How to export CircuitMaker 2 PCB to Gerber and drill files

Gerber and NC Drill setup

First, we need to configure some options. Once the settings are configured, CircuitMaker will remember the settings for a specific project, which means you don't need to set these options every time before you generate the files.

Right click on the project name (the **.PrjPcb** file) on the **Projects** panel, a pop-up menu will appear, click **Generate Outputs**.
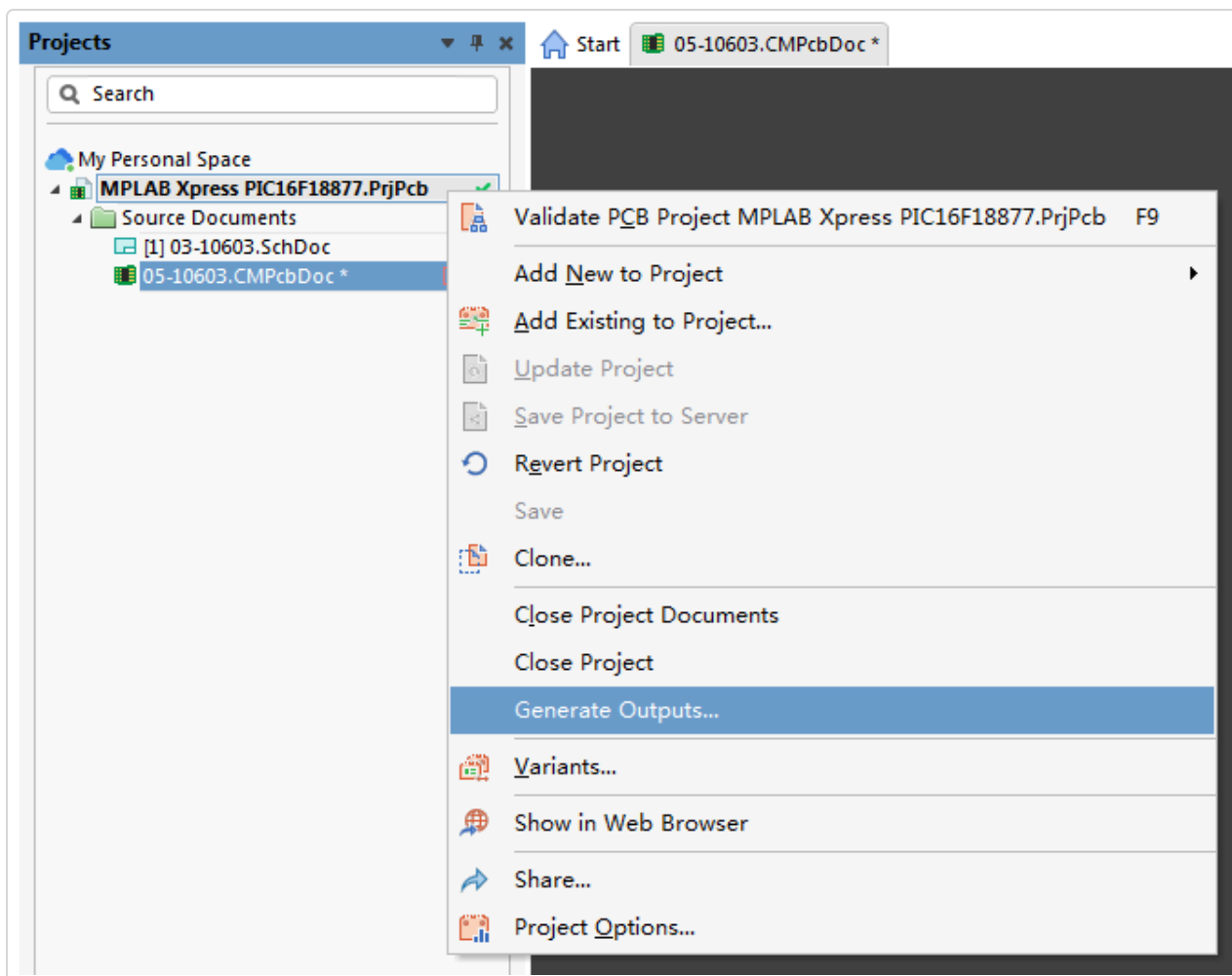

Figure 1. Generate Outputs menu item

Some Outputer are listed in the **Generate output files** dialog, click **Configure** for Gerber Files.
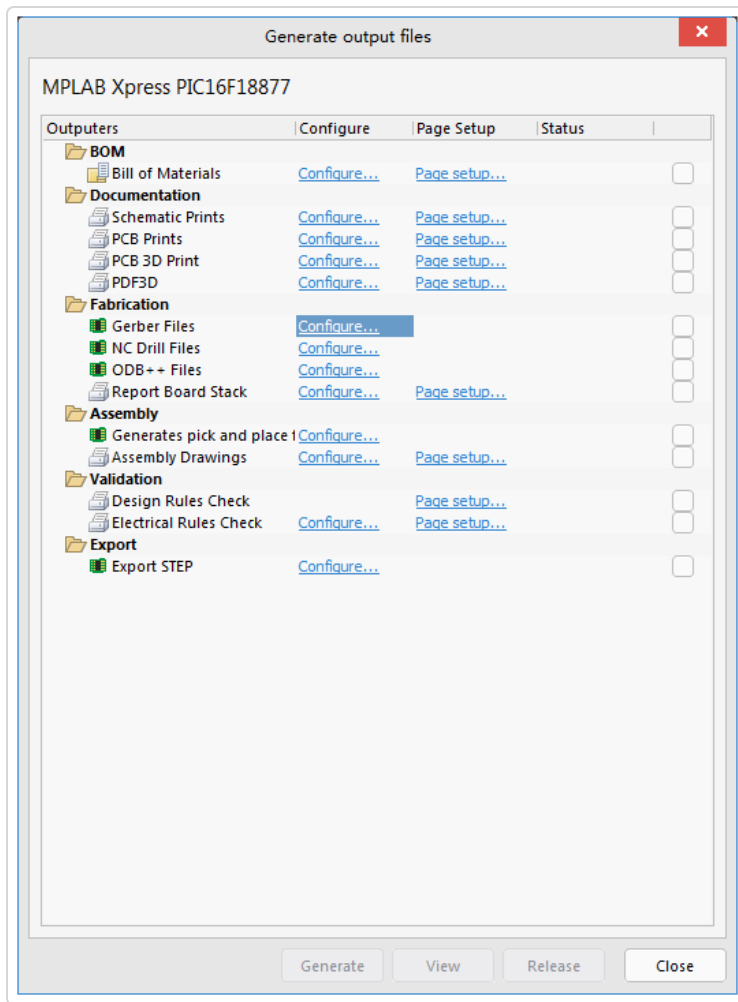
Figure 2. Configure Gerber options

