Producing JLCPCB Artifacts

Guide for how to create PCB artifacts for ordering PCBs through JLCPCB

1. Run the DRC (Design Rule Checker)

In Altium, go to `Tools > Design Rule Check..`. In the pop-up window, click `Run Design Rule Check...`.

Verify that there are no *important* issues. For example, you can ignore violations regarding "Silk" (silk screen) and some clearance violations. This comes with practice to know which violations are or aren't important.

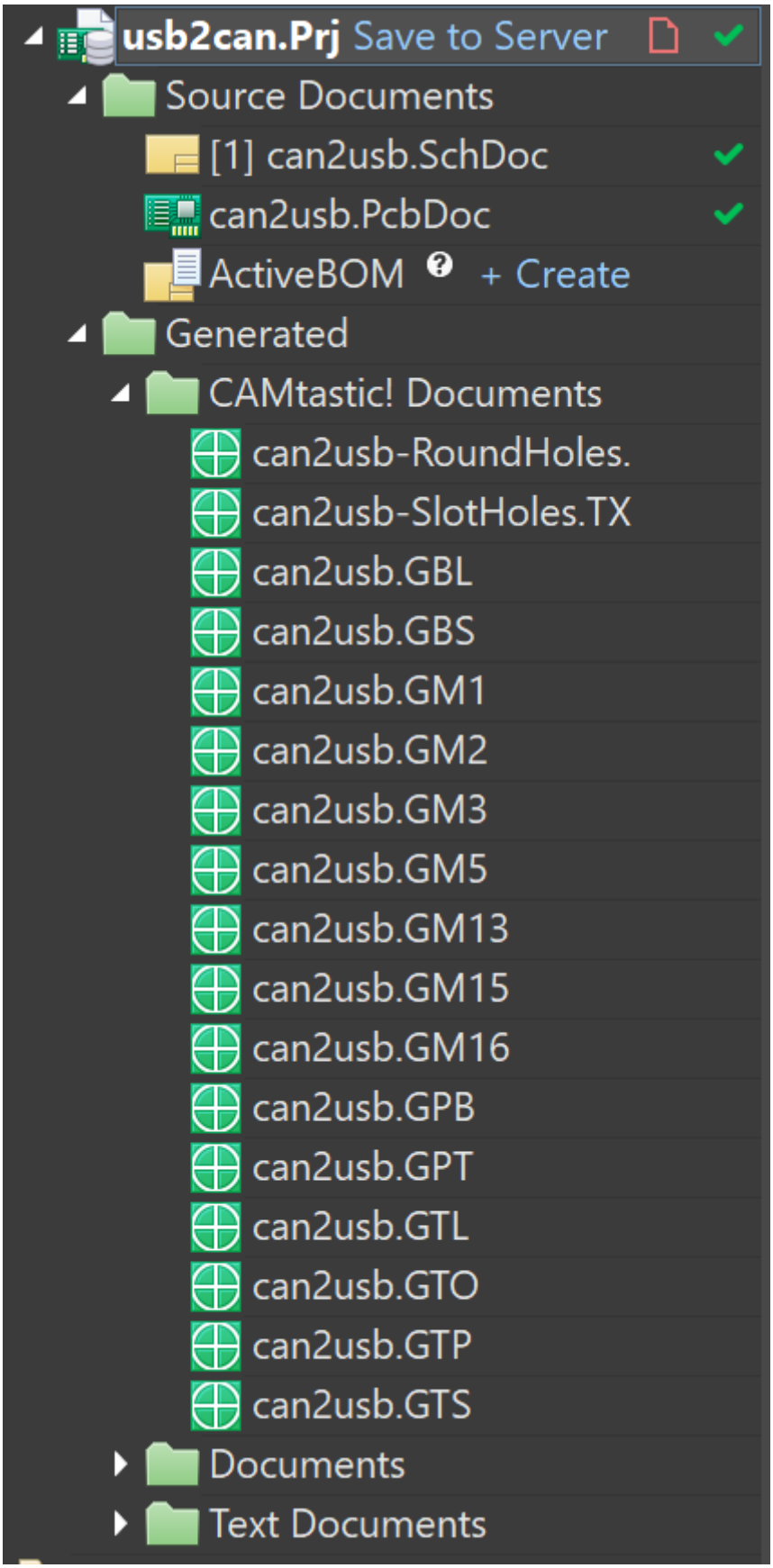
2. Create the artifacts

