



 Coupons APP Download Sign In Help Center

Help Center > PCB Files Pr... > How to export Altium PCB to gerber files

How to export Altium PCB to gerber files

 Last updated on Aug 18, 2025

This tutorial will guide you through generating Gerber and NC Drill files from Altium Designer 25 for manufacturing at JLCPCB.

The procedure is similar to earlier versions, but the interface and some options have been updated.

Design Rule setup

Before exporting any files, make sure your PCB layout follows the manufacturing capabilities of JLCPCB.

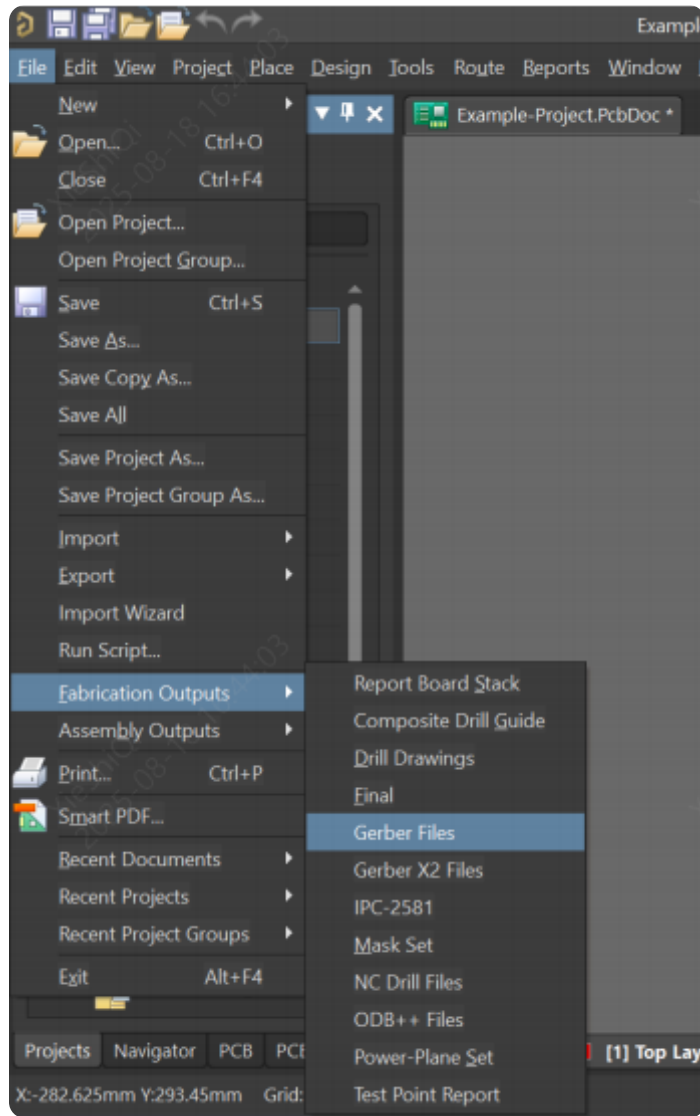
You can find the latest specifications on the official JLCPCB Capabilities page: <https://jlcpcb.com/capabilities>

Generate Gerber Files

Open your PCB design tab in Altium Designer 25.

