

# Gerber files

Ever asked yourself what comes after the design process? When we finish designing our schematics and PCB layout we start performing some design rule check for our PCB, if everything is fine and green we'd move to the next stage which is the fabrication process.

The fabrication process starts by selecting the PCB manufacturer and then you will find on the manufacturer website a place to upload your design files. The design files are a special type of files named "Gerber files" which is a file format. We export our design in this format with some considerations from the manufacturer. After you send your Gerber files to the manufacturer, the Gerber files go to the computers responsible for operating the drilling machines and start drilling the holes and making the copper traces. Sounds very cool process but we're having an important question here which is "Why do I have to do all of this and not just send the design from the CAD program?". The answer to this question is that Gerber files are not only useful because drilling machines can understand them more easily but there's another crucial importance which is the avoidance of stealing the design. If you sent the CAD files directly to anyone what's stopping them from copying it and fabricating the design under their own name? In this case the Gerber files play their main role which is you cannot easily steal a design from Gerber files unless you will start doing some reverse engineering thing.