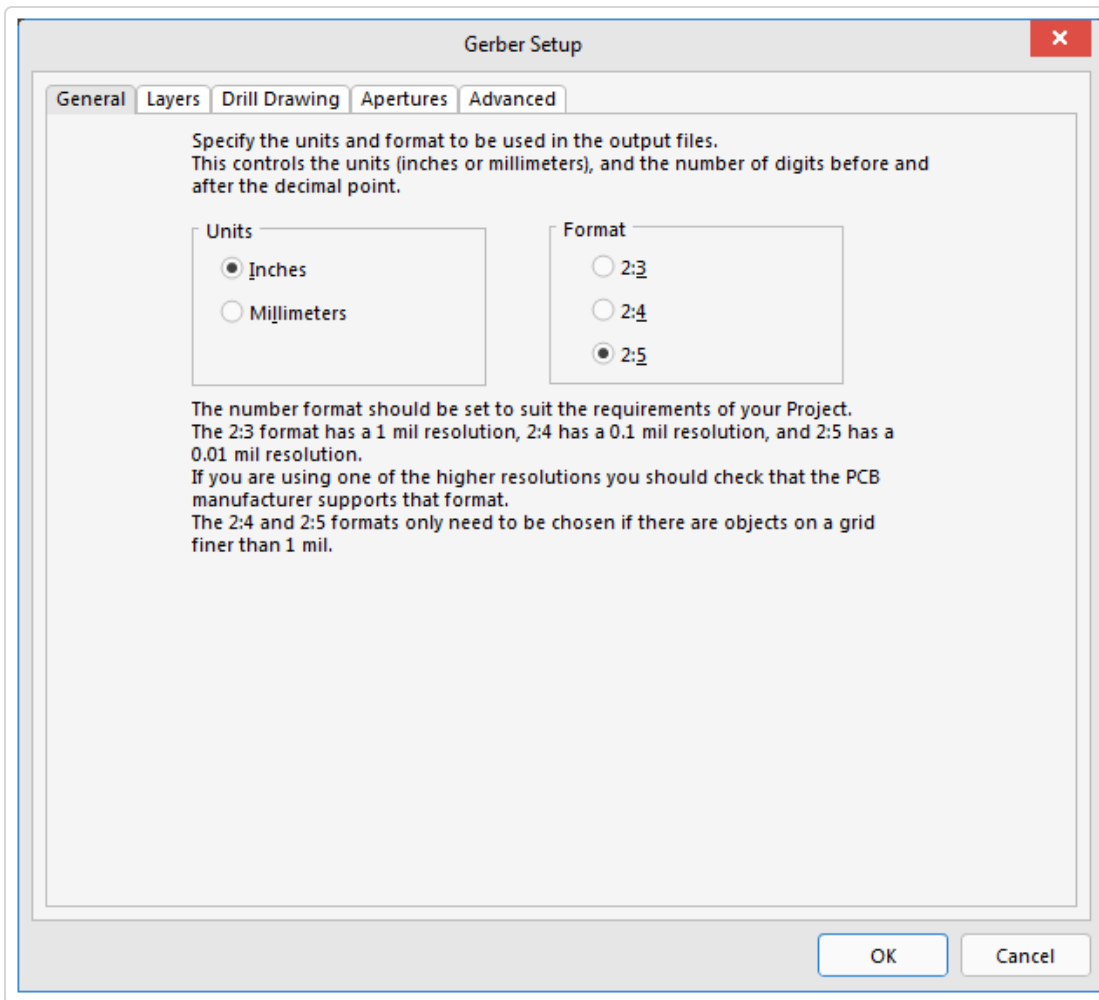A window will appear that allows you to specify some options.

General tab

Figure 3. General tab setting

Layers tab

Select necessary layers, for example, for a 2-layer design, you need to select the following layers:

1. Top Overlay

2. Top Paste

3. Top Solder

4. Top Layer

5. Bottom Layer

6. Bottom Solder

7. Bottom Paste

8. Bottom Overlay

9. Outline (Note that different Mechanical layer can be used for outline, in this example, it's M2)

If it's a 4-layer or 6-layer design, don't forget to select the inner layers.

Warning   Please don't check Keep-Out Layer (it will be exported to a .GKO file), our system looks for .GKO and may grab it as the outline. The Keep-Out Layer is used to put objects that blocks or prevents the placement of copper objects within an area.
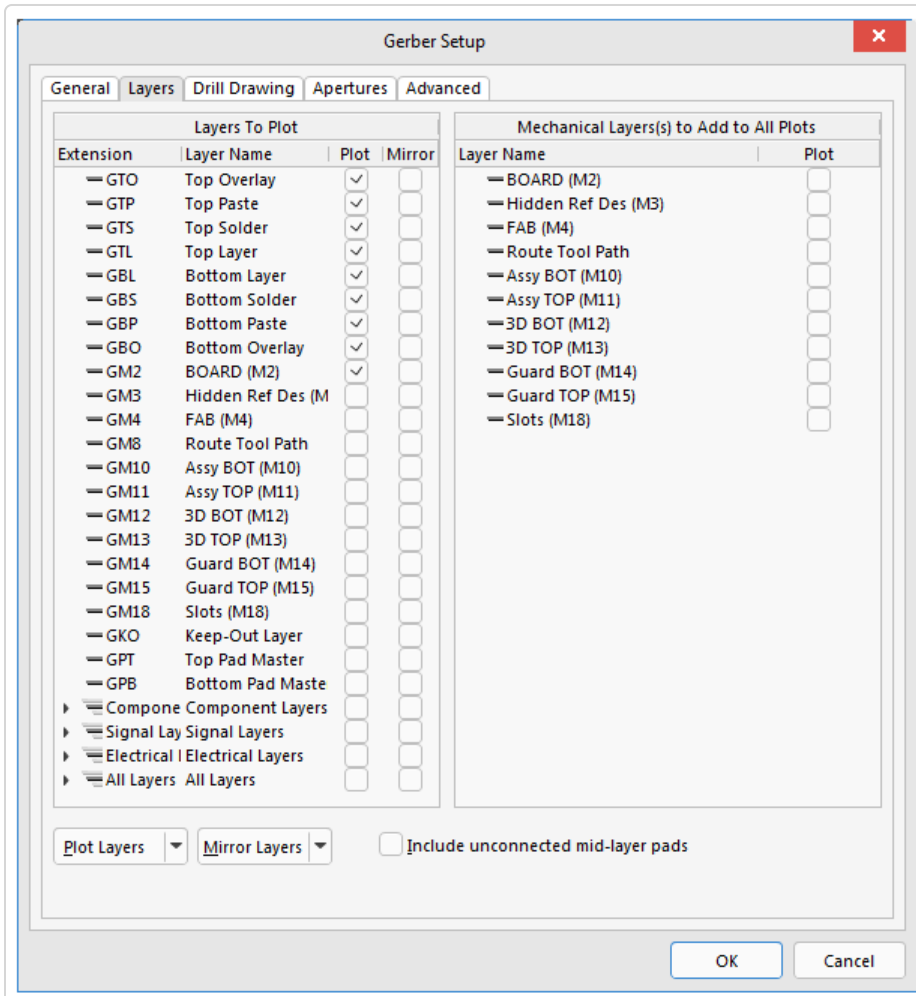

