



HOW TO PREPARE A PRINT MODEL

After creating a 3D model, it is necessary to prepare data for a 3D printer. There are several programs which can prepare the model for 3D printing. Some of the programs must be paid, some of them are free of charge. These are for example: Slic3r, Cura, OctoPrint, Repetier (Free); NetFabb, Simplify3D (Pay).

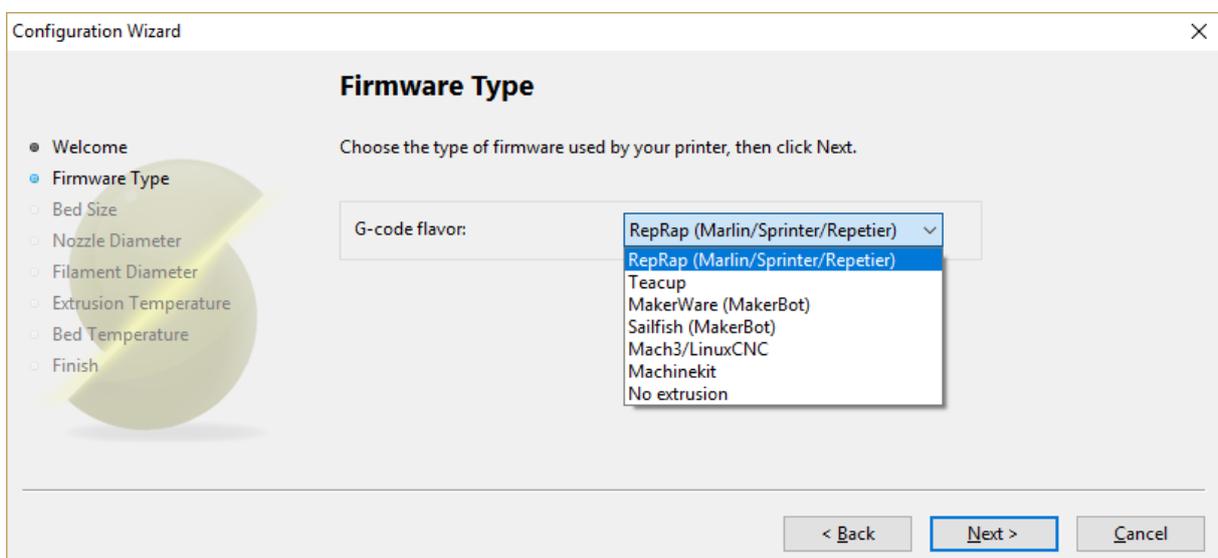
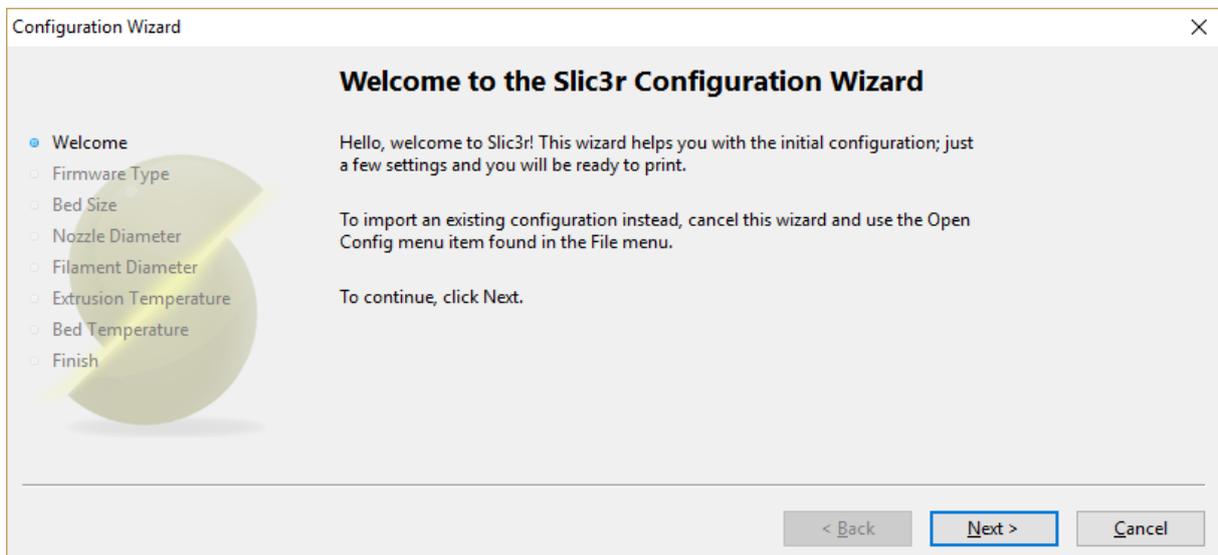
We recommend using the program Slic3r. It provides users with valuable pros - It is free, simple to set up. Unfortunately, it does not contain some functions which are only available in the paid software.

