EAGLETOOLS is the very first toolbar made in FlamingoTools workbench. In fact it is so simple that all its features can stay in one dialog.

At that time I was having fun with electronics in the Arduino's way, so I was looking for one tool that could help me understand how to arrange the electronic board on the 3D assembly of my project. That was made, among other things, with aluminum short beams and an LCD and I wanted how it would look like when completed.

FreeCAD is perfect for this kind of first "feasibility studies" and it already got the add-on "FreeCAD PCB" (**ThankYou**, *marmni!*) that can translate one 2D PCB drawn with the freeware software *EAGLE-Light* in one 3D model made with FreeCAD. See the good documentation of that FreeCAD-PCB workbench for detailed information.

The only boring step of this add-on, if you don't use it so often, is to "synchronize" the EAGLE's library with the database of 3D parts.

That's because FreeCAD-PCB collect all the components of the board in a group called Parts and create the board from the data of the EAGLE's file IF it the various components are already stored in its database and are linked to libraries of the other program.

This is the reason why I made this shortcut:

- One function imports from the .brd file (practically one .xml with the PCB's data) the names and the positions on the PCB of electronic components.
- Another function moves the components, already created in the 3D model, to the positions imported in the previous step according ONLY TO THEIR NAME if it matches in both lists, i.e. the imported list and the group "Parts". Other components are ignored.

In this way there is no need that components in the 3D models database are hard-linked to those in the EAGLE's library. You can make an approximate model of the PCB just caring that you give the same name to the component of .brd file and that of .FCStd file.

Let's clarify with an example, very straightforward.

Suppose that you designed with EAGLE the arrangement of a PCB shown at the side and you want to show it on the assembly of the complete project made with FreeCAD. You need something that makes a 3D drawing of it or something that resembles to it.

That tool is FreeCAD-PCB.

Suppose also that you don't have time, or are too lazy to link each component of the libraries of both programs. You need something that moves stuff already imported in your 3D model to the corresponding point of the PCB.

That tool is represented by the hammer at the top of this page.
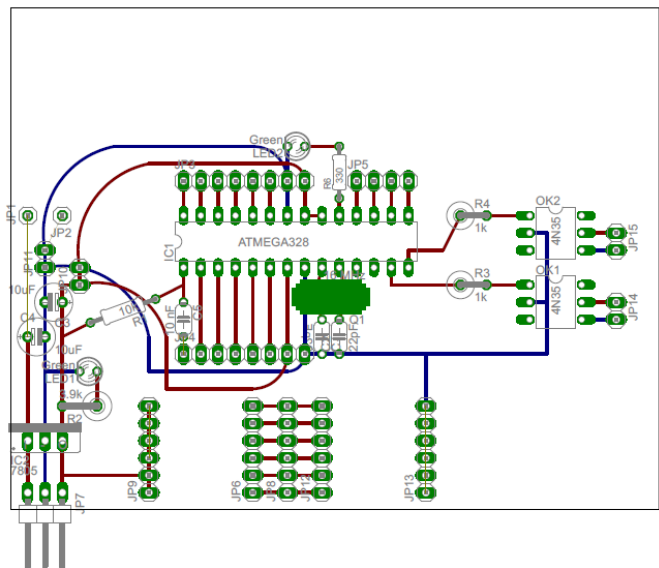
So you start drafting your PCB. Then grab some resistor, capacitor, led etc. from the database of FreeCAD-PCB and throw them in the model. You don't care about their initial position so they will all overlap at the origin point, at the beginning.
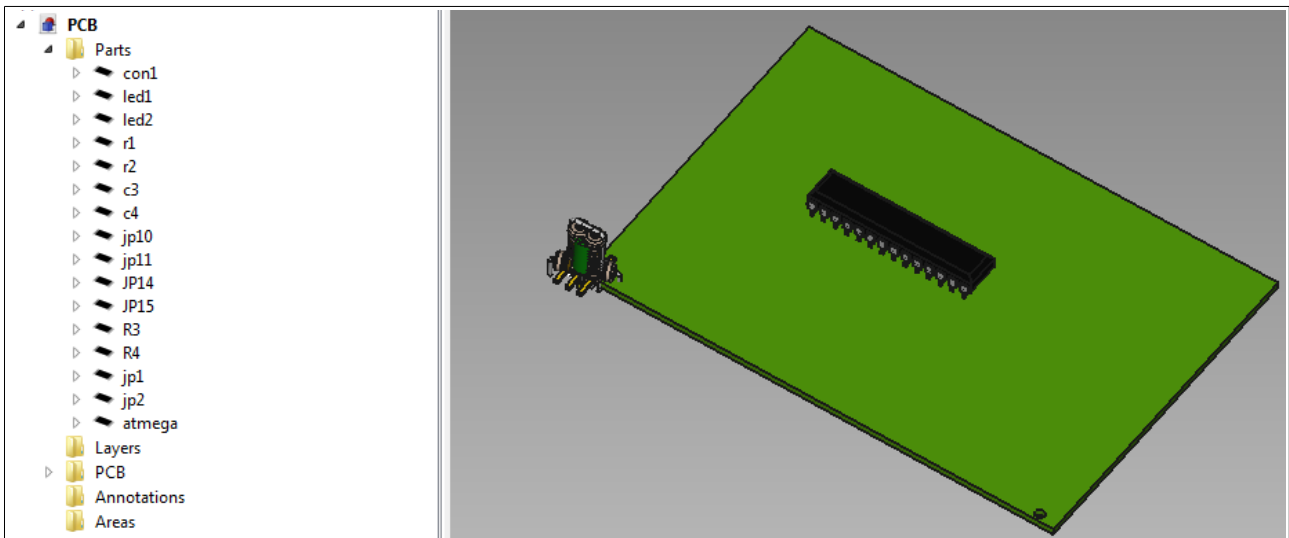
The only thing you have to care is that the 3-pin connector is named "*con1*" in both files, the red led is named "*led1*", the other led is named "*led2*" and so on...
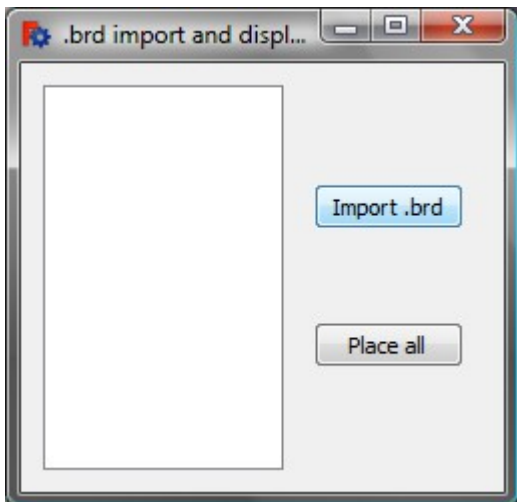
When you've done, you click on the hammer and the dialog to the left side appears.
When you open the dialog, the __init__() function will first check the model and alert you in the console if the group "Parts" is missing, if it's empty or how many components it includes.

