

# How to generate the Gerber files?

To place an order on JLCPCB, please provide us with the data files needed for production. These are:

**Gerber files in RS-274x format** for the copper layers, soldermask and legend layers, mechanical layer and SMD paste layers. Plus carbon, peel-off and via-fill layers as needed.

**Excellon drill file(s) for drilling.**

**Note:**

1. Gerber files and the Drill file must be stored in the same folder/archive.
2. The board outline must be provided.

For specific guides on how to generate the necessary files in the correct format, here is a list of some of the most popular circuit design programs we have notes for.

Altium Designer ([//support.jlcpb.com/article/42-how-to-export-altium-pcb-to-gerber-files](https://support.jlcpb.com/article/42-how-to-export-altium-pcb-to-gerber-files))

Altium CircuitMaker ([//support.jlcpb.com/article/48-how-to-export-altium-circuitmaker-pcb-to-gerber-files](https://support.jlcpb.com/article/48-how-to-export-altium-circuitmaker-pcb-to-gerber-files))

Eagle ([//support.jlcpb.com/article/43-how-to-export-eagle-pcb-to-gerber-files](https://support.jlcpb.com/article/43-how-to-export-eagle-pcb-to-gerber-files))

KiCAD ([//support.jlcpb.com/article/44-how-to-export-kicad-pcb-to-gerber-files](https://support.jlcpb.com/article/44-how-to-export-kicad-pcb-to-gerber-files))

Diptrace ([//support.jlcpb.com/article/46-how-to-export-diptrace-pcb-to-gerber-files](https://support.jlcpb.com/article/46-how-to-export-diptrace-pcb-to-gerber-files))

Sprint Layout ([//support.jlcpb.com/article/67-how-to-export-sprint-layout-pcb-to-gerber-files](https://support.jlcpb.com/article/67-how-to-export-sprint-layout-pcb-to-gerber-files))

## PCB Gerber File

If your Gerber filenames are good, this will save us a lot of time in checking your design.

There are many different PCB design software packages so there are many variations of Gerber filenames and filename extensions.

Gerber Type

File Type	
.TXT, .DRL .dri , .drl	Drill holes
.GML, .GKO, .GML, outline EDGE_CUTS.GBR	Outline
.GTP F.PASTE.GBR	Solder Paste Top
.GBP B_PASTE.GBR	Solder Paste Bottom
.GTS .stc, .smt, F.MASK.GBR	Solder Mask Top
.GBS .sts, .smb B.MASK.GBR	Solder Mask Bottom
.GTO .sst F.MASK.GBR	Silkscreen Top
.GBO .ssb B.SILKS.GBR	Silkscreen Bottom
.GTL .cmp, .top F.CU.GBR	Top Layer
.GBL .sol, .bot, B.CU.GBR	Bottom Layer
Inner layer in 4 layers PCB:	
.GL2	Layer 2
.GL3	Layer 3





If you don't know how to map your files, don't worry about changing the file names and please contact our support for help.

For more information on how to place a PCB order, click [here](https://support.jlcpb.com/article/21-how-do-i-place-an-order) (//support.jlcpb.com/article/21-how-do-i-place-an-order).

✉ [Still need help? Contact Us \(/contact\)](/contact)

*Last updated on April 27, 2020*

#### RELATED ARTICLES

-  [How do I place an order? \(/article/21-how-do-i-place-an-order\)](/article/21-how-do-i-place-an-order)
-  [How to remove order number from your PCB? \(/article/28-how-to-remove-order-number-from-your-pcb\)](/article/28-how-to-remove-order-number-from-your-pcb)
-  [How to order a stencil \(/article/25-how-to-order-a-stencil\)](/article/25-how-to-order-a-stencil)
-  [How do I order a panel? \(/article/24-how-do-i-order-a-panel\)](/article/24-how-do-i-order-a-panel)

© JLCPCB (<https://jlcpcb.com/>) 2020. Powered by Help Scout ([https://www.helpscout.com/knowledge-base/?utm\\_source=docs&utm\\_medium=footerlink&utm\\_campaign=Docs+Branding](https://www.helpscout.com/knowledge-base/?utm_source=docs&utm_medium=footerlink&utm_campaign=Docs+Branding))