INSTALL THE PROGRAM SLIC3R

Slic3r can be downloaded directly from the address <http://slic3r.org/download>

At the beginning of the installation, the configuration wizard starts.

The settings for our 3D printer /Prusa MK2/ can be seen in the following screens.





Configuration Wizard

Bed Size

Set the shape of your printer's bed, then click Next.

- Welcome
- Firmware Type
- **Bed Size**
- Nozzle Diameter
- Filament Diameter
- Extrusion Temperature
- Bed Temperature
- Finish

Shape:

Settings:

Size:	x: 200	y: 200
Origin:	x: 0	y: 0

< Back **Next >** Cancel

Configuration Wizard

Nozzle Diameter

Enter the diameter of your printer's hot end nozzle, then click Next.

- Welcome
- Firmware Type
- Bed Size
- **Nozzle Diameter**
- Filament Diameter
- Extrusion Temperature
- Bed Temperature
- Finish

Nozzle diameter: mm

< Back **Next >** Cancel

Configuration Wizard

Filament Diameter

Enter the diameter of your filament, then click Next.

Good precision is required, so use a caliper and do multiple measurements along the filament, then compute the average.

- Welcome
- Firmware Type
- Bed Size
- Nozzle Diameter
- **Filament Diameter**
- Extrusion Temperature
- Bed Temperature
- Finish

Diameter: mm

< Back **Next >** Cancel



Configuration Wizard

Extrusion Temperature

- Welcome
- Firmware Type
- Bed Size
- Nozzle Diameter
- Filament Diameter
- Extrusion Temperature**
- Bed Temperature
- Finish

Enter the temperature needed for extruding your filament, then click Next.

A rule of thumb is 160 to 230 °C for PLA, and 215 to 250 °C for ABS.

Temperature:

< Back **Next >** Cancel

Configuration Wizard

Bed Temperature

- Welcome
- Firmware Type
- Bed Size
- Nozzle Diameter
- Filament Diameter
- Extrusion Temperature
- Bed Temperature**
- Finish

Enter the bed temperature needed for getting your filament to stick to your heated bed, then click Next.

A rule of thumb is 60 °C for PLA and 110 °C for ABS. Leave zero if you have no heated bed.

Bed temperature:

< Back **Next >** Cancel

Configuration Wizard

Congratulations!

- Welcome
- Firmware Type
- Bed Size
- Nozzle Diameter
- Filament Diameter
- Extrusion Temperature
- Bed Temperature
- Finish**

You have successfully completed the Slic3r Configuration Wizard. Slic3r is now configured for your printer and filament.

