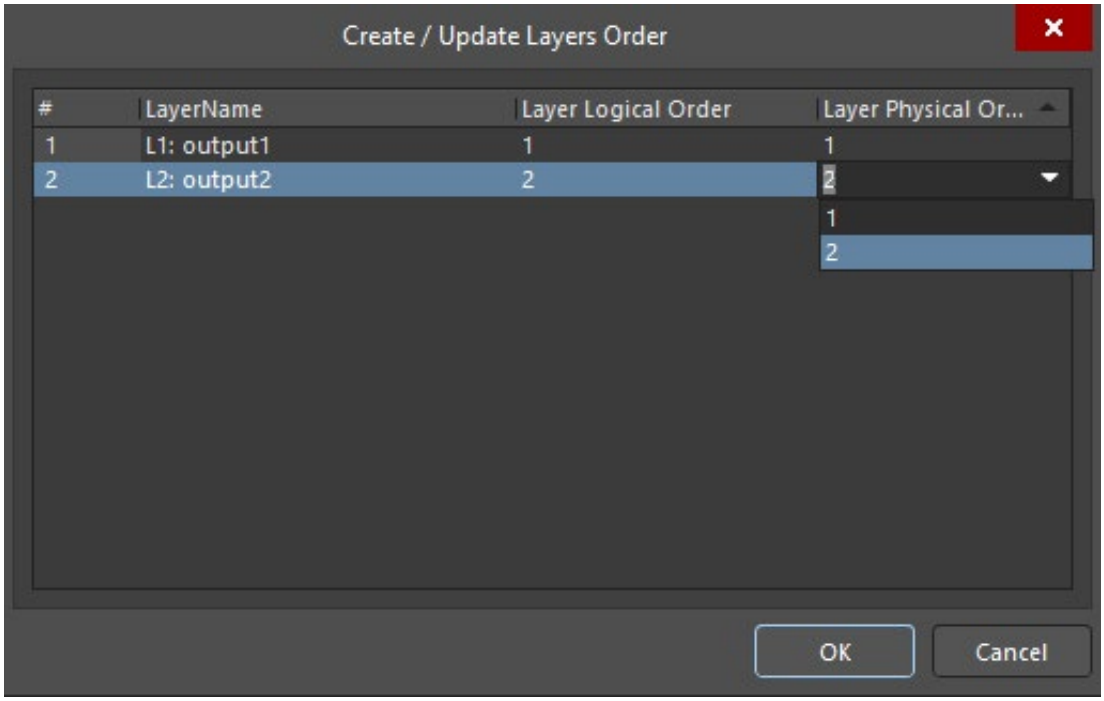


**Figure 14. Setting and Confirming the Physical Layer Order**

2. Extract the netlist by going to Tools → Netlist → Extract

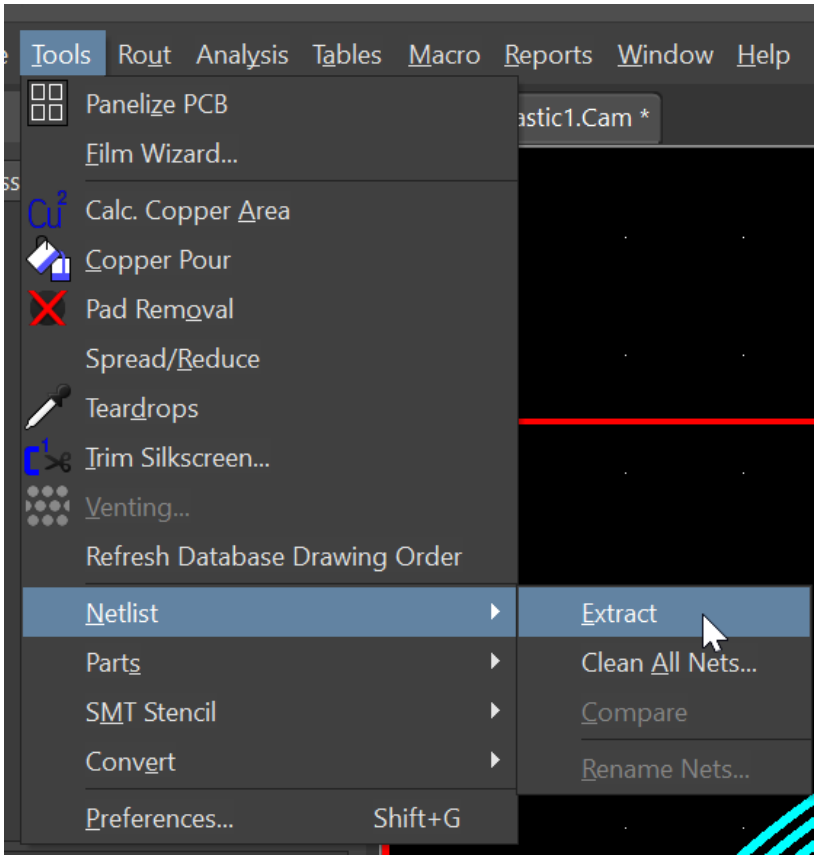


**Figure 15. Extracting the Netlist from CAM Data**

3. Export to PCB by doing the following:
   a. Use Export/Convert to PCB from the CAM environment by going to File → Export → Export to PCB.
   b. Create a new document from the imported data.

