

# Manufacturing PCBs with Kitronik

The process of commercially manufacturing PCBs is a little different to the way you may have made them in school. For a start the level of accuracy that can be achieved is much better than etching boards in school. The bottom of a commercial board is covered in a solder resist layer that stops you from soldering to the tracks, only the pads are left exposed. The top of the board is screen printed with component identification, allowing students to see which component goes where. Boards manufactured by Kitronik will be single sided (just have copper on one side).

## Gerber data

To manufacture PCBs a data file is needed for each of the layers that make up the board. This data file describes what shapes have been used to make the layer and also where all of these different shapes are located. When the data is output from the PCB layout software you will get the following layers:

- A board outline
- The copper tracks (on the bottom of the board)
- The pads (on the bottom of the board – used to make the solder resist stencil)
- The identification (on the top of the board)

There will also be two files with drill information. One of these describes what size drills are used, whilst the other gives the co-ordinate information of where the holes are located.

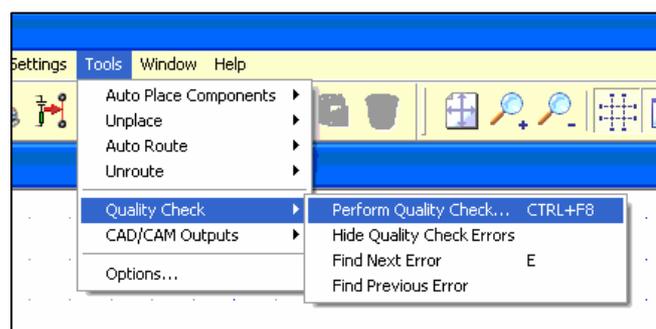
There is normally a report file that describes which file contains which layer. So the data you get for your project should have seven files. The four layers described above, two drill files and one report.

## Real PCB (Crocodile Clips)

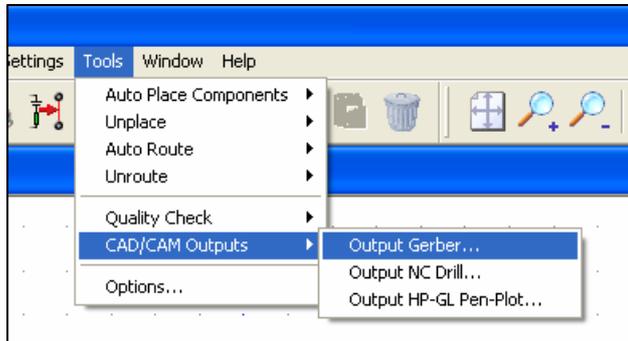
The real PCB software defaults to a single sided board, so as long as you don't change this it will only let you have a bottom copper and a top silk screen so you shouldn't manage to get the wrong layer. Don't forget when you are laying your board out that everything is viewed from the top including the tracks on the bottom of the board.

Before you generate the manufacturing data check the board:

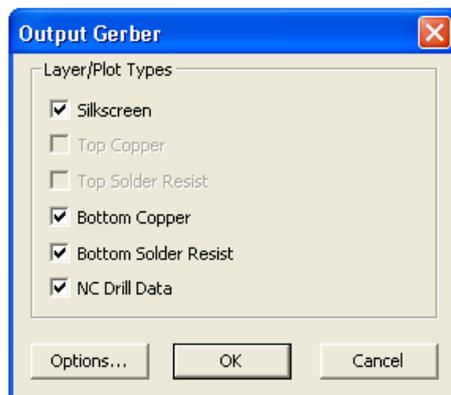
Tools > Quality Check > Perform Quality Check



Once the report from the check is good, you can generate the gerber and drill data. To generate the gerber data use the following menu:  
Tools > CAD/CAM Outputs > Output Gerber...



The following menu will be displayed:



You don't need to change any of the tick boxes or any of the options – just press OK.

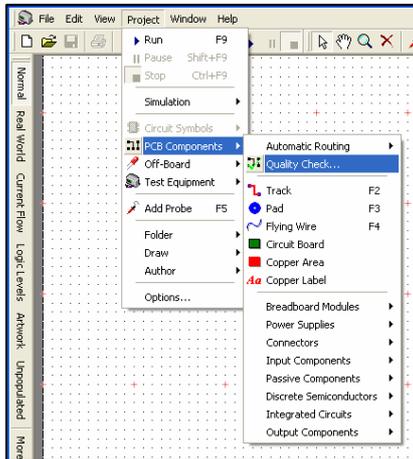
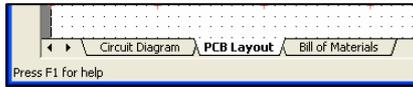
This gives you the following output files:  
MyPcbDesign - Bottom Copper (Resist).gbr  
MyPcbDesign - Bottom Copper.gbr  
MyPcbDesign - Drill Data.drl  
MyPcbDesign - Silkscreen.gbr  
MyPcbDesign.rep

The .rep file is the report and the drill sizes table combined in one file.

The board outline is not exported in its own file, however is included in all of the other three files. There aren't any options to change this it's something that Kitronik need to manually remove from the files and use to generate the outline ourselves. Leave that bit to us, as long as we have the five files listed above then we can do the rest.

## Circuit Wizard (New wave concepts)

At the bottom left of the circuit wizard application are tabs to select what you are viewing. Make sure you have “PCB layout” selected.



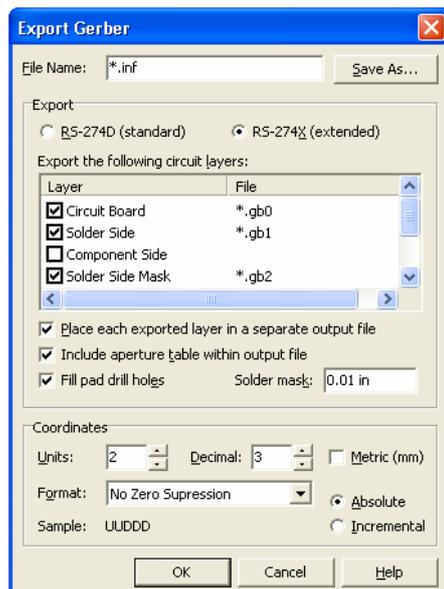
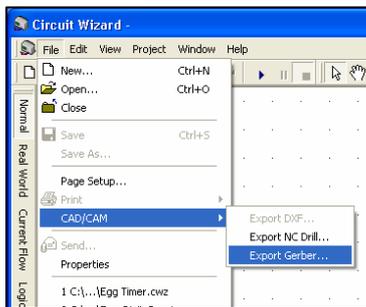
Before generating the gerber data it is worth checking the board is correct and can be manufactured. To do this you need to select:

Project > PCB Components > Quality Check...

Make sure the report that is generated indicates “No issues found”.

Additionally visibly check that all the tracks are red, indicating they are on the solder side.

From the PCB layout screen select the export gerber menu:  
File > CAD/CAM > Export Gerber...



The menu (left) will be displayed:

In the list of files to export you need to tick the boxes as follows:

- Circuit Board
- Solder Side
- Component Side
- Solder Side Mask
- Component Side Mask
- Silk Screen
- Drill holes

You don't need to change anything else, just enter a file name and press OK.