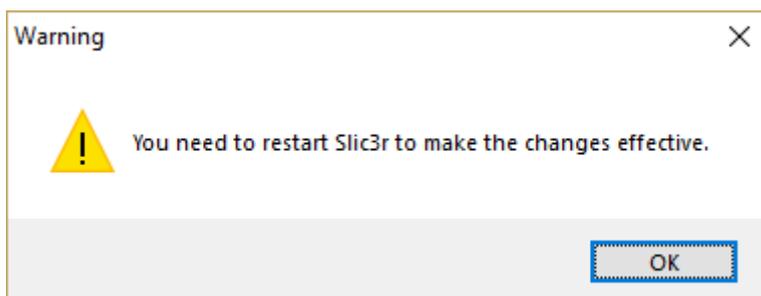
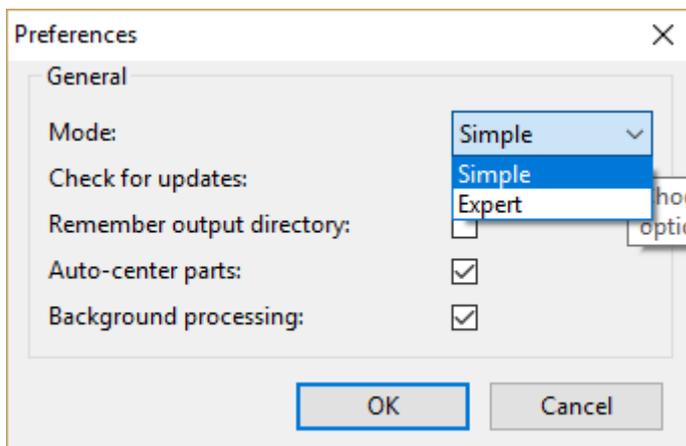
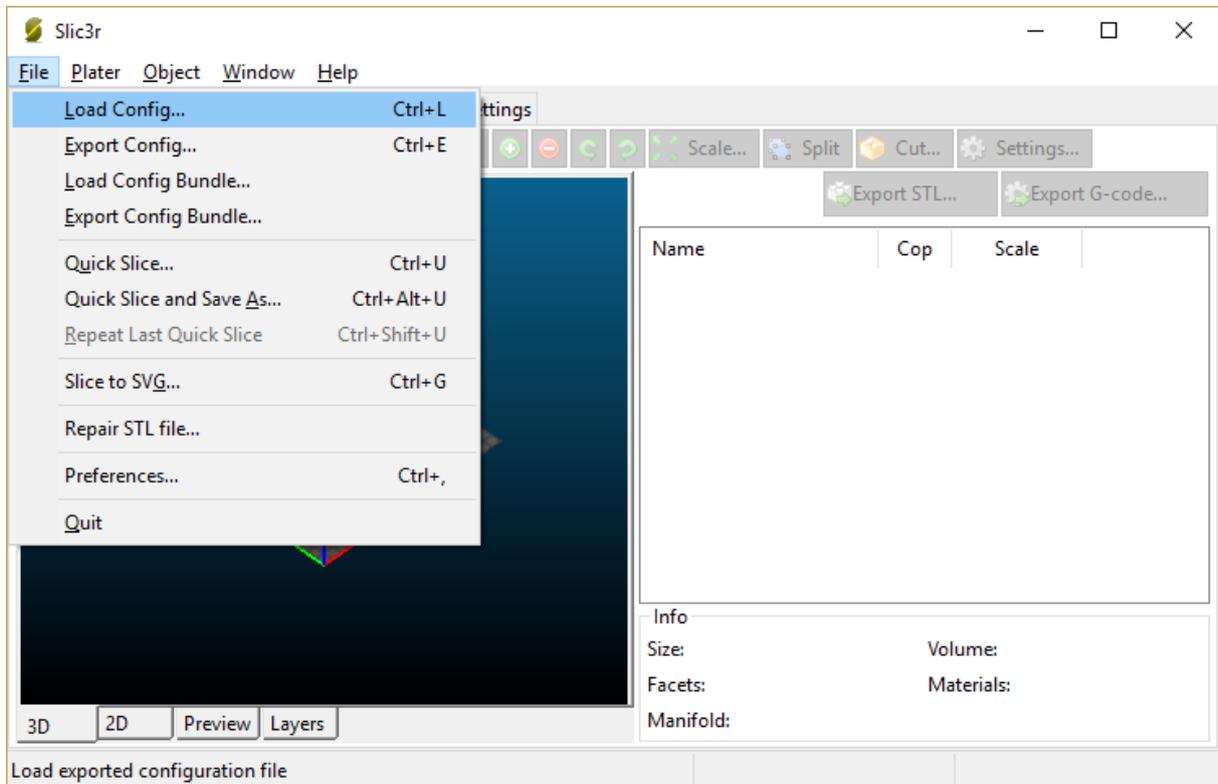
To close this wizard and apply the newly created configuration, click Finish.

< Back **Finish** Cancel



After installing, we will have to switch the program from simple to the expert mode. The restart of the program is required to apply the change.