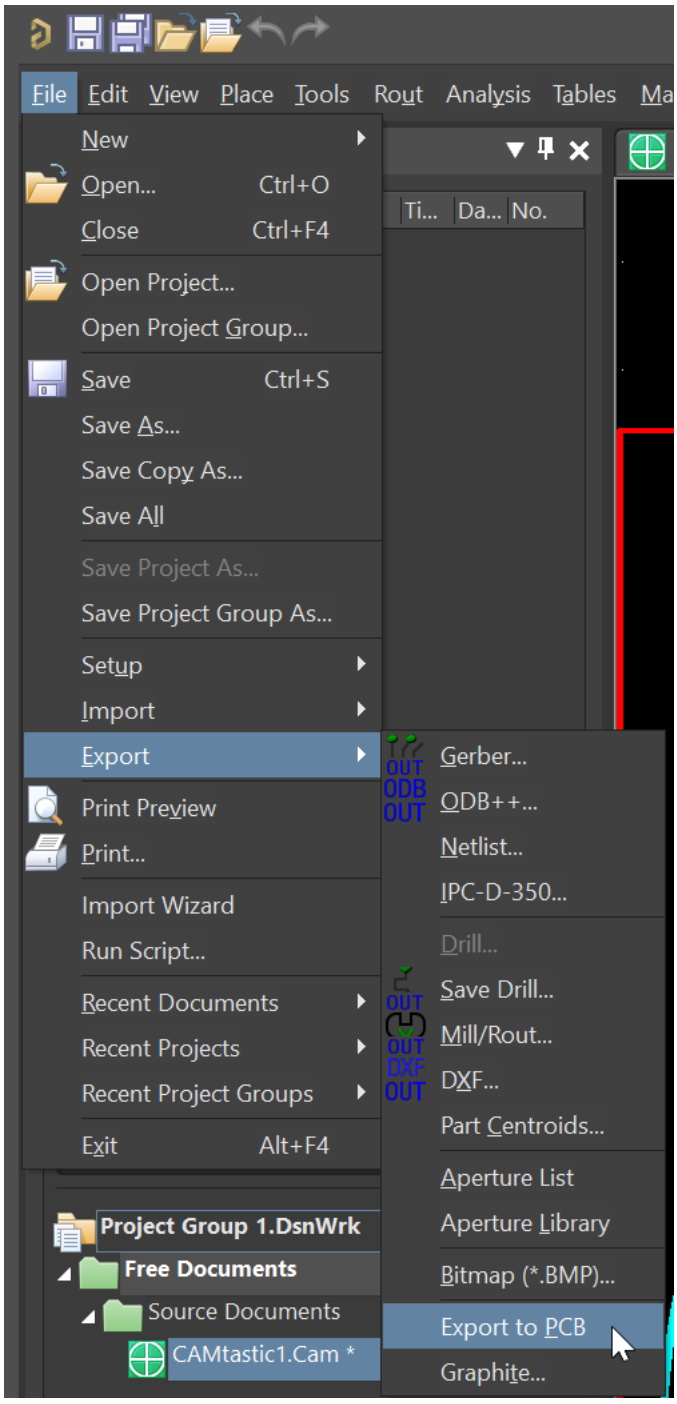


**Figure 16. Exporting CAM Data to a PCB Document**

# 3. Adjusting the PCB After Conversion

With the PCB document successfully generated from the imported Gerber data, the final phase involves optimizing the design for proper functionality and manufacturability. The imported data requires refinement to ensure that all design elements are correctly classified and configured according to industry standards and design rules. The following procedures address the conversion of imported pad objects to proper via structures, adjustment of via parameters to meet manufacturing specifications, and completion of final design