Figure 4. Layers tab setting

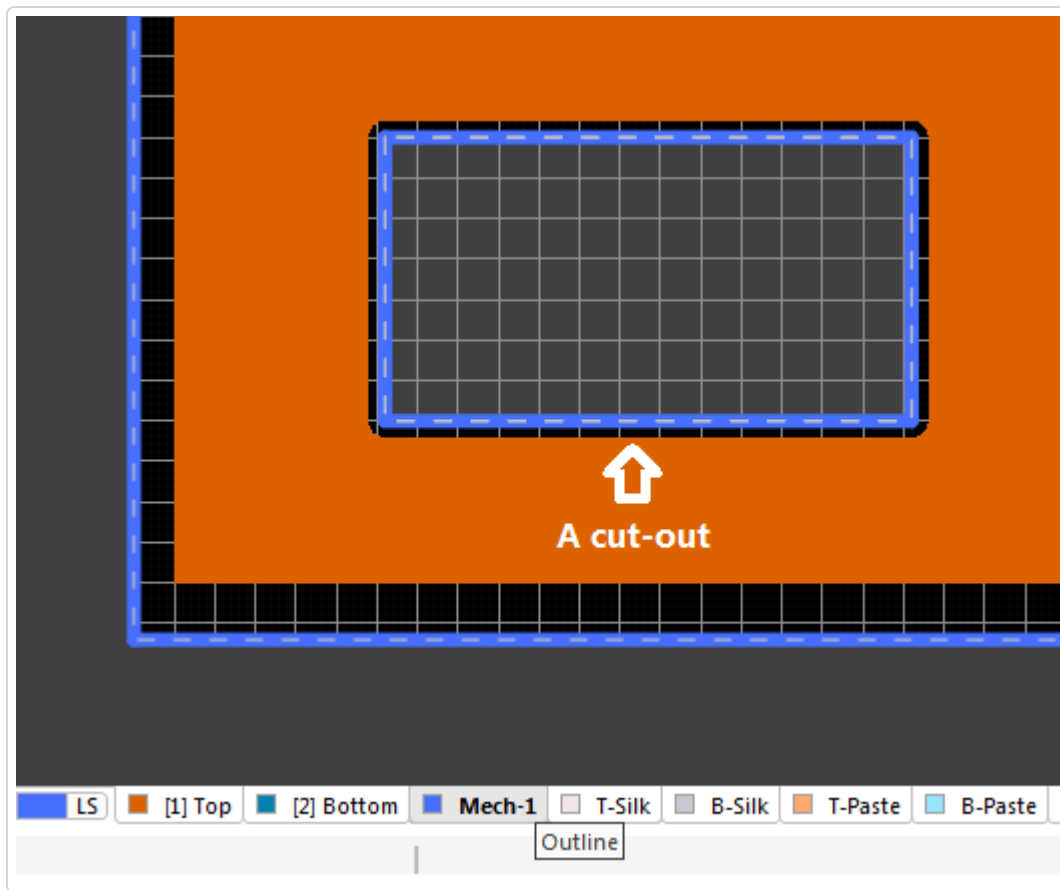To have a cutout (remove all material), just draw the shape on **outline** layer.