File/Preferences





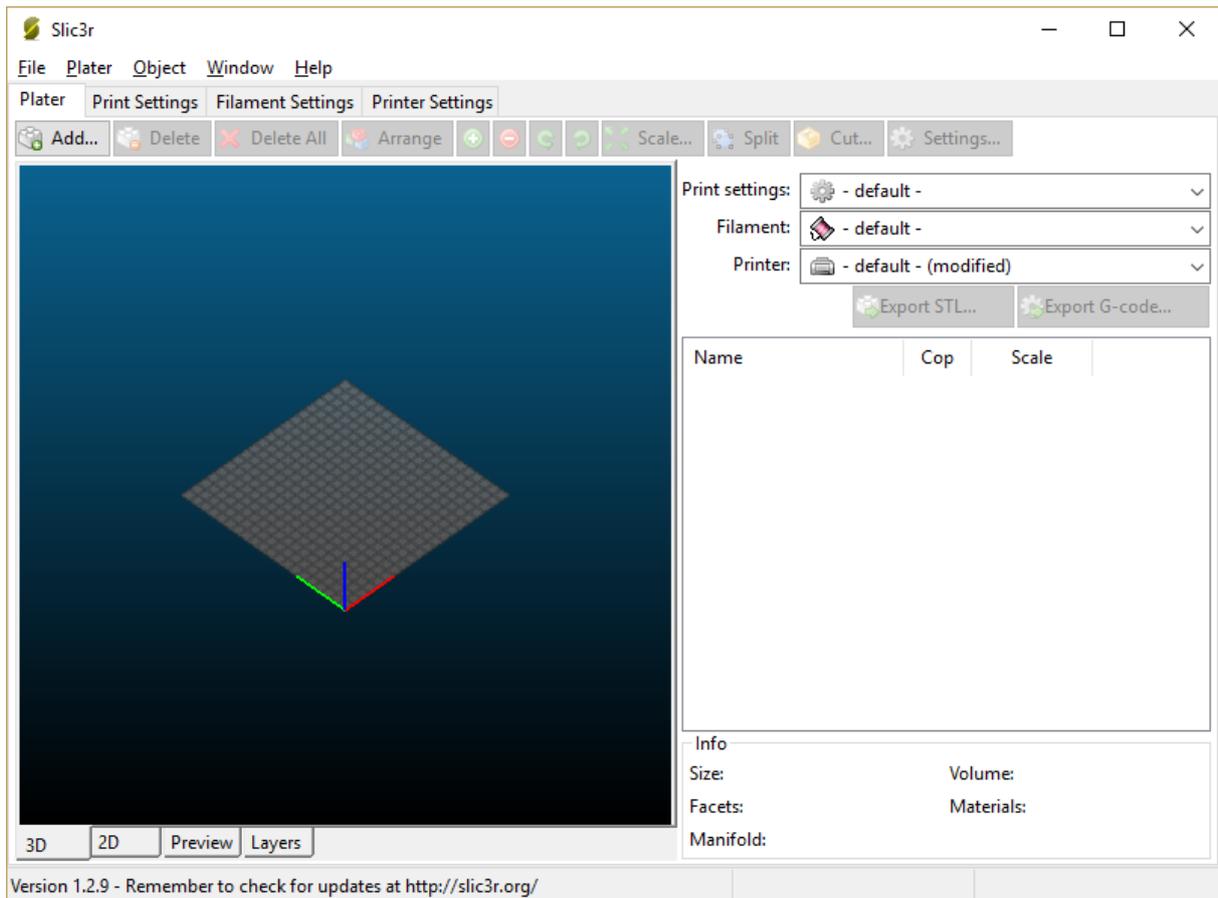
The program interface consists of four main tabs: Plater, Print Settings, Filament Settings, and Printer Settings.

Plater - serves to distribute objects after the print area

Print Settings - includes print settings such as speed, fill, support others.

Filament Settings - contains the material print settings such as temperature, diameter.

Printer Settings - includes printer settings for example nozzle diameter, print area.



