



Lena Quambusch - Mar 11, 2023 at 11:13 AM GMT

## General Information (short)

Lena Quambusch - Mar 11, 2023 at 11:47 AM GMT

The Prusa Software needs and .STL or .OBJ file to generate a printing code for the printer (.GCode).  
There are several possibilities to find or create such files.

- 1a. Use a model from websites like <https://www.printables.com/en/model> (Prusa), or <https://www.thingiverse.com/>
- 1b. **Create your own model** with a 3D modeling software web-based or with desktop software.
2. Once the model is ready just load into **PrusaSlicer** (Print Software).
3. And generate **the printing code**.
4. Load the code onto the SD-Card at the Mac (card reader at right site).
5. Transfer the card to **the Printer** again. Select your file from the LCD display at the front of Bruce.  
(> Home Button /Menu > Print from SD > *Your file name.gcode*)
6. Wait till print has finished.

Done!

For more detailed descriptions see below.

Lena Quambusch - Mar 11, 2023 at 11:23 AM GMT

## Creating your own 3D Model

Lena Quambusch - Mar 11, 2023 at 7:18 PM GMT

### Software:

There are several 3D modeling software applications one can use (Blender, SelfCAD, LibreCAD, Sketchup...) or a web based version TinkerCAD ([www.tinkercad.com](https://www.tinkercad.com)).

To guide through the process we use TinkerCAD here as an example, but any other program will do the job as well. Just a matter of user friendly/beginner friendly interface and so on.

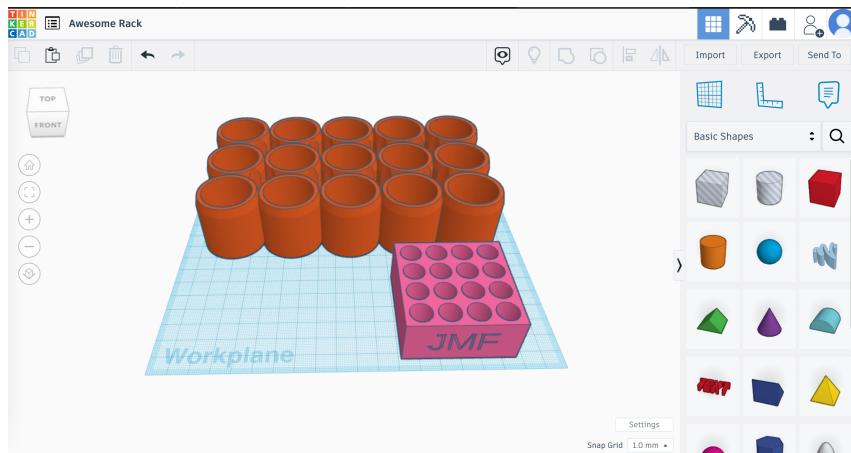
### **3D Model:**

Within those modeling softwares one can create structures/models from scratch or alter/transform models originating from platforms like thingiverse/printables. Any .stl and obj. will do.

As a simple example we will modify some existing objects and create a new one with MH branding to explain what kind of operations and things to consider during the process.

Creating a rack combination for Falcons and Eppendorf tubes base don two different designs with TinkerCAD.

1. Import Model(s) you want to modify into TinkerCAD (need to have an account, free of charge). Go on > Import and just drag and drop the file that you downloaded from the above mentioned websites.