Figure 5. A cut-out
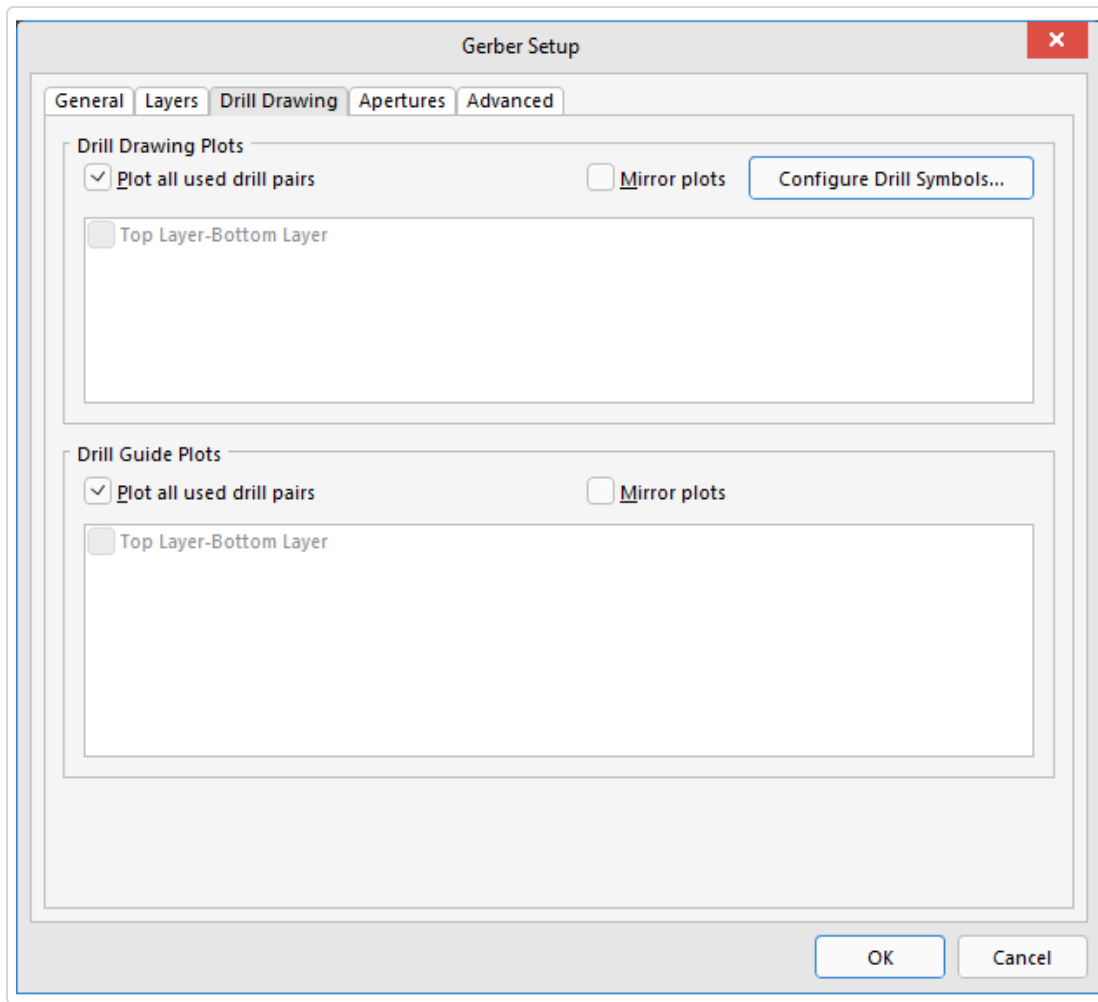
Drill Drawing tab

Figure 6. Drill Drawing tab setting

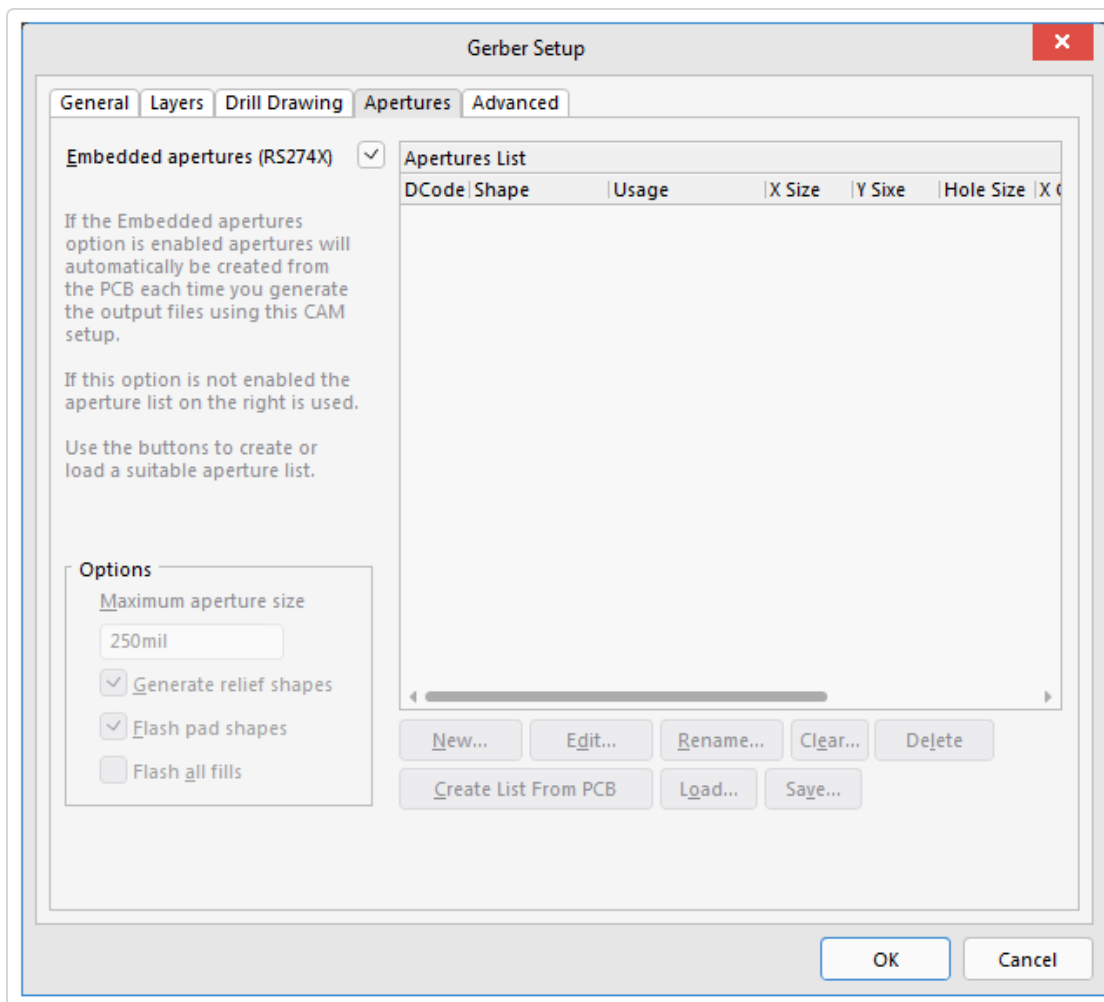## Apertures tab

Check **Embedded apertures (RS274X)**
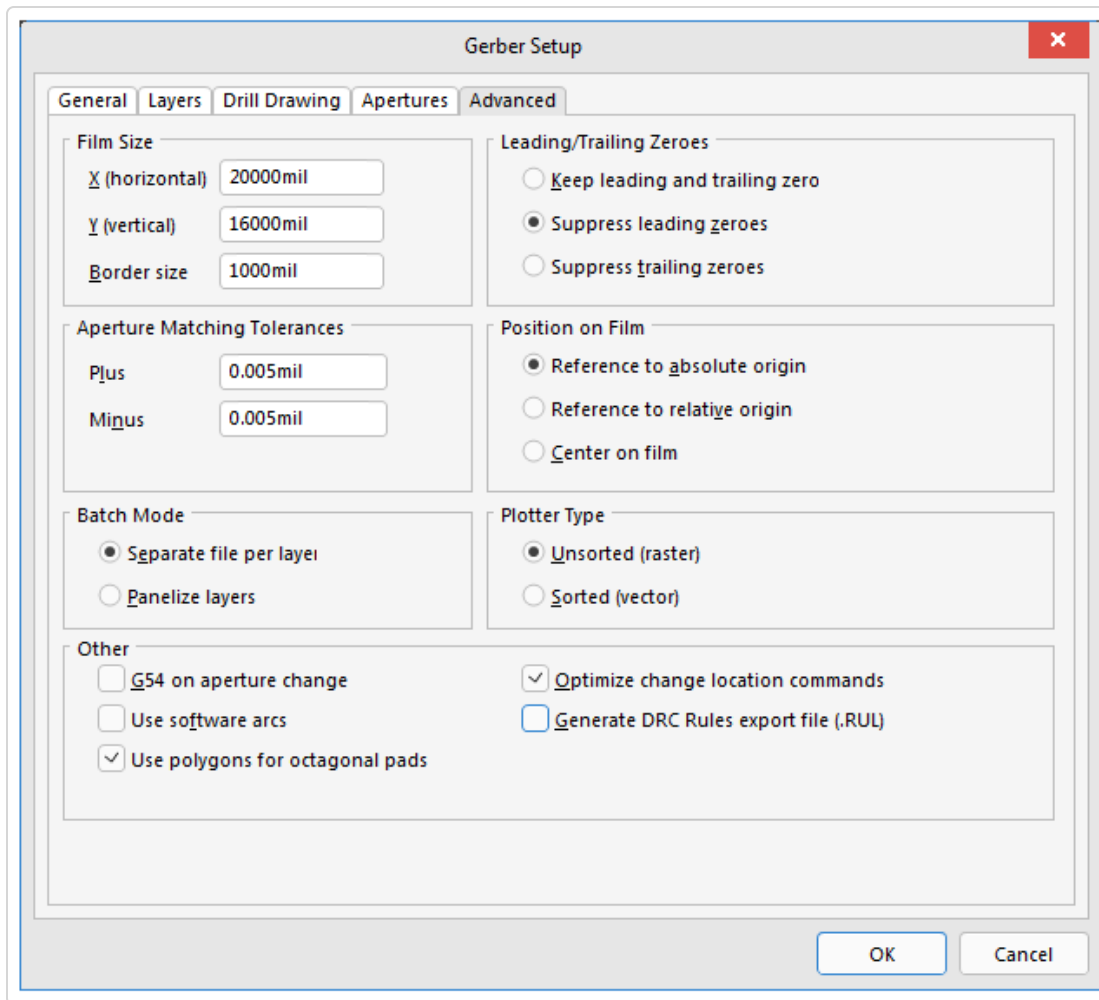
Figure 7. Apertures tab setting