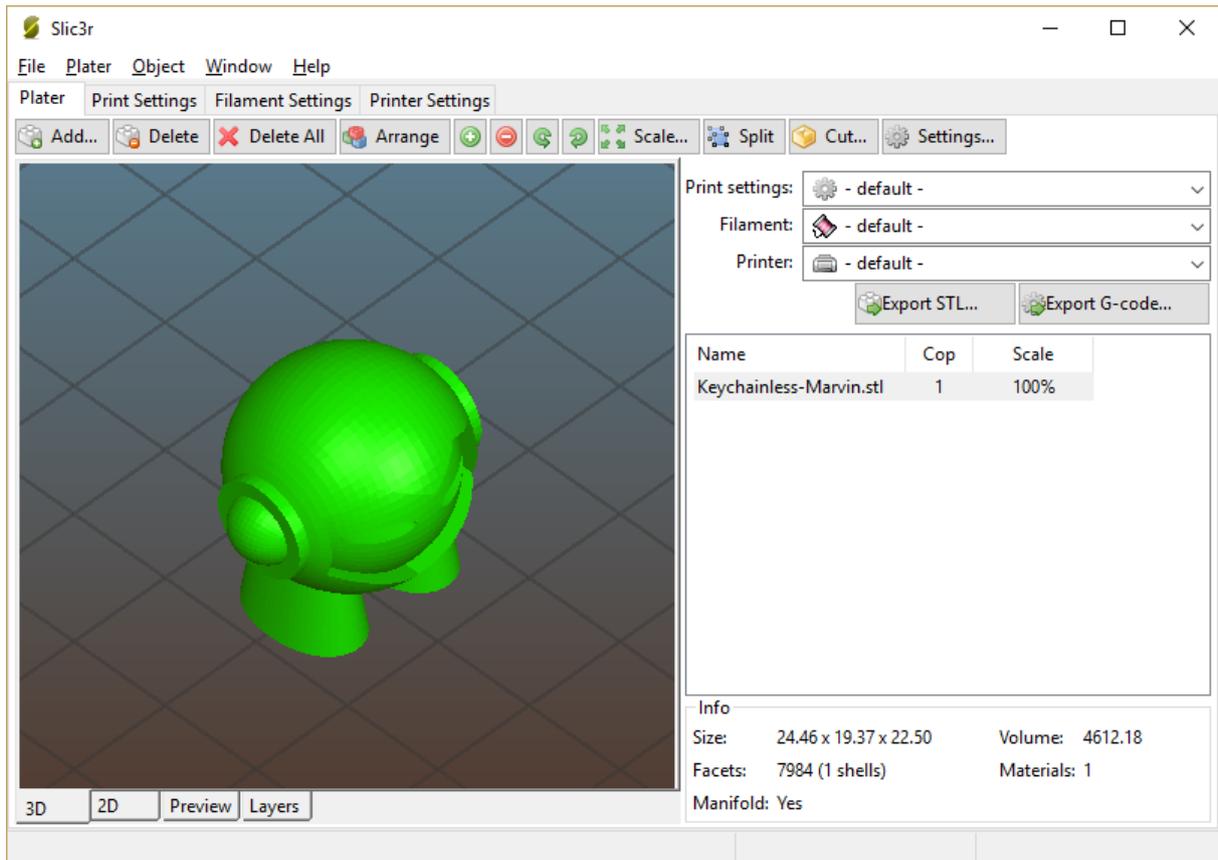
A pre-saved configuration file can be loaded. You can save the print configuration to your own file. (File/Export Config; File/Load Config)

BASIC SETTINGS

PLATTER TAB

Import model in STL format

The model is added by pressing the button Add...



The loaded model enables to use more functions - scale, rotate etc.

For generating a g-code for a 3D printer, press the Export G-code button.