tasks including net verification, component integration, and comprehensive design rule checking.

Follow these steps to transform the imported manufacturing data into a fully validated, production-ready PCB design:

1. Converting Pads to Vias by doing the following:

a. Open the Properties panel.
By default, via pads and drills are imported as combined "multilayer pads" with plated drill holes. While these can remain unchanged, converting them to the "via" class is recommended for proper design rule handling.

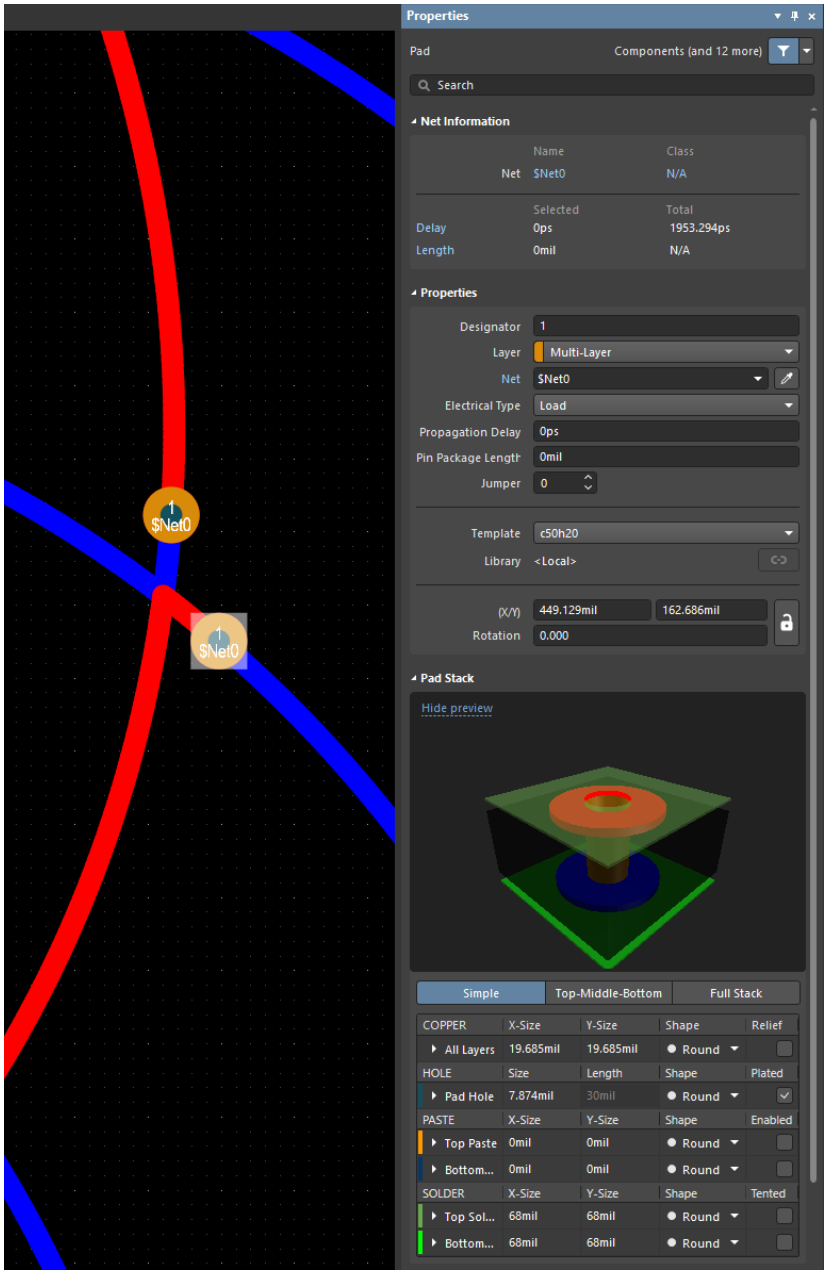


**Figure 17. Properties Panel View**

b. Filter selection to pads only by going to Filter icon → Pads.