1. Go to File → Fabrication Outputs → Gerber Files.

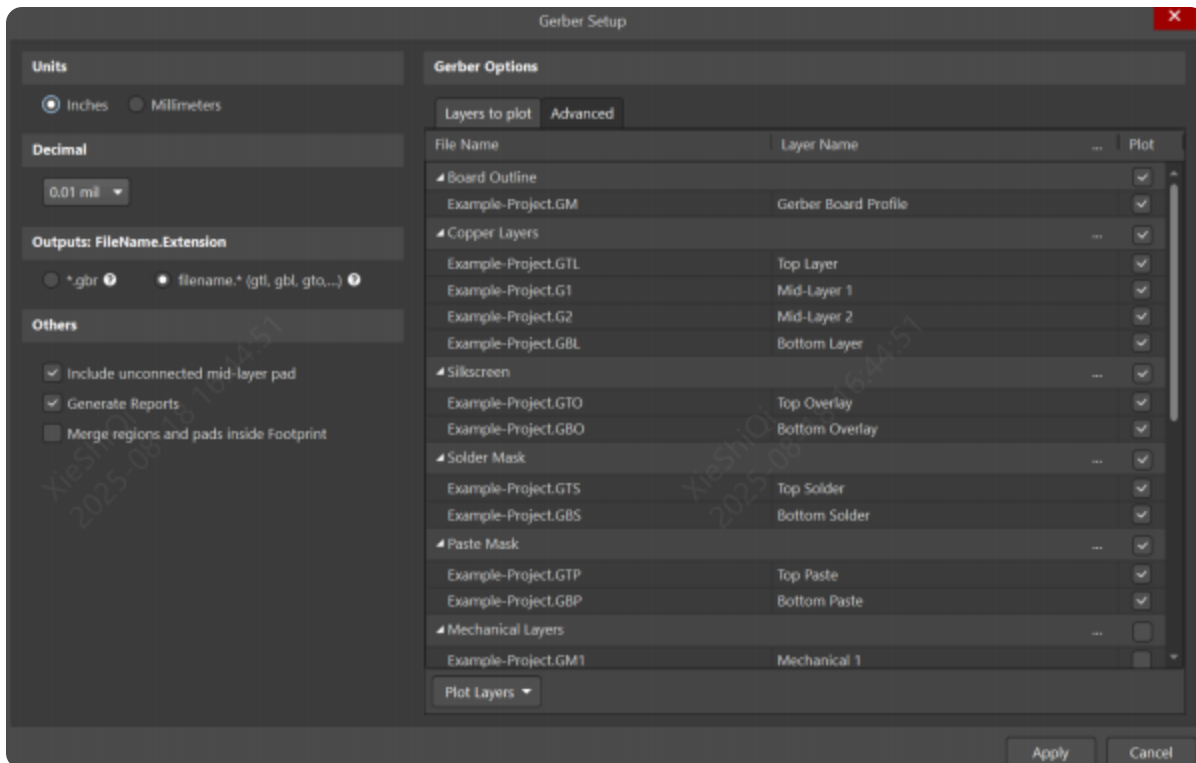
[Get Coupon >](#)[Live Chat >](#)
Home
Cart
Files
Message
Me



2. In the Gerber Setup dialog:

Get Coupon > Live Chat >

Home Cart Files Message Me



General Tab

- Units: Inches (in)
- Decimal: 0.01mil
- Outputs: filename
- Include unconnected mid-layer pad
- Generate Reports

Layers to plot

- Board Outline (.GM)
- Top Layer (.GTL)
- Inner Layers (.G1, .G2, ...)

[Get Coupon >](#)

[Live Chat >](#)