PRINT SETTINGS TAB

LAYERS AND PERIMETERS

Layer height

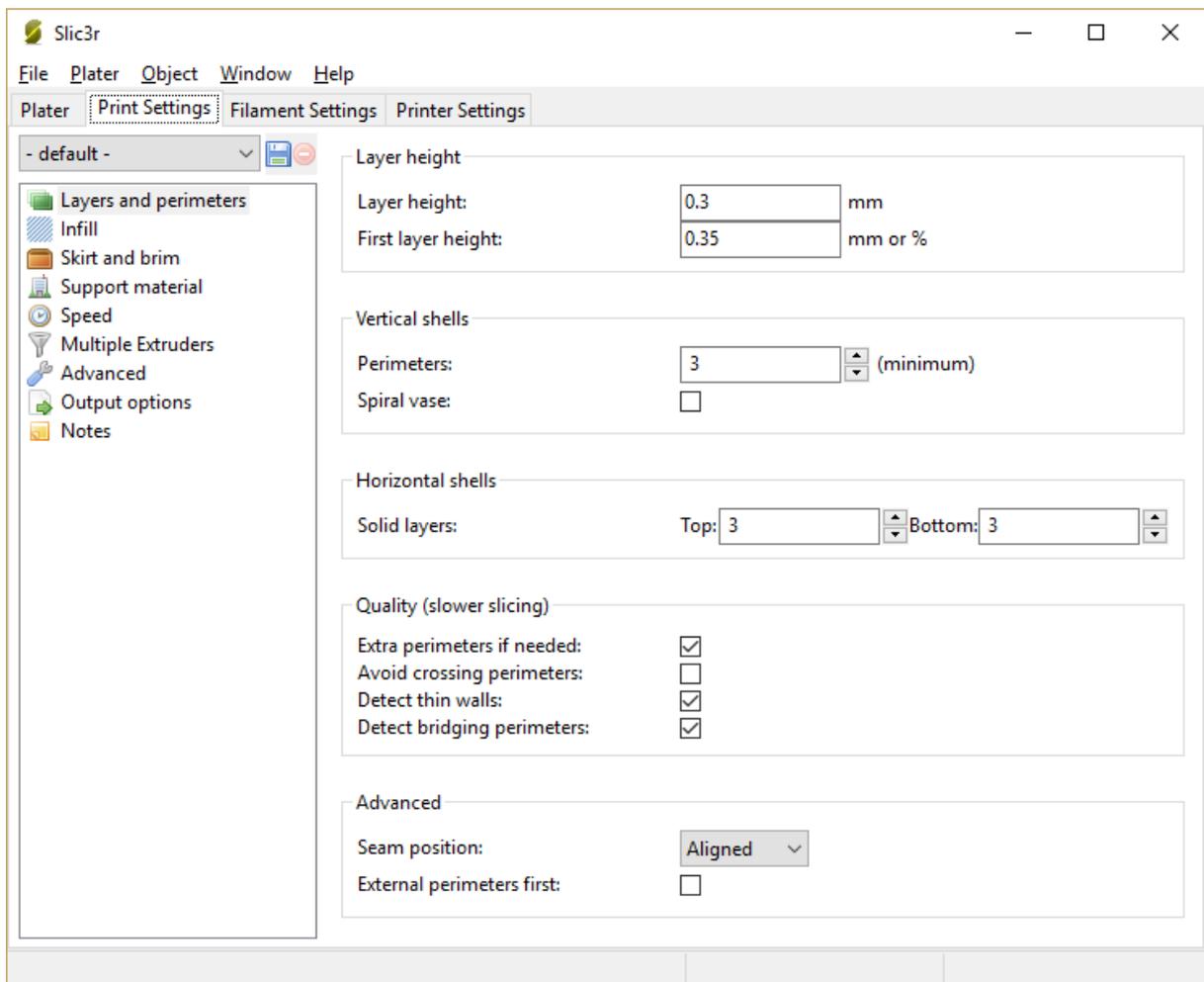
The value indicates the height of the printed layer. It affects, for example, the smoothness of the walls, the print speed, etc.

Vertical shells

The value indicates how many "threads" will form the shell of the object. A higher number means more fibers and therefore a thicker wall.

Horizontal shells

The value means how many full layers will form the top (Top) and bottom (Bottom) in the base of the object. For example, to print a vase or box is a value of the top equals to zero.