Advanced tab

Figure 8. Advanced tab setting

Click **OK** button to finish the Gerber Setup.

NC Drill Setup

Click on **Configure** on the right side of **NC Drill Files**.

- Use **2:4** for format

- Check **Suppress leading zeroes**

- Check **Reference to absolute origin**

- Check **Generate separate NC Drill files for plated & non-plated holes**

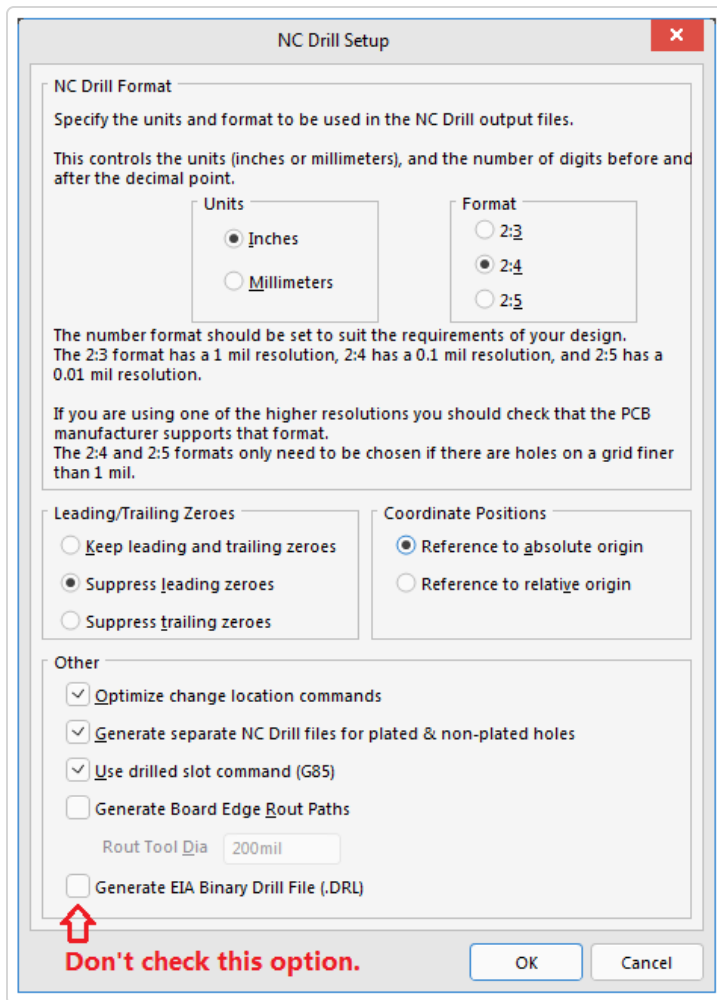- Check **Use drilled slot command (G85)**

Figure 9. NC Drill Setup

Generating files

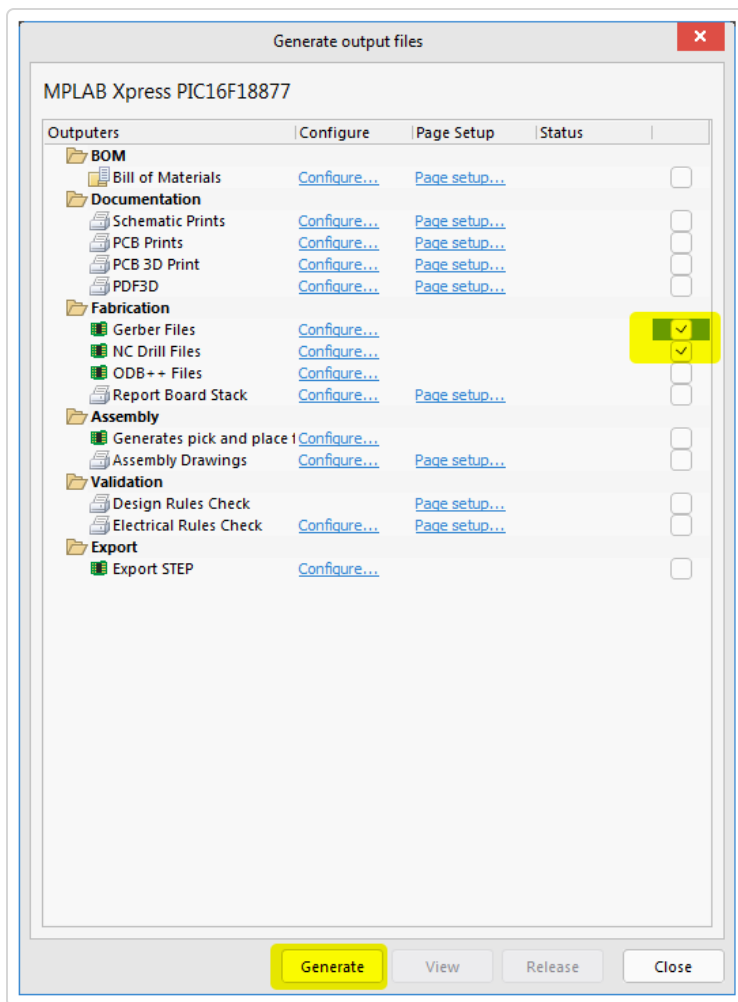Check both **Gerber Files** and **NC Drill Files**, click **Generate** button.