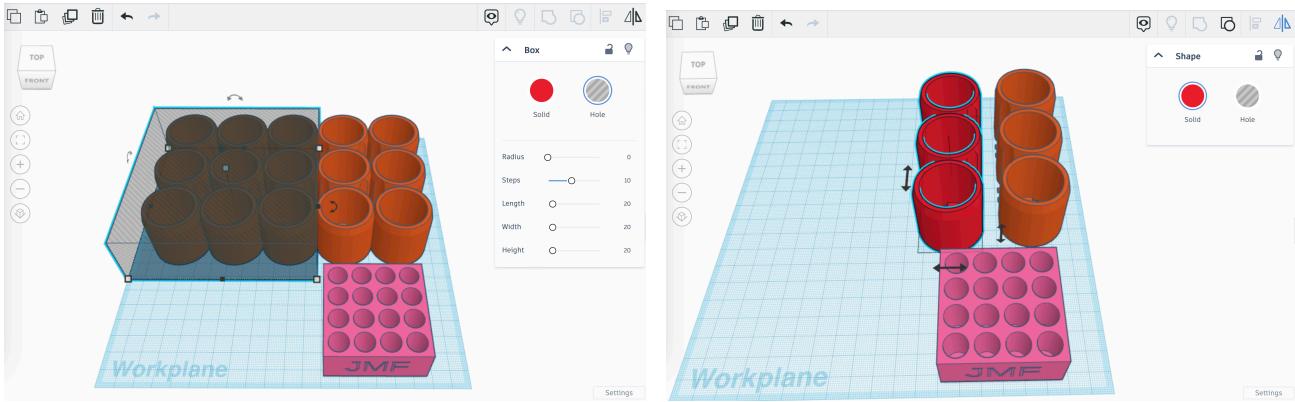


The GUI is really simple. You have the blue workplane, the objects on it and on the right a lot of shapes one could add or use to delete things.

---

2. Now one can easily **modify the objects** in size and height.

But in this case the overall size shouldn't be altered since it is already fitting to the desired tube formats (falcons/eppis). It is also possible to basically cut out thing or reduce the 3D volume using other volumes and creating a hole in the present design.



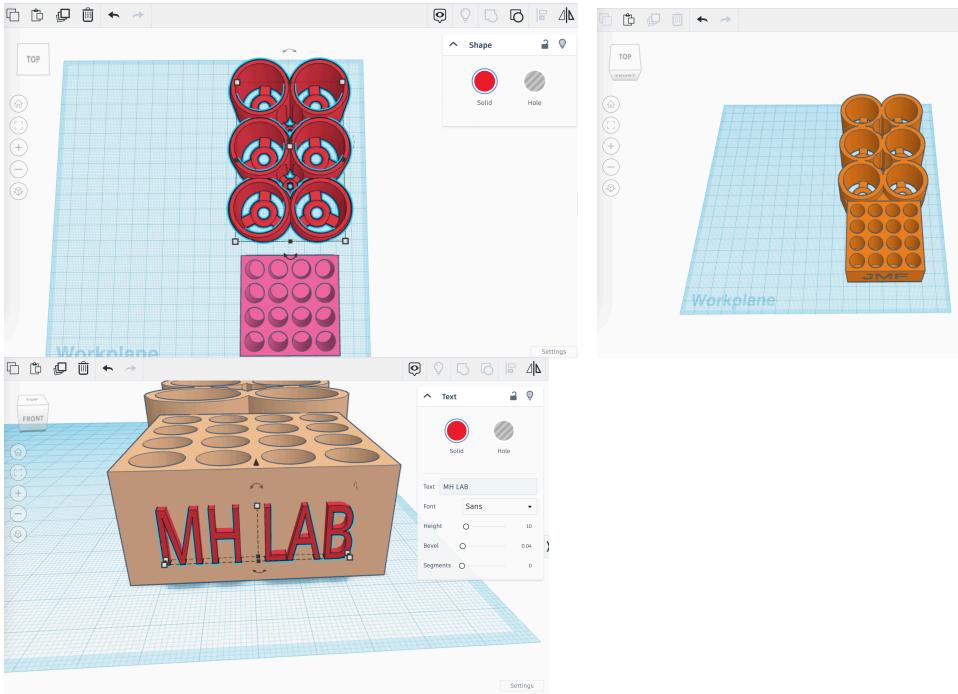
Take a cube from the right and select hole instead of solid filling. Adjust position on the workplane and size to fit the actual volume you want to cut out. Then select both object and pick > group (ctrl + g), icon on the top right corner above "Box/shapes".

It is always possible to ungroup things later.

Deleted 4 rows (not 3 as shown) of the rack and copied the last one remaining to a new object (right picture). Then just mirror the object before grouping those two rows together. This way the outside at the corners of the falcon holder part will be the same.

Added some filling between the two racks and then just grouped them again to result in one object.

There is also the align tool next to the mirror command on the top right corner.



For Fun we will add MH LAB as branding in the front of the eppi tube rack, where we already have a branding from the original file. Model ready!