Follow the instructions for `Generating Gerber file` and `Generating NC Drill File` at https://jlcpcb.com/help/article/8-How-to-export-Altium-PCB-to-gerber-files

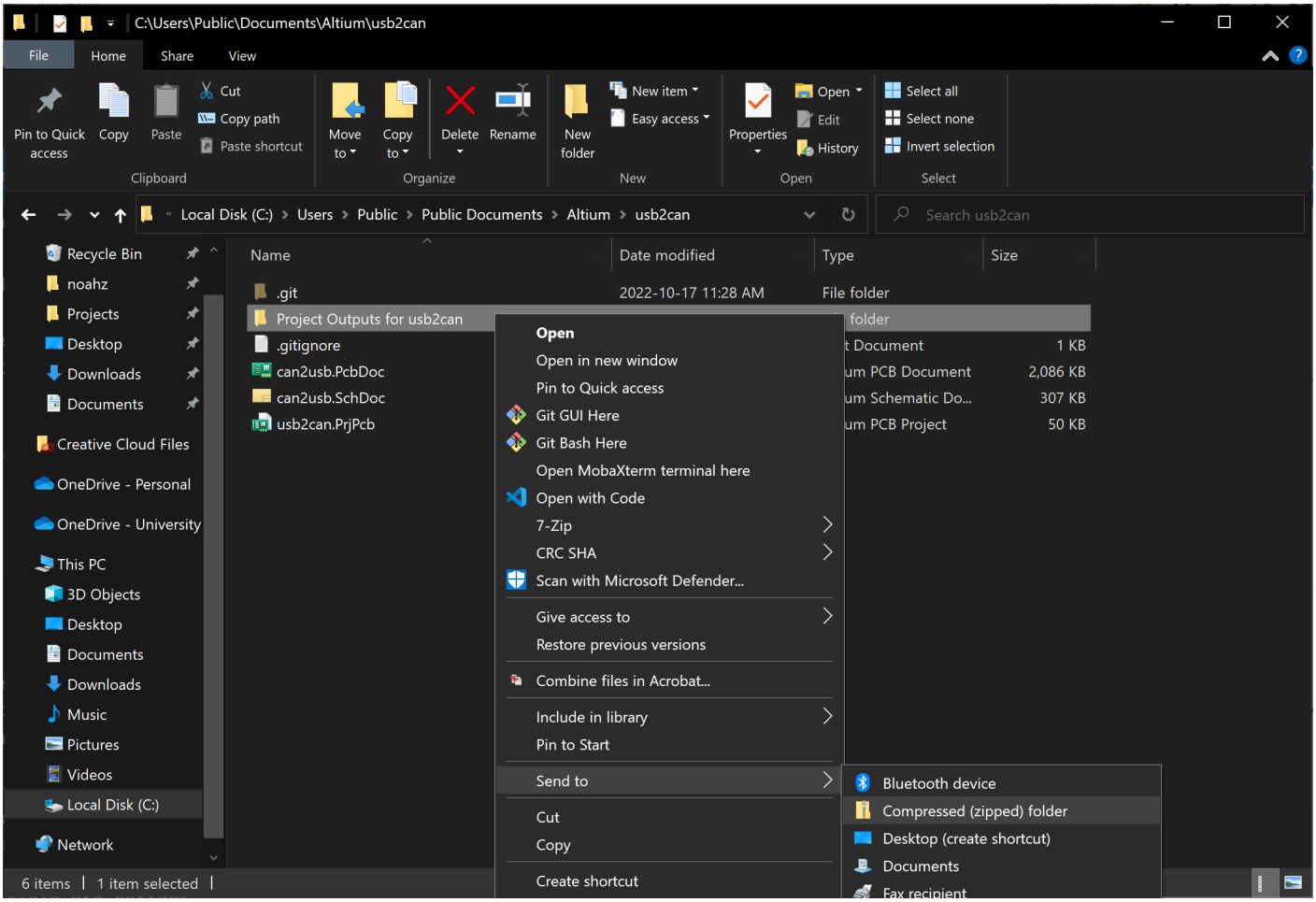
> **Advanced Tip**
> You can customize your DRC to use rules that are specifically tuned for JLCPCB. Download the `RUL` (rules) file available on the above page. See https://www.altium.com/documentation/altium-designer/defining-scoping-managing-pcb-design-rules#exporting-and-importing-rules for instructions on how to import rules. Then, run the DRC again using the new rules which are optimized for JLCPCB.