 Home

 Cart

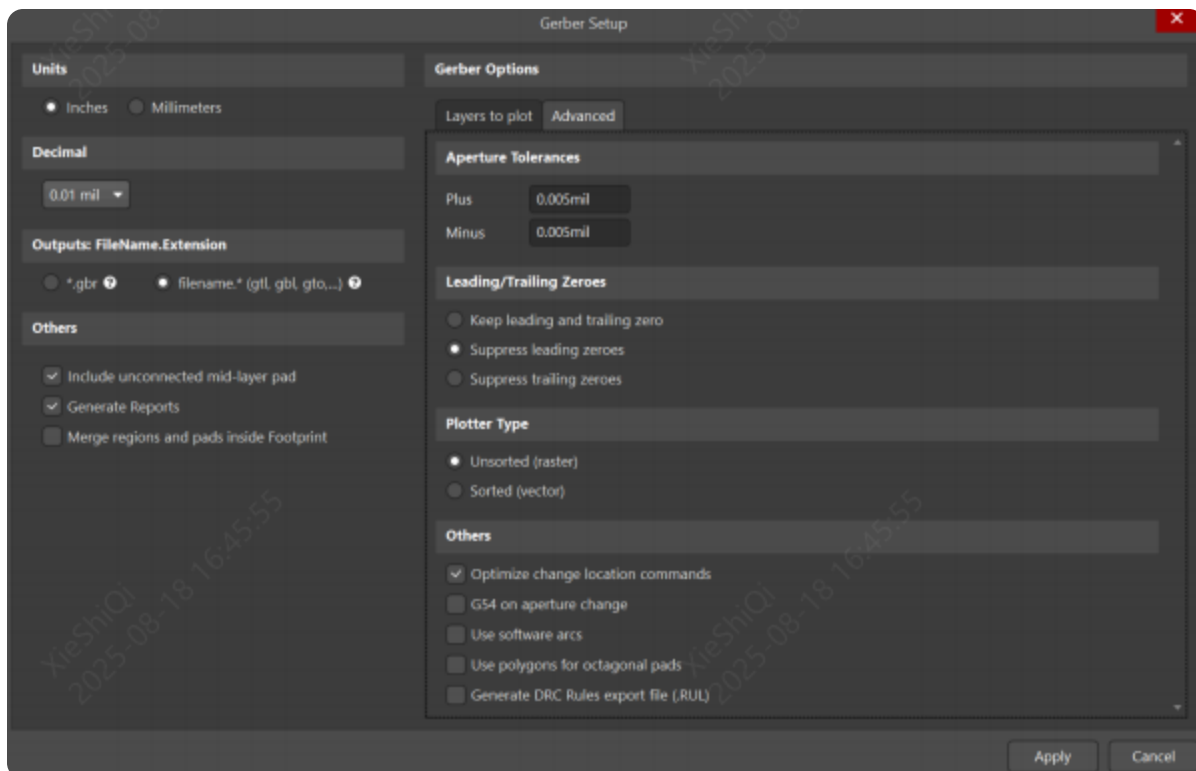
 Files

 Message

 Me

- Bottom Overlay (.GBO)
- Top Solder (.GTS)
- Bottom Solder (.GBS)
- Top Paste (.GTP)
- Bottom Paste (.GBP)

3. In the Advanced Gerber options dialog:



Select options:

- Aperture Tolerances: ($\pm 0.005\text{mil}$)
- Leading/Trailing Zeroes: Suppress leading zeroes
- Plotter Type: Unsorted (raster)
- Others: Optimize change location commands

[Get Coupon >](#)

[Live Chat >](#)