3. Now we need to **export the model** as .stl or .obj file in order to loaded in to the Prusa Software. Maybe do a final check of the design. Are there any unexpected or unintended holes? Everything aligned correctly. Are you HAPPY with the design?

The printer's resolution can be as low as 0.1 mm, consider if any edges or thin lining are above this size.

Lena Quambusch - Mar 12, 2023 at 10:36 AM GMT



**Awesome\_Rack.stl (728 kB)**

Lena Quambusch - Mar 12, 2023 at 10:37 AM GMT

## PrusaSlicer and the printing code

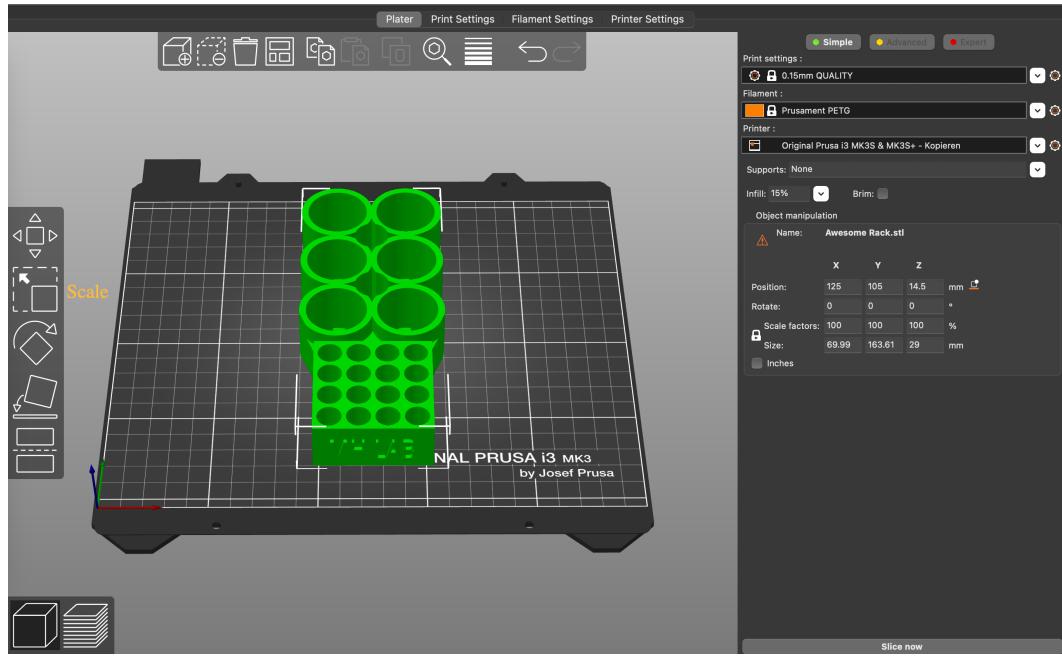
Lena Quambusch - Mar 12, 2023 at 10:38 AM GMT

The **PrusaSlicer** software is available on the Prusa website, but we also have it on the Mac next to Bruce. Transfer your Model via dropbox or USB drive to the computer and use the desktop app there.

## 1. Load your model via drag&drop, cmd + l, or > file > import > import stl.

Once opened the object will be placed in the middle of the plater (print area). It will have the exact measurements as saved with the 3D modeling software. But one can adapt the size by choosing the "scale" icon on the left toolbar or enter number under Object manipulation on the right.