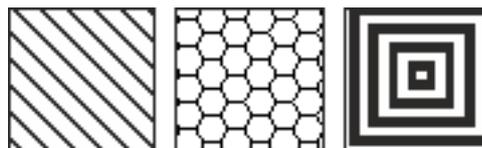
INFILL

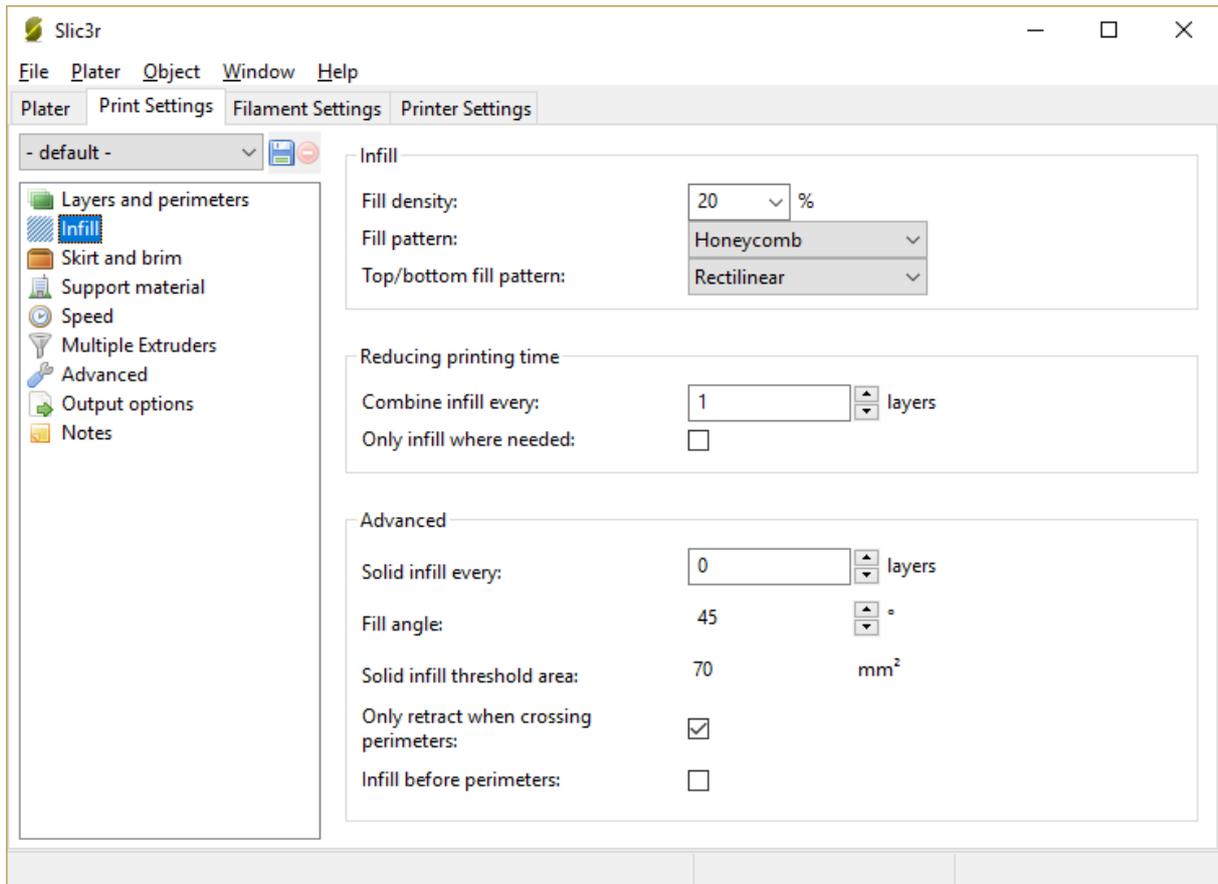
Fill density

The value indicates the "density" of the object's fill. Typical setting is 0 % for hollow objects, 20-40 % for common prints.

You can also set the Fill pattern within the object and optionally the Top and Bottom layers.

Fill pattern (example)