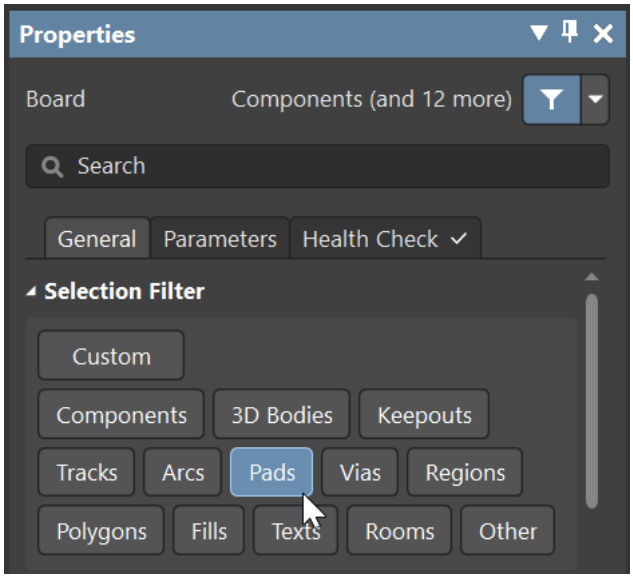


**Figure 18. Selection Filter Configured to Pads Only**

c. Select the entire imported design region to mark all pads.