Figure 10. Check Gerber and Drill then Generate

CircuitMaker will say "The project has modifications…", just click **Save and Commit changes**.
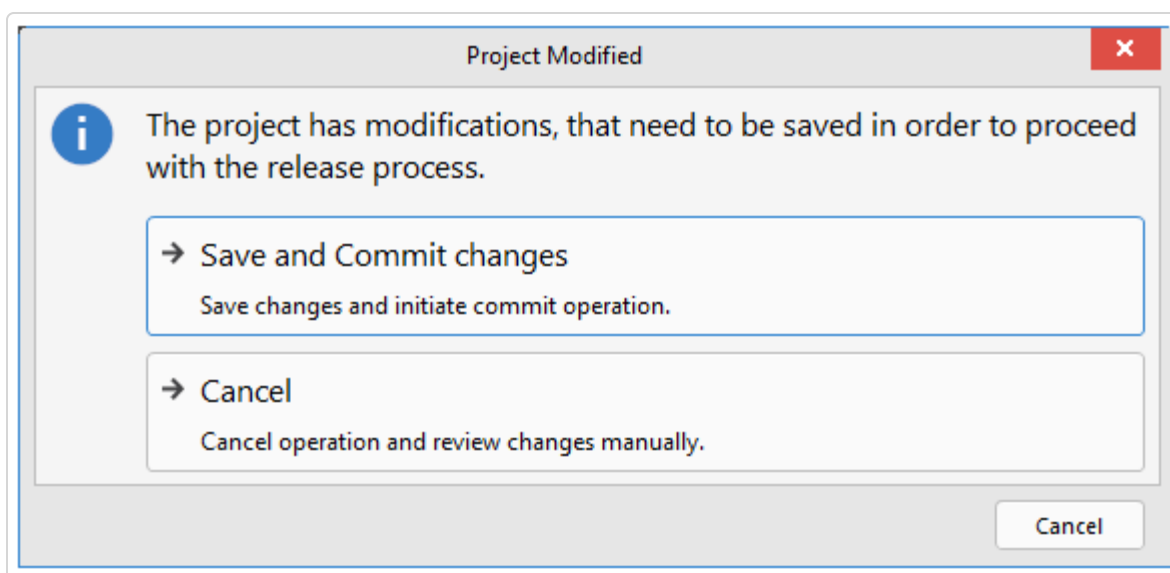

Figure 11. Save and Commit changes

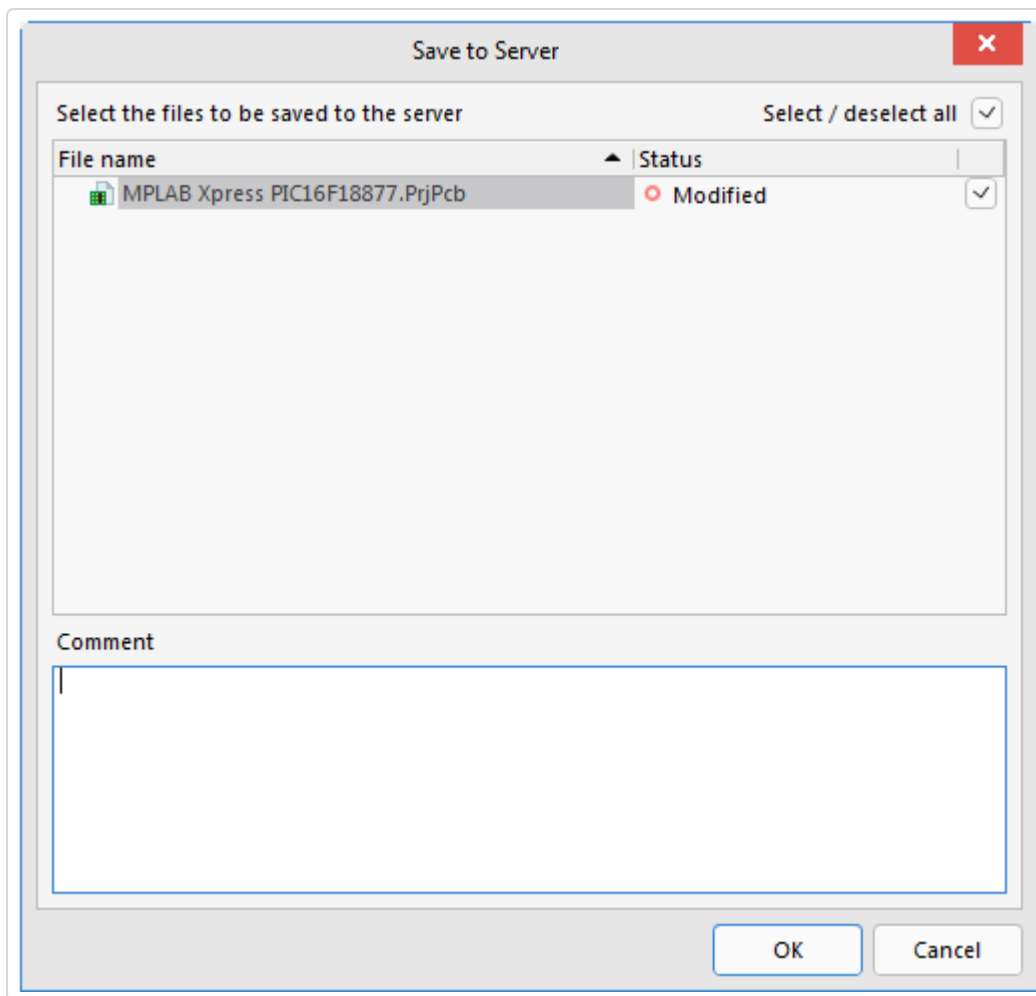Then click **OK** in the **Save to Server** dialog.

