PCB and assembly services of Eurocircuits.com

Eurocircuits.com offers services to manufacture the printed circuit boards (PCB) and populate them with (most) of the required parts, including all SMD parts.

To order a populated PCB of a ElektronIQ project (e.g. PIDmareike), proceed as follows:

First, notice that this will take some time!

As preparation, you need the following data files:

1. Gerber files.
   Use Kai-Martin's script "do_..._gerbers.sh"
   You end up with a .zip file, e.g. "gerberdaten_pidmareike_v2_.zip"

2. A bom file.
   Use Kai-Martin's script "print-bom_....sh"
   You end up with two .bom files, one listing each component separately ("….bom") and one grouping identical components ("…_koll.bom").

3. A xyfile, containing position and rotation of all components.
   Use this command: pcb –x bom –xyfile newfilename.xy –xy-unit mm yourPCBfile.pcb
   With "newfilename.xy" any name you want to give the generated xyfile,
   And "yourPCBfile.pcb" the name of the .pcb file of your project.

It is ok if these file also contain components that shall not be populated by eurocircuits. You can choose to ignore them later.

Notice, that eurocircuits.com will try to automatically identify the components by the info given in the bom file. **However, for most components you will have to do this by hand! This will take time!**

Reading in the data on eurocircuits.com:
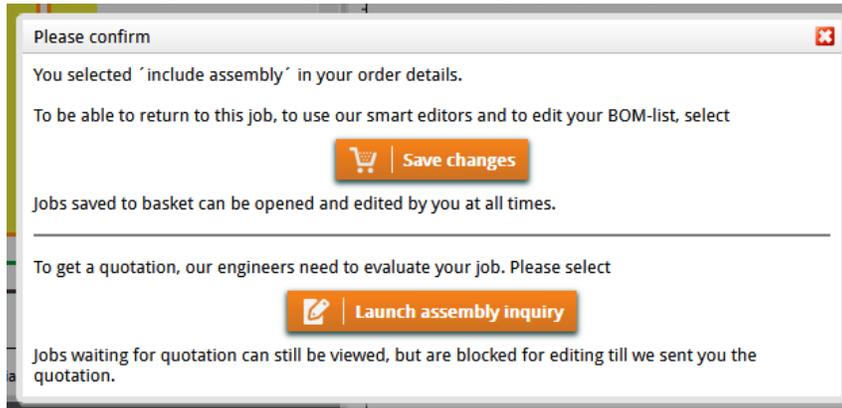
4. Visit eurocircuits.com and sign in. Kai-Martin provides an account that everyone to use:
   username: knaak@iqo.uni-hannover.de
   password: ElektronIQ

5. Go to "Calculate and Order" -> "PCB Proto" -> "Analyze your data"

6. As PCB data choose your gerber files (the .zip file, see 1.).

7. Click on "PCB Visualizer" and check if all layers are read in correctly.

Choose how much PCBs you want to order (You can also do this later).
Click "include assembly"!
For too large PCBs, a warning message pops up: "The PCB dimensions are too large for eC-registration compatible panels. The maximum dimensions are 236x336 mm."

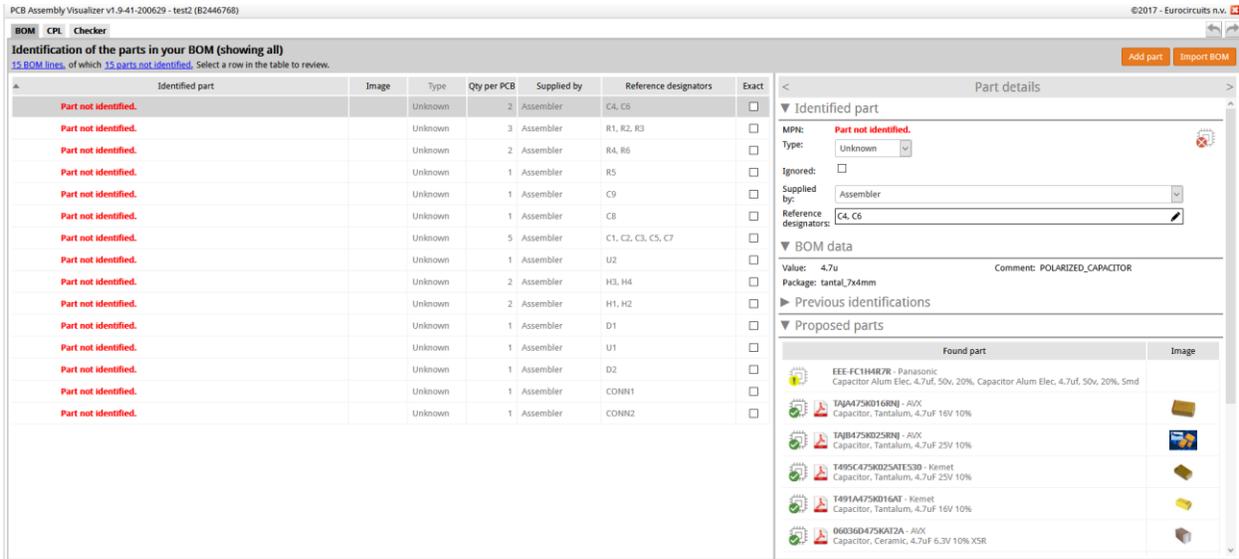When clicking "Save changes", this message popped up:

Just Save changes for now, and press "confirm" without choosing a bom file or a cpl file right now (I guess you can if you want, but then the following steps will be different).

8. Click on "Analyse BOM & CPL". Then click on "Import BOM". Choose your "....bom" file (see 2.; the _koll.bom should also work). Make sure the data is read in correctly.
   For the column names choose:
   a. refdes -> Reference Designator
   b. device -> Comment (You may try something else here as well, e.g. "Description")
   c. value -> Value
   d. footprint -> Package name
   e. quantity -> Quantity

You will end up with something like this:

Before choosing all the parts, read in the xyfile.

9. Click on CPL (top left of the window). Click "Import CPL". Choose your xyfile (see 3.). Make sure the data is read in correctly.
    For the columns choose:
    a. Column1 -> Reference Designator
    b. columnn1 -> "None"
    c. columnn2 -> "None"
    d. columnn3 -> "Center X"
    e. columnn4 -> "Center Y"
    f. columnn5 -> "Rotation"
    g. columnn6 -> "Side"
10. You now see the PCB and the outlines of the already chosen or automatically identified parts.

    Go back to "BOM". For each part, choose the correct component from the list of proposed parts, or use the search function.
    If you are unsure about the correct size/footprint, you can click on CPL and check if the outline fits the copper pads.
    You can check the "ignore" box for components that eurocircuits shall not populate.

11. If you feel good about the read in data, you can just order.
    If you don't maybe use that "Launch inquiry" button (at the PCB Visualizer).

Some notes:

*   For chip capacitors, choose X7R parts.
*   It may be necessary to check on the website of one of the suppliers (e.g. Farnell) for the price, to see if you by accident chose some very special part (e.g. a resistor with resistance accuracy up to 0.1%), where also a simpler version would have been ok (any SMD resistor).