 Home

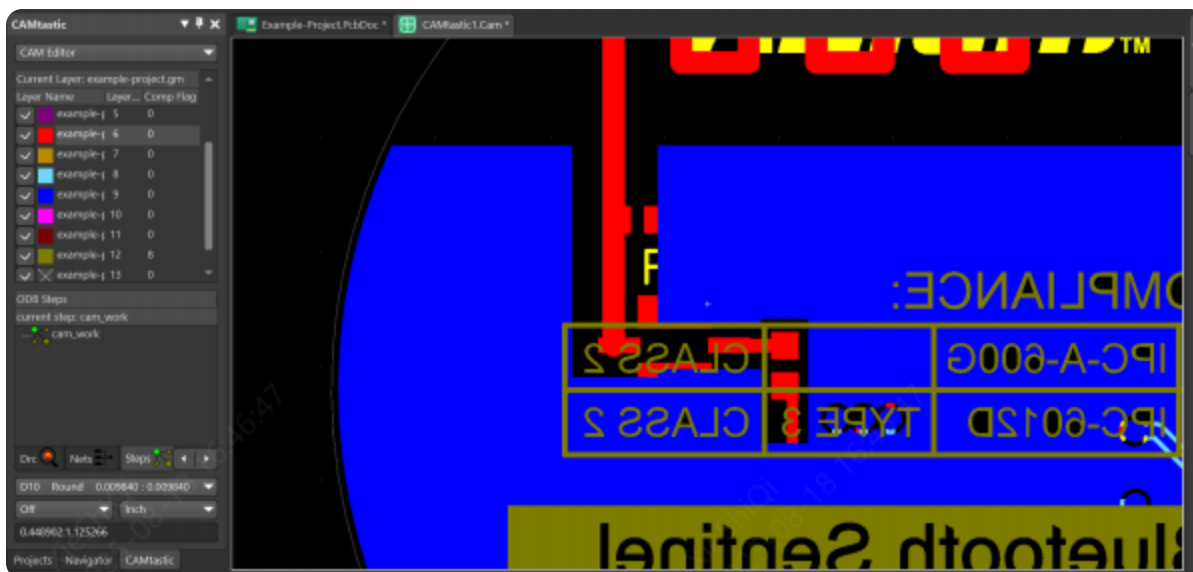
 Cart

 Files

 Message

 Me

check the generated gerber files.



Generate NC Drill files

1. Go to File → Fabrication Outputs → NC Drill Files

[Get Coupon >](#)

[Live Chat >](#)

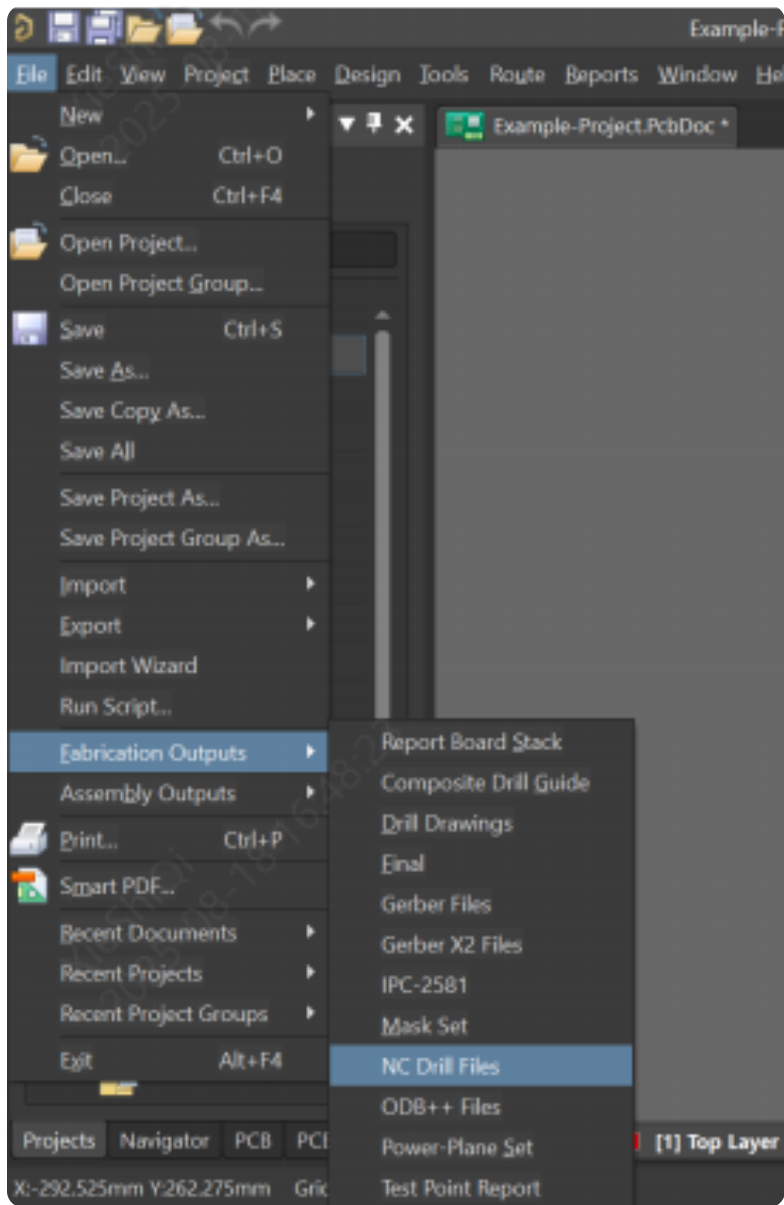
 Home

 Cart

 Files

 Message

 Me



2. In the NC Drill Setup dialog:

[Get Coupon >](#)