Figure 12. Save to Server

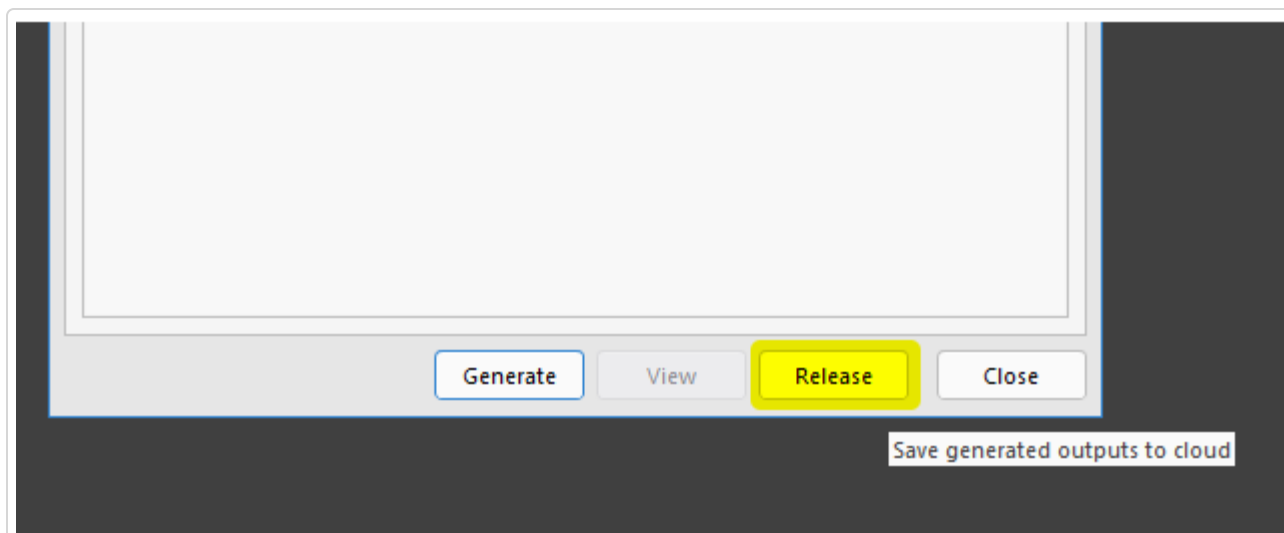Click the **Release** button, this will save the generated files to cloud.

Figure 13. Release

A window called **Confirm Release** will appear, you can write a release note if you want, then click **OK**.
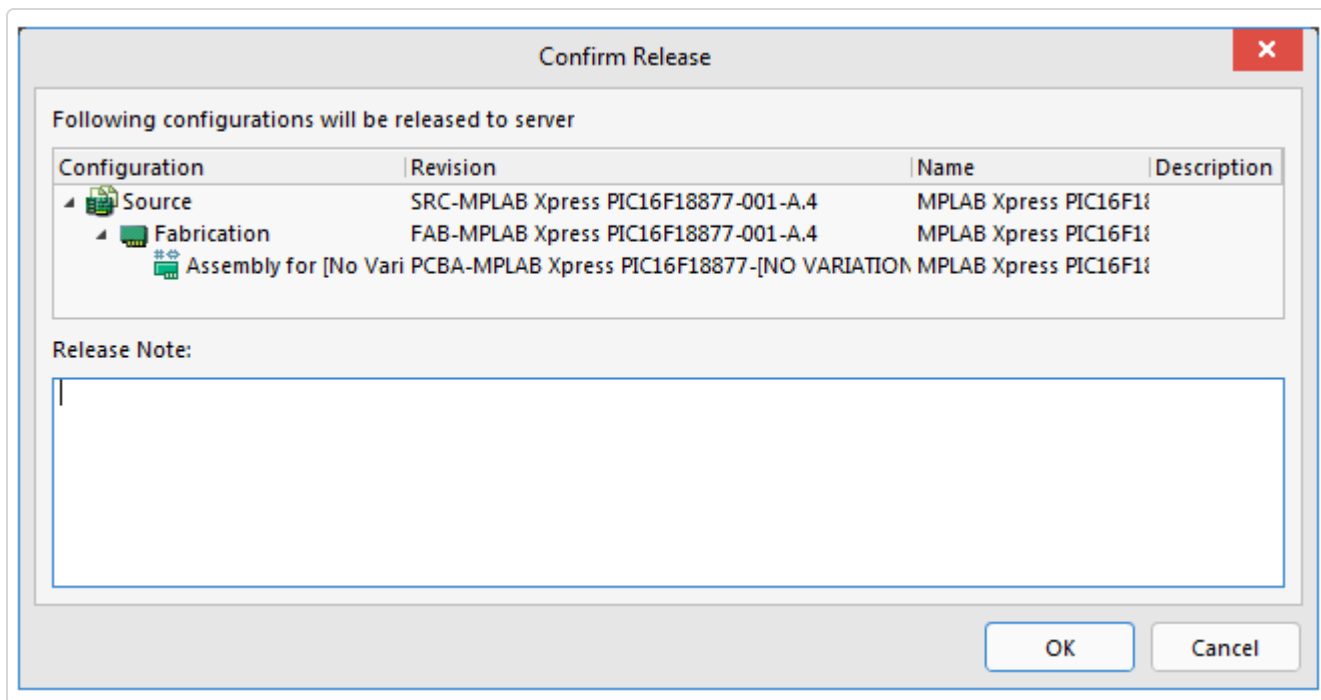

Confirm Release

Following configurations will be released to server

Configuration	Revision	Name	Description
Source	SRC-MPLAB Xpress PIC16F18877-001-A.4	MPLAB Xpress PIC16F18	
Fabrication	FAB-MPLAB Xpress PIC16F18877-001-A.4	MPLAB Xpress PIC16F18	
Assembly for [No Vari	PCBA-MPLAB Xpress PIC16F18877-[NO VARIATION	MPLAB Xpress PIC16F18	

Release Note:

OK   Cancel

Figure 14. Confirm Release

A new windows will say "Project has been released successfully, open in Web Browser."

Project Release

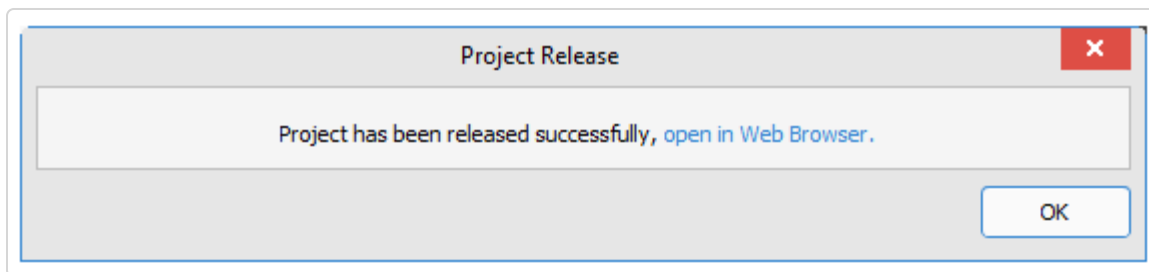Project has been released successfully, open in Web Browser.