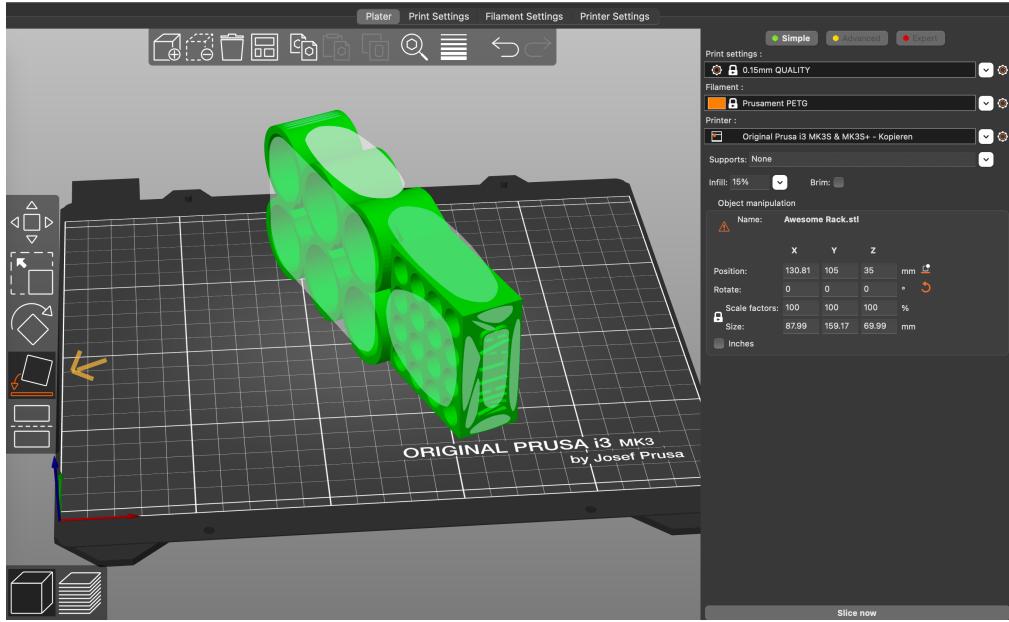
The actual position on the print area can be changed as well.



2. Decide for **the base layer side**. What should be the bottom/top of the 3D model.

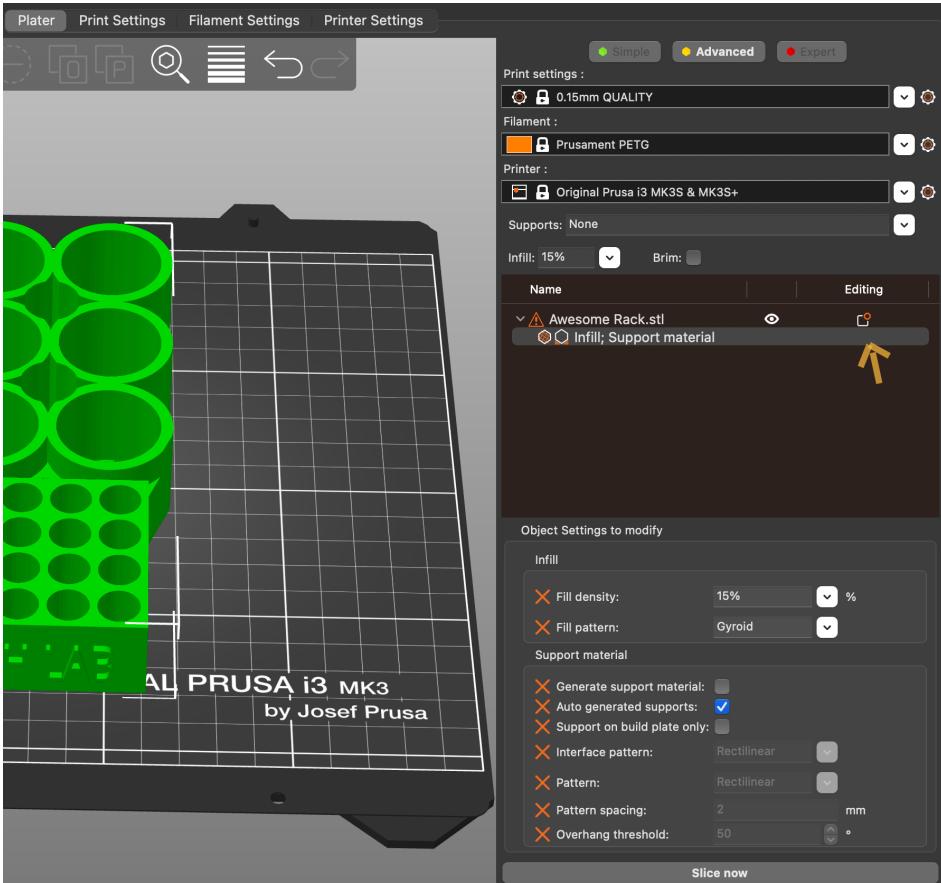
The printer prints basically many 2D layers on top of each other, so one needs to consider in which orientation a 3D model is printed. Ideal would be if the printer has no or little bridging to do, meaning printing a layer without anything below it. The software can add support material for such things, BUT less is always better.