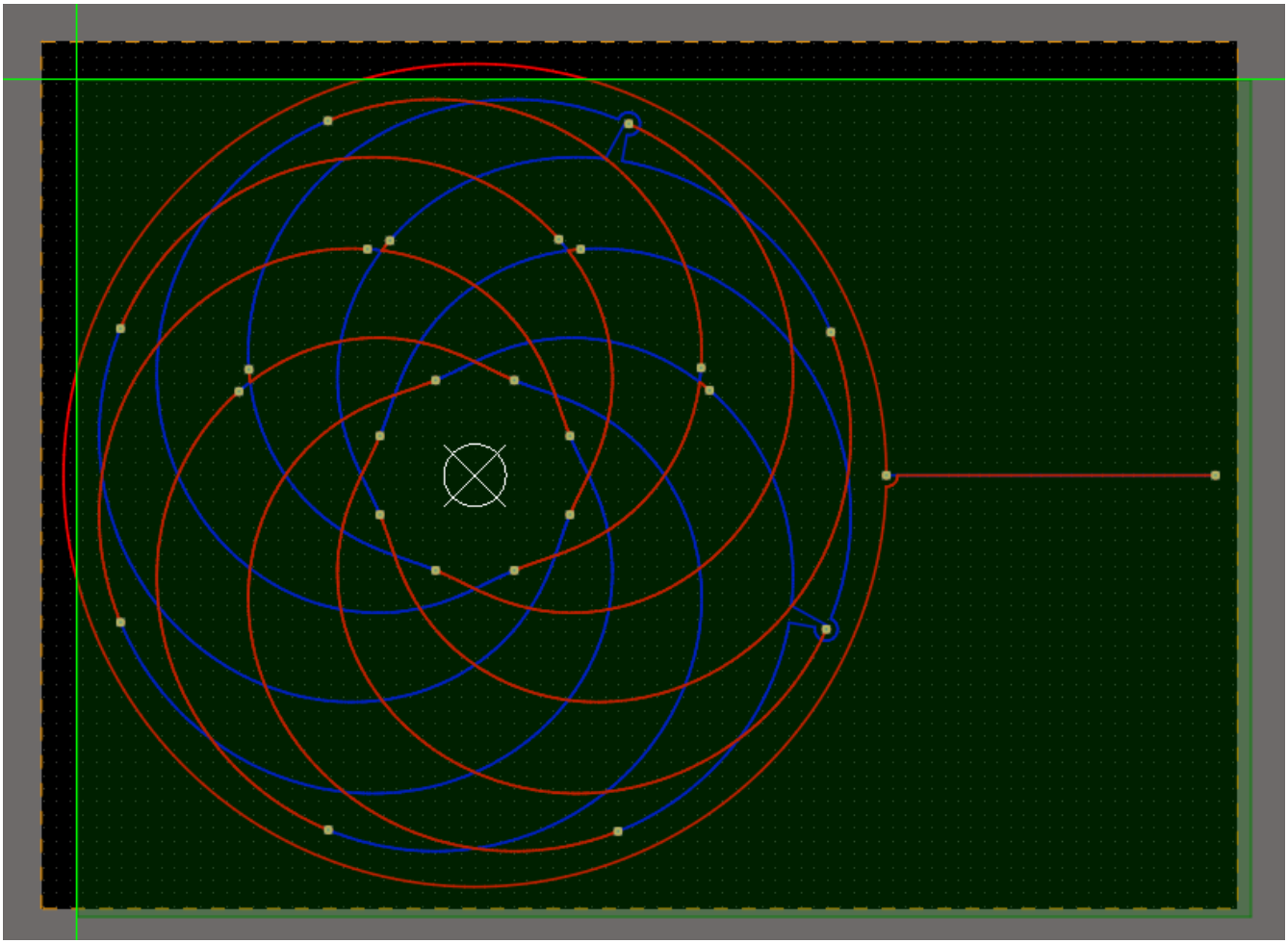

**Figure 19. Entire Imported Design Region Selected to Capture all Pad**

d. Convert free pads to vias by either:

- Go to Tools → Convert → Convert Selected Free Pads to Vias
- Hotkeys: T, V, P

2. Adjusting vias by doing the following:
   a. Filter selection to vias only by going to Filter icon → Vias.

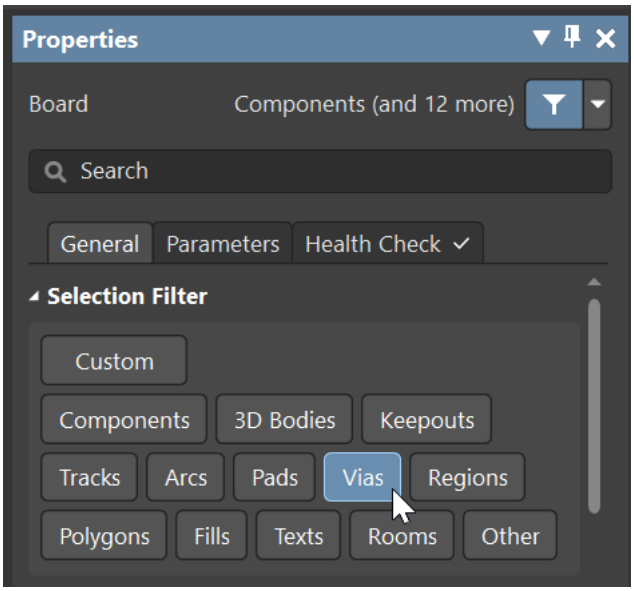


**Figure 20. Selection Filter Configured to Vias Only**

   b. Select all vias by using Ctrl+A (or select the region) to select all vias.