OK

Figure 15. Released

Click on **open in Web Browser**, this will open the released files in a web browser. Click **Hide Files** to show the file, then click the download icon to download the Gerber and drill files.
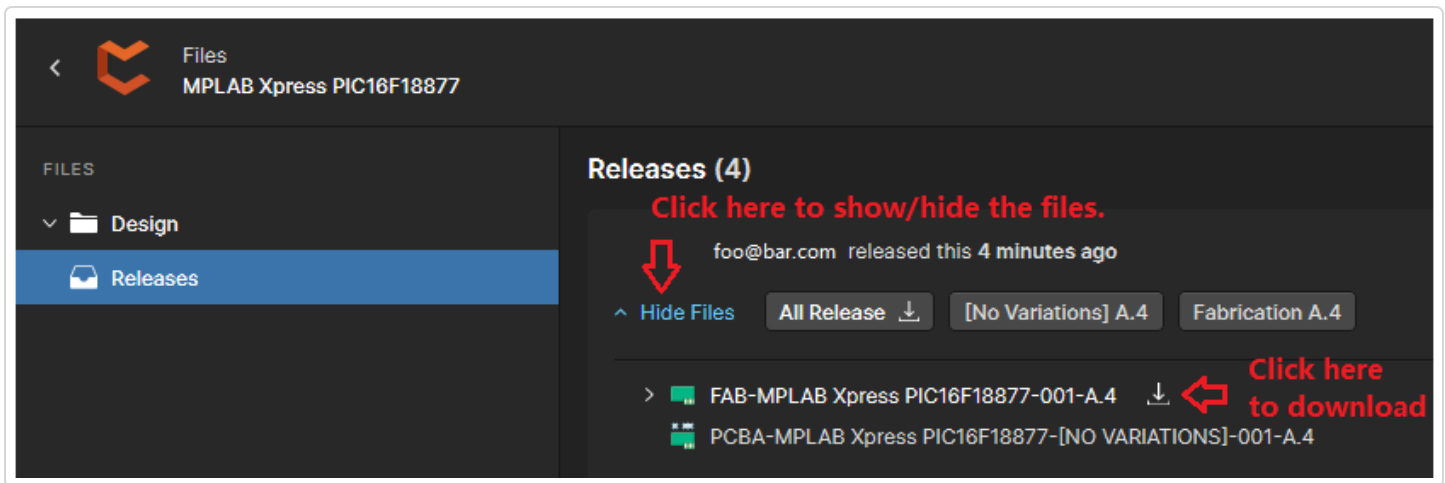
Figure 16. Released files in web browser

Verify the files

Before uploading your Gerbers and drill file to JLCPCB for production, it's highly recommended to cross-check the generated files with a 3rd-party Gerber viewer.

If you find any issues, fix them and export the Gerber/Drill files and check them in the Gerber viewer again.

There are some nice Gerber viewers here and there, just use the one you feel handy.

- Gerbv (http://gerbv.geda-project.org/)

- tracespace view (https://tracespace.io/view/)
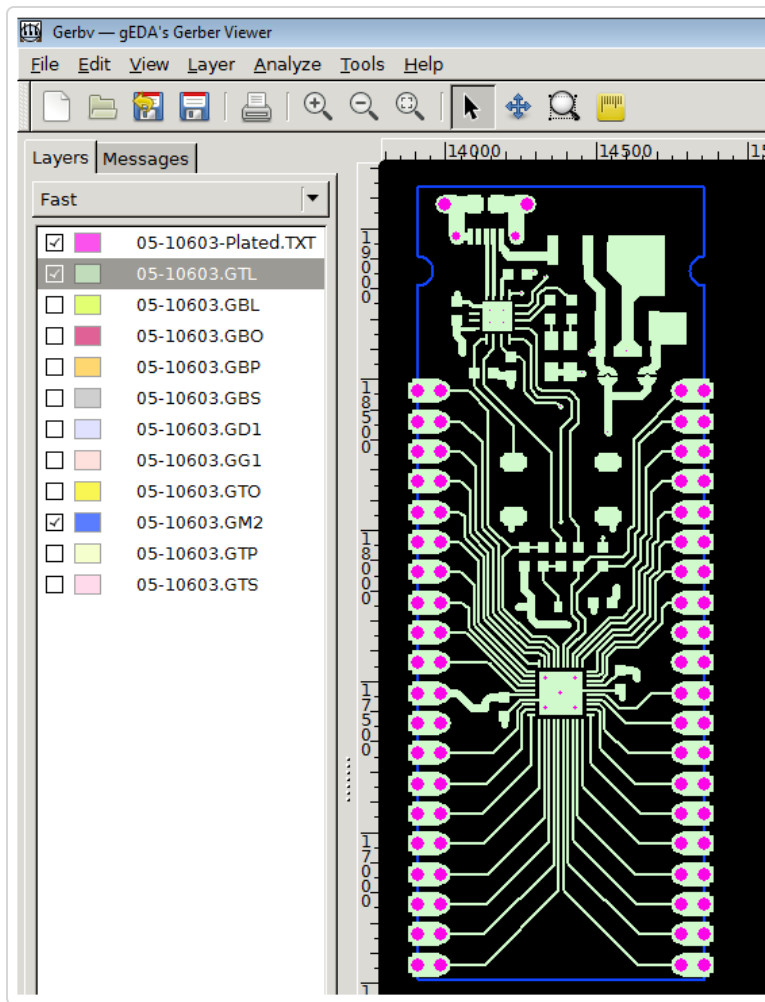
- Reference gerber viewer (https://gerber.ucamco.com/) from ucamco

Figure 17. Inspect layers in Gerbv
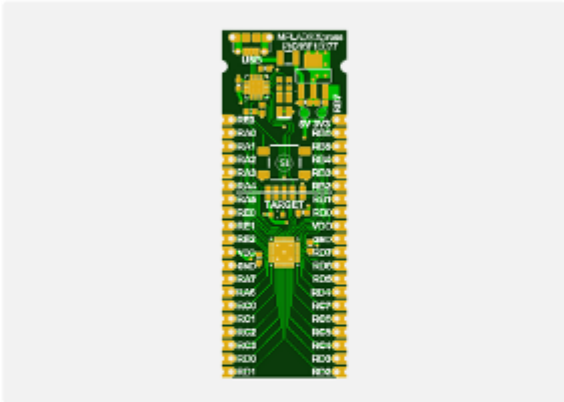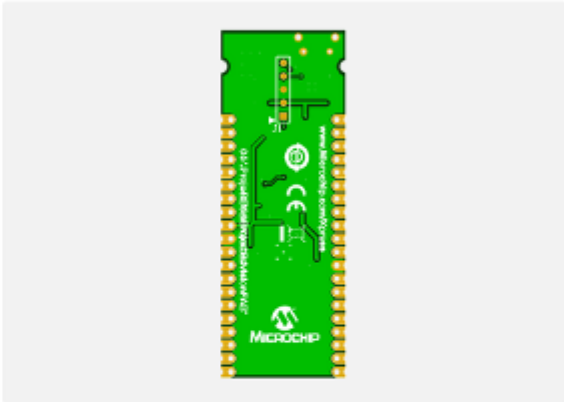
If everything is OK, now you can upload the zip to JLCPCB to place the order.

Figure 18. Preview of the board in JLCPCB.com after uploading