

Suggested Naming Patterns

We do our best to anticipate the default naming schemes from many PCB design packages. However, if you have problems, this is a naming scheme that is known to work.

Filename	Corresponding Layer
boardname.GTL	Top Layer
boardname.GBL	Bottom Layer
boardname.GTS	Top Soldermask
boardname.GBS	Bottom Soldermask
boardname.GTO	Top Silkscreen
boardname.GBO	Bottom Silkscreen
boardname.GKO	Board Outline
boardname.G2L	only if you' re uploading a four layer board
boardname.G3L	only if you' re uploading a four layer board
boardname.XLN	Drills

In some cases, your operating system hides the file extension. This usually occurs on Windows when the design tool generates .gbr or .ger files. You can get Windows to show the extensions by following this guide: [Show or Hide Filename Extensions](http://windows.microsoft.com/en-us/windows/show-hide-file-name-extensions#show-hide-file-name-extensions=windows-7) (<http://windows.microsoft.com/en-us/windows/show-hide-file-name-extensions#show-hide-file-name-extensions=windows-7>)

Our system is case-insensitive. Any mix of capital and lowercase letters works for all filenames and extensions.

Common filename related issues

The most common naming issue we see comes from the board outline layer. Typically, the errors fall into the following categories:

Outline must be larger than 0.25"x0.25" - This often occurs if the zip file contains multiple "board outline" file names. Typically, this is encountered when "GKO", "GM1", and "GM2" files are present. Removing the extra files typically corrects this.

Could not find Board Outline file - Either the board outline is missing, or we didn't detect the filename. If there's an outline file, rename it to GKO to get our system to detect it. If the file is not present, see our Board Outlines page for generating the file in the format we expect.

For single sided designs, we often see the following issues:

- **Could not find Drill File**
- **Could not find — Mask Layer**
- **Could not find — Copper Layer**

As our system's error checking is aimed for 2 layer board files, we require both top and bottom mask layers, as well as a drill hit. For most design tools, the easiest fix is to simply add an exposed via outside of the board outline layer. If the board stackup is configured for 2 layer boards, it will generate all the layers necessary, and will be trimmed before panelization without affecting your design.

For 4 layer boards, ensure that the site indicates "Detected 4 layer board". If our site is not detecting the internal layers, go to our 4 Layer Stackup (<https://jlcpcb.com/quote/pcbOrderFaq/PCB%20Stackup>) to ensure that we can detect the layers with the correct ordering.

KiCAD

Our site understands KiCAD's gerber naming patterns. We suggest checking the **Use Protel Filenames** option, but it's not required.

Note, if the language is set to something other than English, you will need to check **Use Protel Filenames** and manually change the extension of **Edge_Cuts.gbr** to **.GKO**.

For assistance in generating gerbers, check out our Generating Gerbers with Kicad guide ([//support.jlcpcb.com/article/44-how-to-export-kicad-pcb-to-gerber-files](https://support.jlcpcb.com/article/44-how-to-export-kicad-pcb-to-gerber-files))

Eagle

Autodesk EAGLE includes a handy computer-aided manufacturing (CAM) processor that allows you to load a CAM file and quickly generate the specific files you need for your design. Check out this guide ([//support.jlcpcb.com/article/43-how-to-export-eagle-pcb-to-gerber-files](https://support.jlcpcb.com/article/43-how-to-export-eagle-pcb-to-gerber-files)) on how to export Eagle PCB to gerber files.

The following issues may occur when submitting gerbers:

- **Could not find drill file** - For 4 layer boards, this generally means that blind/buried vias was enabled. As we don't support blind/buried vias, you will need to do a DRC check to correct the stackup (<https://jlcpcb.com/client/index.html#/impedance>).
- **Board Outline Not Found** - Either the board outline was not on the Dimension layer, or the outline was placed on the wrong gerber layer.

Altium

Altium typically produces a naming pattern understood by our site. The issues we typically see are the following:

Altium often uses the .TXT extension for drill files, which our site will understand. If multiple .TXT files are included, then we may generate a "Drill files have been merged" warning, which can generally be ignored.

For 4 layer boards, our system will detect .G1 and .G2 files, which are generated for internal signal layers. Our site will not look for GP1 or GP2 files however, as those are typically generated as "negative" polarity gerbers which our system may not process correctly.

Proteus Ares

Ares typically produces all gerbers with the extension .TXT. Our site does not parse this effectively, so the files must be manually renamed to the pattern suggested above.

In some cases, the gerbers do not have a board outline. See our Board Outlines page for how to correct this.

Often, Ares will generate incorrect drill formats, which may result in an "Error" message. Modifying the drill setup to match our Drill File Format will usually resolve this.

A customer has provided a renaming utility to assist with this package. This utility is available at <http://www.hardware.com/gerber-file-zip-utility/> (<http://www.hardware.com/gerber-file-zip-utility/>)

ORCAD/Cadence Allegro

ORCAD usually produces all gerbers with a .PHO or .ART extension. Our site does not parse this effectively, so the files must be manually renamed to the pattern suggested above.

Some configurations produce various other extension patterns. Some of these patterns work except for the Board Outline layer which must be renamed. If you're encountering errors renaming all gerbers will usually correct the issue, or help you identify a missing layer.

Often, ORCAD produces Drill Drawing files instead of NC Drill files, or produces incorrect NC Drill format. See this video tutorial (<https://www.youtube.com/watch?v=mXdW8fig-XQ>) for the correct format options to generate a usable drill file.

Older versions named the drill file thruhole.tap, which our site will recognize. However, other than thruhole.tap, our system will not look for the .tap extension, so using .XLN for the drill file is suggested.

✉ *Still need help? Contact Us (/contact)*

Last updated on June 11, 2019

RELATED ARTICLES

- 📄 [How to export Eagle PCB to gerber files \(/article/43-how-to-export-eagle-pcb-to-gerber-files\)](/article/43-how-to-export-eagle-pcb-to-gerber-files)
- 📄 [How to export Altium PCB to gerber files \(/article/42-how-to-export-altium-pcb-to-gerber-files\)](/article/42-how-to-export-altium-pcb-to-gerber-files)
- 📄 [How to export Kicad PCB to gerber files \(/article/44-how-to-export-kicad-pcb-to-gerber-files\)](/article/44-how-to-export-kicad-pcb-to-gerber-files)
- 📄 [How to generate the Gerber files? \(/article/22-how-to-generate-the-gerber-files\)](/article/22-how-to-generate-the-gerber-files)
- 📄 [How to export Diptrace PCB to gerber files \(/article/46-how-to-export-diptrace-pcb-to-gerber-files\)](/article/46-how-to-export-diptrace-pcb-to-gerber-files)
- 📄 [How to export CircuitMaker PCB to gerber files \(/article/48-how-to-export-altium-circuitmaker-pcb-to-gerber-files\)](/article/48-how-to-export-altium-circuitmaker-pcb-to-gerber-files)
- 📄 [How to export Sprint Layout PCB to gerber files \(/article/67-how-to-export-sprint-layout-pcb-to-gerber-files\)](/article/67-how-to-export-sprint-layout-pcb-to-gerber-files)

