

PCB and assembly at Eurocircuits

January 2023

1 Introduction

This is a tutorial to order printed circuit boards from Eurocircuits. When ordering you first have to decide if you want Eurocircuits to do the assembly or if you want to assemble the PCBs yourself. If you want to assemble the PCBs yourself, the beginning of this tutorial is enough and the whole process takes only a few minutes. If you also order the assembly, the order is more demanding (about 2 h), but you save the work of assembly.

2 Uploading the gerber data to Eurocircuits

The gerber data can be generated by using the do.....gerber.sh scribt from Kai-Martin. This should be located in the folder gerber in your project. If the script isn't working or not there you should generate the folder yourself and download the script used here. Before using, you have to open the script with a text editor and enter the name of your PCB file in line 7, the number of layers in line 9 and (your internal) version in line 10

Then you can execute the script simply by double-clicking. It should create a number of files including a "... .zip" file.

Now you should open "<https://www.eurocircuits.com/>" and log in using "knaak@iqo.uni-hannover.de" as the e-mail and "ElektronIQ" as the password. Under calculate/order you should upload the .zip file.

Next you will be asked if you want to upload BOM data. Since you can also upload them manually later, press "skip assembly data" here. Then choose "Full PCB Analysis". At the end you are asked to configure parameters. Here you should change the project name to the desired project name and enter the desired number of boards. But don't worry, this can be changed later.

3 Managing data in Eurocircuits

Now you should have a project in your shopping basket with the name you have chosen. Here you can click on PCB Visualizer and check if your PCB was

uploaded properly. You can also change the number of boards here. If you want to order the boards unpopulated you are done at this point and you can place your order.

Otherwise you should generate your BOM and CPL data:

3.1 Getting the BOM

The BOM is generated in a similar way as the gerber data. Make a folder called bom in your project and copy the "print-bom.sh" to it. Open it with a text editor and change the name to the name of your project in line 4. Execute the File. This should create the BOM as a file called "... .bom". Sadly the BOM still needs some editing. You should open "... .bom" with "libreoffice calc" changing the designation of every column to "text" for a proper import. This should keep libreoffice from miss-interpreting things (e.g. 0805 → 805).

Then you should expand abbreviated units in the Document like:

10n → 10nF

1k → 1kOhm

50R → 50Ohm

4u7 → 4.7uF

To do this the command replace all is useful.

Congratulations you generate yourself a BOM.

3.2 Getting the CPL

To generate CPL open your PCB in geda.

Use "file" → "export" → "bom" and name the xy file: "..._xy.txt".

Now you need to convert this file to ".csv" and delete the second column where things like "0805" and "SO8" are written. A convenient way of doing this is opening the file with "libreoffice calc" (choosing text as the import format for all columns, deleting the second row and saving it as a ".csv"-file.

3.3 Uploading BOM and CPL

Now go to "Analyse BOM and CPL" and Upload your BOM and CPL (the _xy.csv file is the CPL). You have to upload the BOM first and you have to check that the website is interpreting your columns correctly. Once you have uploaded both files you have to identify all your parts from a list of available parts from eurocircuits. If you want a part to not be placed you can delete it by clicking on the trashcan on the right side. Once you have identified or deleted all objects on the list you should check if everything looks fine in CPL editor (small button on the top left).

Pay special attention to the orientation of the parts, since here most mistakes happen.

Once you are done you can request an offer from eurocircuits. Typically they

will have some problems with your assembly where you have to decide how they shall proceed. Once you have settled all their requests you are ready to order.