[Live Chat >](#)

Home

Cart

Files

Message

Me

**Select options:**

- Units: Inches
- Format: 2:5
- Zero Suppression: Keep leading and trailing zeroes

[Get Coupon >](#)[Live Chat >](#)

Home



Cart



Files



Message

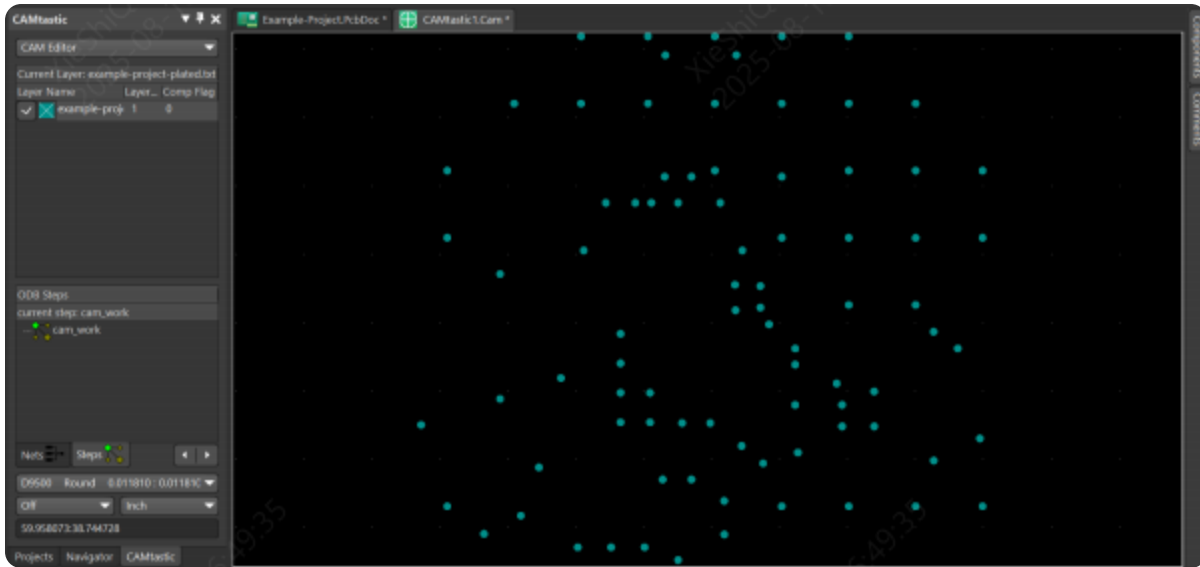


Me

- Leave other options unchecked unless required.

3. Click OK to generate the NC Drill files.

After the files have been generated, CAMtastic windows is opened automatically, so you can check the generated NC drill files.



Collect all files and put them together into a single zip/rar file.


Altium has published a guide on producing those files here:

<https://www.altium.com/documentation/altium-designer/preparing-manufacturing-data-with-output-jobs>

If everything looks OK, upload the zip file to the JLCPCB order page.

Altium Designer version: 25.8.1.

Author: Milos Ilic



New User Bonus

Register and get

[Get Coupon >](#) [Live Chat >](#)

Home Cart Files Message Me



Was this article helpful? Yes No

- Products +
- Support +
- Company +
- Electronics +
- Mechanical +



USD v

Download JLCONE APP

- Windows
- MAC
- Android
- IOS

CONNECT WITH US



© 2026 JLCPCB.COM All Rights Reserved [Privacy Policy](#) [Terms & Conditions](#) [Cookies Policy](#)

Get Coupon >
Live Chat >

- Home
- Cart
- Files
- Message
- Me

[Get Coupon >](#)

[Live Chat >](#)


Home


Cart


Files


Message


Me