In this case the rack is pretty simple, the bottom should be the bottom. In other cases you can easily define a new base with the "place on face" icon on the left.



The rack should not be printed like shown above. The white ovals on the model just exemplify possible sides that one could select to face down.

**3. Adjust Print Parameters.** Now we need to tell the software what filament/material is used, which quality we want and define further details to generate the printing code.



**Print Settings:** 0.15mm is a default setting for the nozzle we have at the printer. Larger numbers will have lower quality and smaller number higher quality surface, but also will take longer (!). Most bigger things should be fine with 0.2mm.

**Filament:** just pick the one that is loaded from the drop down menu. (PET = PETG)

**Printer:** As shown above.

The next two parameters are defined when in "Advanced" mode (selected on the top). One is Infill, in terms of how much material will be used to fill in the volume and which pattern is used. The other one is supporting material, which IS NEEDED when you want to print bridging.

Click with right click onto the symbol under column editing next to the objects name on the right side bar. Then select the features Infill and Support material. Now the *Object Settings to modify* appear in the lower right corner.

### Infill

**Fill density:** 15 % might be the default option, but anything < 30 % should be fine. Depends on how solid and stable the object needs to be in the end. Large objects are probably equally stable even with lower infill.

**Fill pattern:** choose one from the drop down menu. Depending on the pattern the printer needs less or even more filament to print.

### Supporting Material, check the top box.

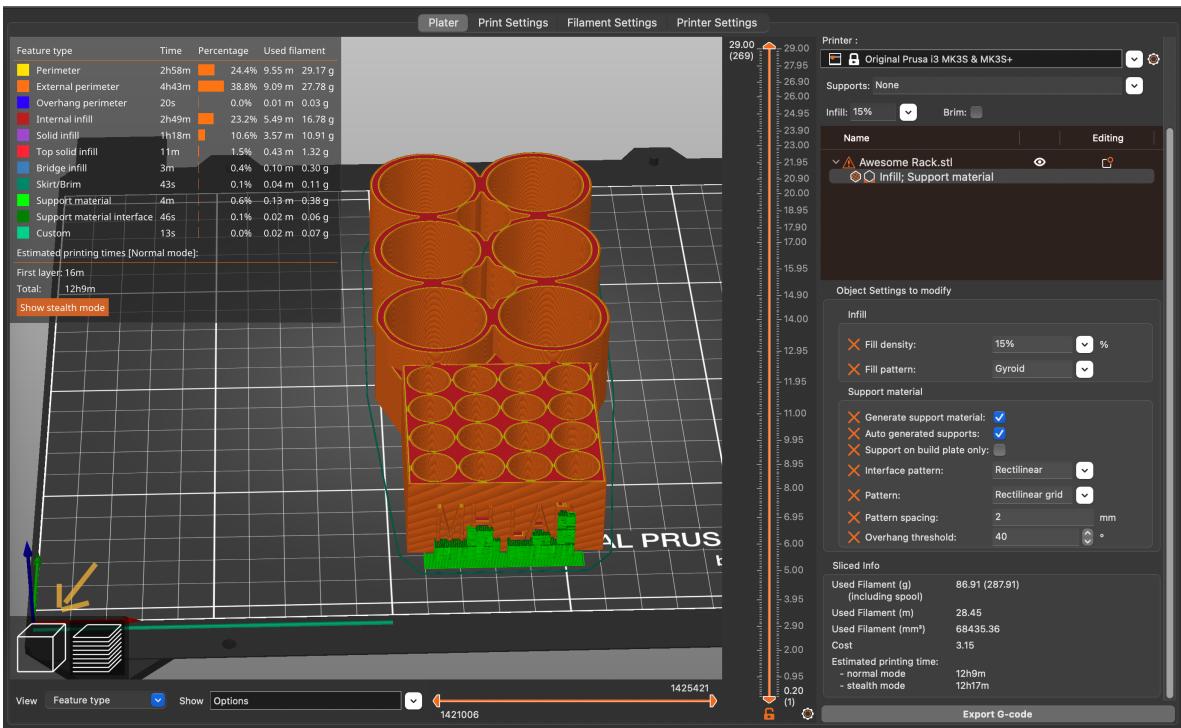
**Pattern/Pattern Spacing:** default setting seem to work fine, but one could optimize here material and time wise.