3. Zip the artifacts

Once you have completed the steps above, you should see several new files under your Altium project, as seen below. Your files may differ depending on your project, but there should be some.



Finally, we need to zip all the files. Right click any of the files and select `Explore`. This should open the folder all your files are contained inside. Navigate to the parent folder. Then, right click the folder with all your artifacts and zip it up, as seen below.



Congrats! You now have a zip file ready to be uploaded to JLCPCB and be ordered.

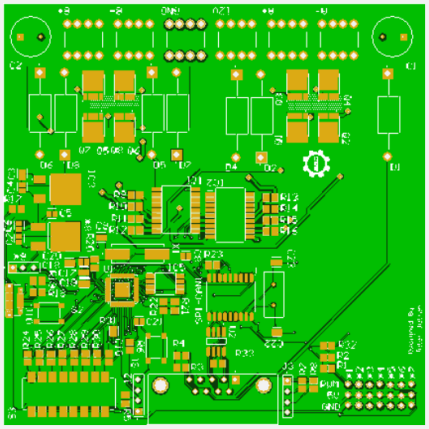
Verifying the artifacts are correct

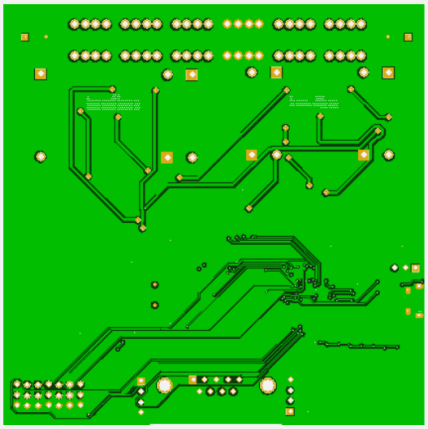
Verify your zip file will work by uploading it to the JLCPCB ordering page at https://cart.jlcpcb.com/quote (This is just to verify, you aren't actually going to order anything!)

Once you upload your zip file, JLCPCB will take a few seconds to process. After it is done, it should look like the screenshot below. Make sure it says `Detected X layer board of YxZ` at the top and that the values are correct. Also make sure the autopopulated values in the `Dimensions` field are correct.

Detected 2 layer board of 101.6x101.6mm(4x4 inches) .

Your upload has finished processing. Enter the project details below and we'll move on to checking all the individual layers to make sure that they're correct.





Back to Upload File

Gerber Viewer

Base Material	FR-4	Aluminum		
Layers	1	2	4	6
Dimensions	101.6	*	101.6	mm

Finally, click on the `Gerber Viewer` link below the PCB preview images. Here you can view exactly how the PCBs will be manufactured. Make sure the top and bottom layers look correct, your silk screen is correct, and the drill holes are actually holes.

Revision #3
Created 2026-10-08 19:13:26 UTC by Dylan Zemlin
Updated 2026-10-08 19:15:38 UTC by Dylan Zemlin