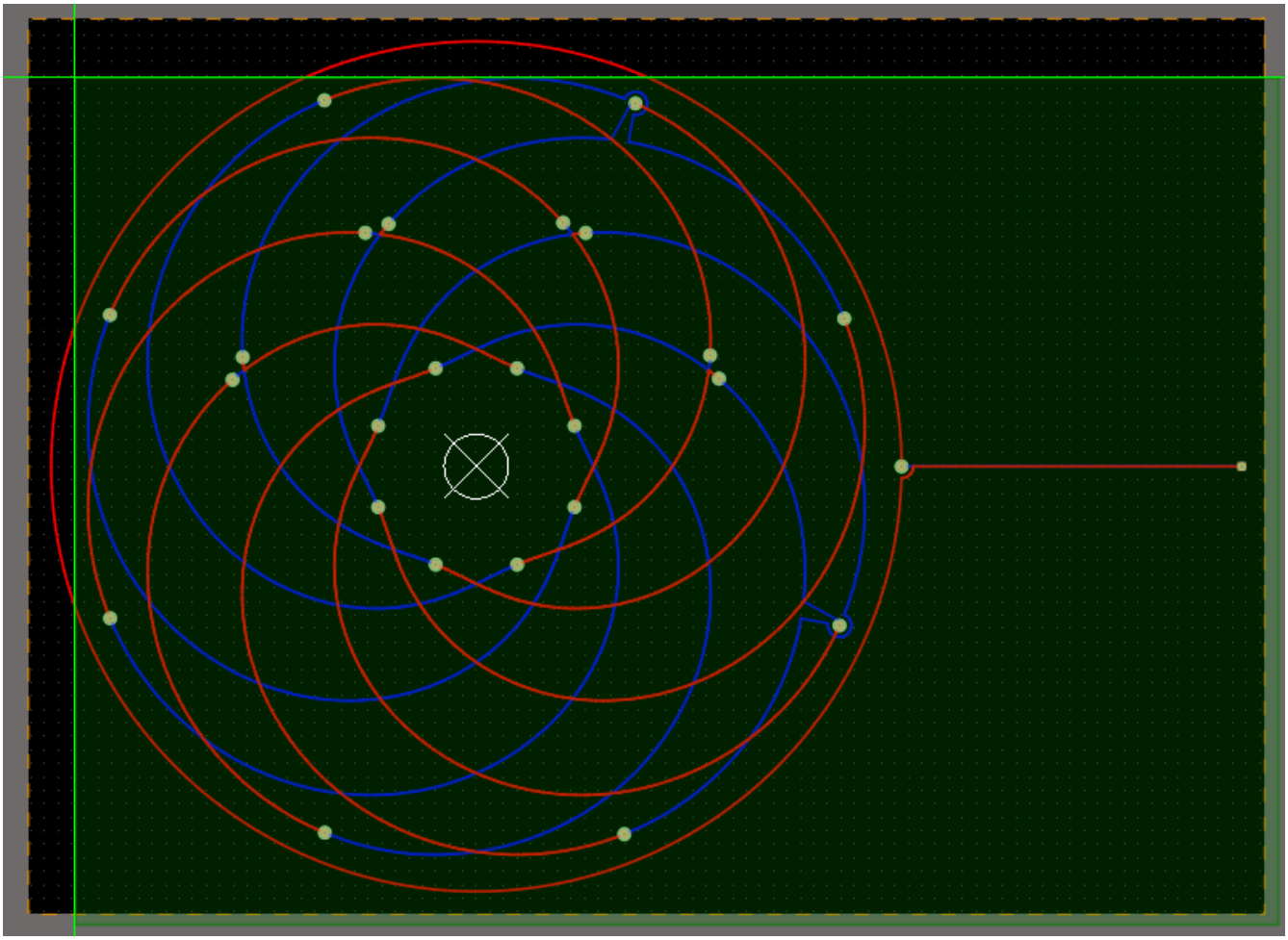

**Figure 21. Selecting all Vias in the PCB Document**

c. Adjust via parameters by the Properties panel to fit user needs, including:

- Via Stack
- Solder Mask Expansion
- Tenting (as needed)
- Any special via requirements (stitching, thermal, etc.)

Notes:

- Via diameter and drill should match the simulated/intended values.
- If you change via sizes, run DRC afterward to catch clearance/spacing violations.

3. Final cleanup and layout checks.
   After import/conversion, typical required tasks include:
   - Confirm and correct net names
   - Fix/align output headers/connectors
   - Copy coils or sensing structures into the correct chip layout template
   - Connect the sensing element to the IPS (or target) chip
   - Verify board outline and mechanical layers
   - Rebuild polygons (if present), repour copper as needed
   - Run DRC (Design Rule Check) and resolve all violations
4. Save the PCB document by:
   a. Go to File → Save
   b. Save the CAM document if you need traceability of the import step.

# 4. Revision History

| Revision | Date | Description |
|---|---|---|
| 1.00 | Jan 6, 2026 | Initial release. |

## 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


### 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/.

### Trademarks

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

© 2026 Renesas Electronics Corporation. All rights reserved.