- [Import .brd] button will let you browse the filesystem to find where you've saved the drawing of your board.
- The listbox on the left will populate with the names of components of .brd file that matches the names of components in the Parts group
- [Place all   ] button will move each component listed to the corresponding position imported from the .brd file.

Notice also that there could be no relation between objects drawn with EAGLE and those in the FreeCAD model. For example, you may have called "led1" the drawing of a flower in the 3D model but it will be moved to the position of the red led of the PCB as well as in the model of a garden...

Now, to complete this tutorial we click on [Import .brd] and browse to "./examples/board6.brd". The program reports in the console the names found and the relevant coordinates.

Then press on [Place all] and the electronic stuff will be moved according to the 2D drawing just imported. While the procedures scan all the names of the "Parts" group, it checks if they are included in the list imported, it reports in the console and moves the corresponding component.

If you used the PCB example found in the a.m. folder, the result should be as below.
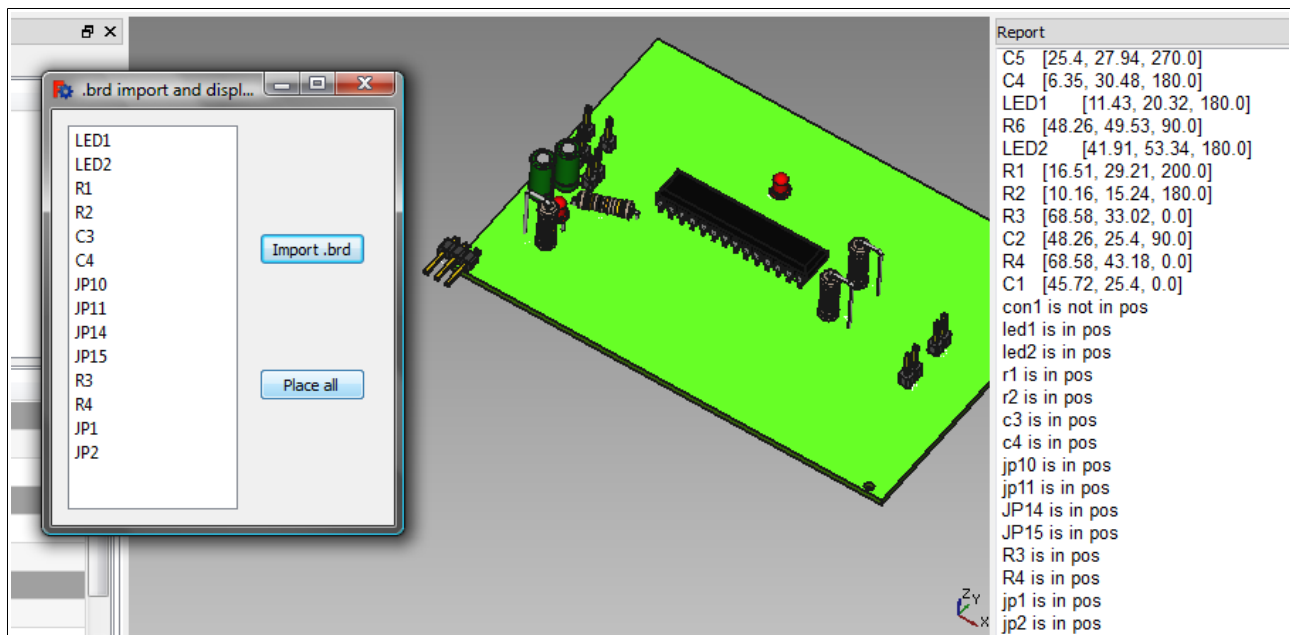
Finally notice that there's no need to create in 3D model ALL the components defined in the .brd file. You can simplify the drawing as you like, if it's sufficient for your purpose.