**Overhang threshold:** depends on what kind of angle should be supported during printing. Safe options would be to support anything that is 40° off vertical axis, so threshold to 40°. Depends on the Model in the end.

Everything defined? Then hit "Slice now", bottom right corner.

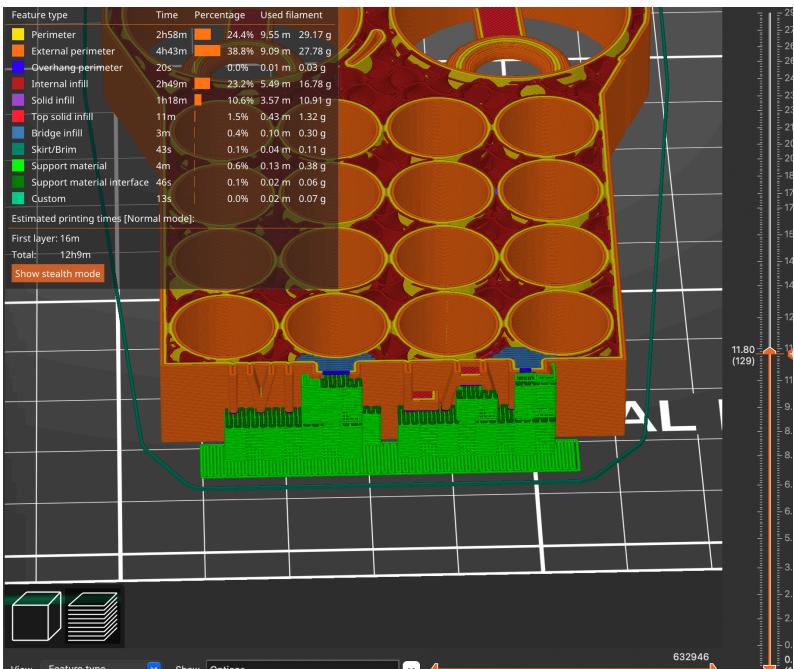
#### 4. The printing code.

The program calculates every layer of the model. In the top left corner is a summary of all the different infill and perimeter types (color coded). It also gives the total printing time and how much filament/material is needed, even an estimated cost (lower right corner).



To return to the original model just hit the cube in the lower left corner. The two icons allow to switch between print layer visualization and your model.

The numeric bar next to the model on the right allows to go through every layer of the print. As the Blue bridging and overhang perimeter are the most challenging ones for the printer it is sometime ideal to check again if those parts have enough support (as seen below). Without the green support material probably the printer would mess up the H in "MH LAB".



If you change any printing parameters you always need to calculate a new G-Code. As mentioned the parameters like infill, support and quality are things that have impact on the overall printing. Changes here can reduce the time significantly. Maybe just play around and have a look at statistics.

If you want to **add another color** to the print select the layer with the scale on the right and then click on "+" button next to the arrow. From the selected layer on the printer will demand for filament exchange during the print and only progresses if confirmed manually at the LCD display.

Once happy with all the setting one needs to "Export G-code "and save it. The automated file name will tell resolution, filament type, printer, and print time.

**Done and ready to load for printing!**

---

Marko Hyvonen - Oct 13, 2023 at 11:06 AM BST

Assembly manual

[https://help.prusa3d.com/category/original-prusa-i3-mk3s-kit-assembly\\_1128](https://help.prusa3d.com/category/original-prusa-i3-mk3s-kit-assembly_1128)

---

Lena Quambusch - Mar 12, 2023 at 10:37 AM GMT



**Awesome\_Rack\_0.15mm\_PETG\_MK3S\_12h9m.gcode (36.1 MB)**