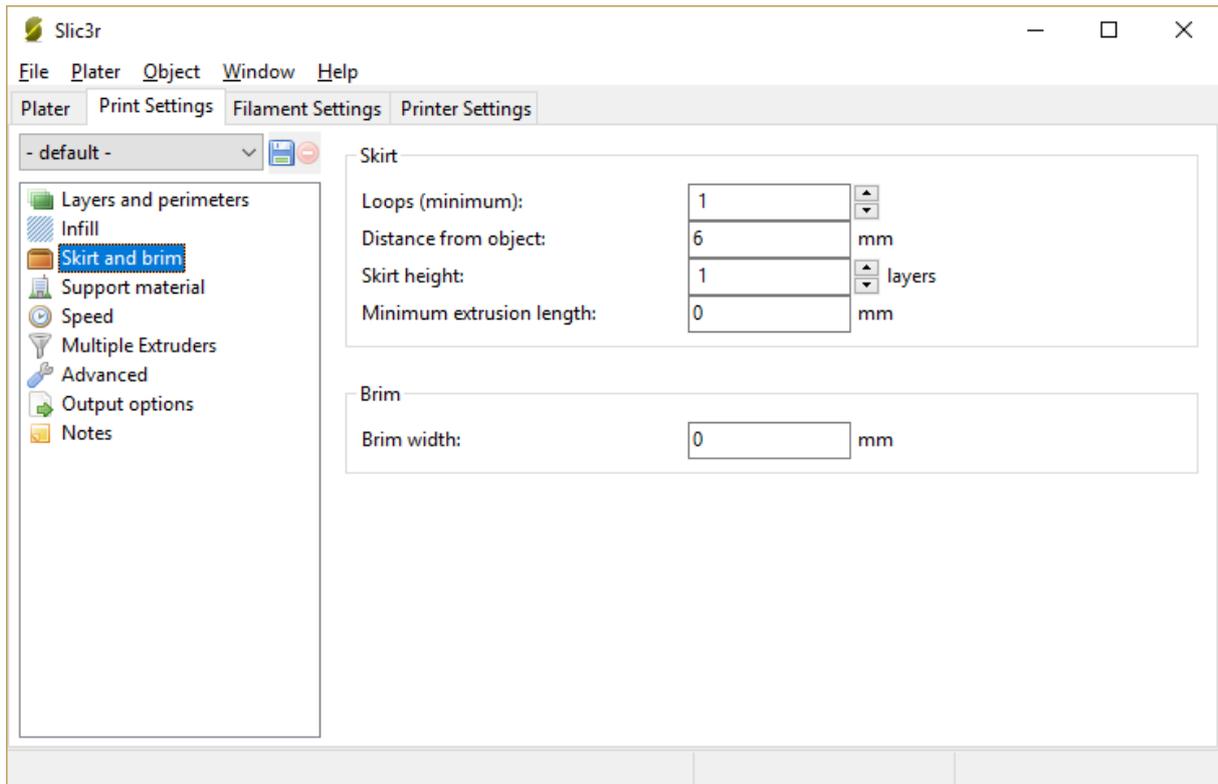




SKIRT AND BRIM

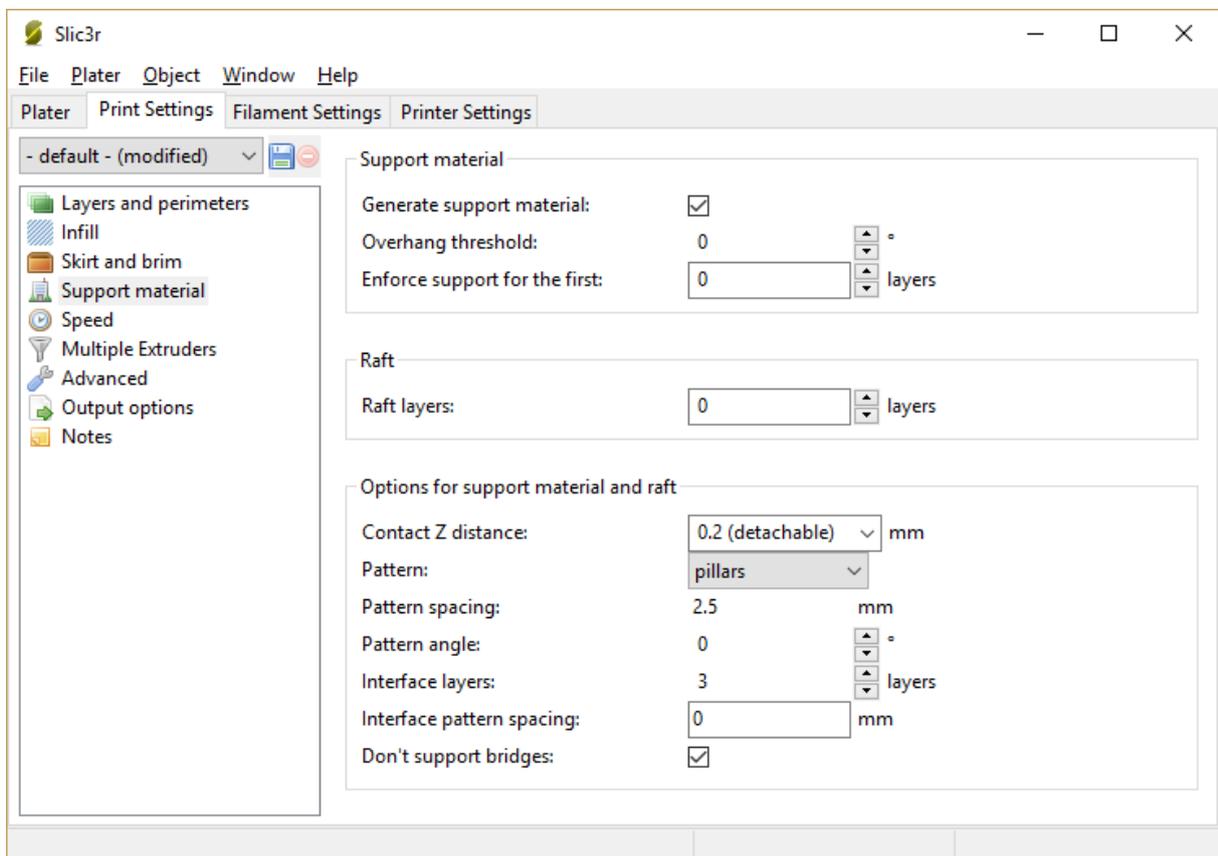
Skirt is the contour line spaced a few millimeters (Distance from object) from the contour of the object (the number is the parameter Loops). Used to allocate the print head. It also serves as a visual check of the nozzle distance from the pad.

Brim is used to increase adhesion of the object to the print heat bed. The brim extends the surface of the first layer along its perimeter.



Support material

If printing is in “the air”, it is necessary to generate supports.





Speed

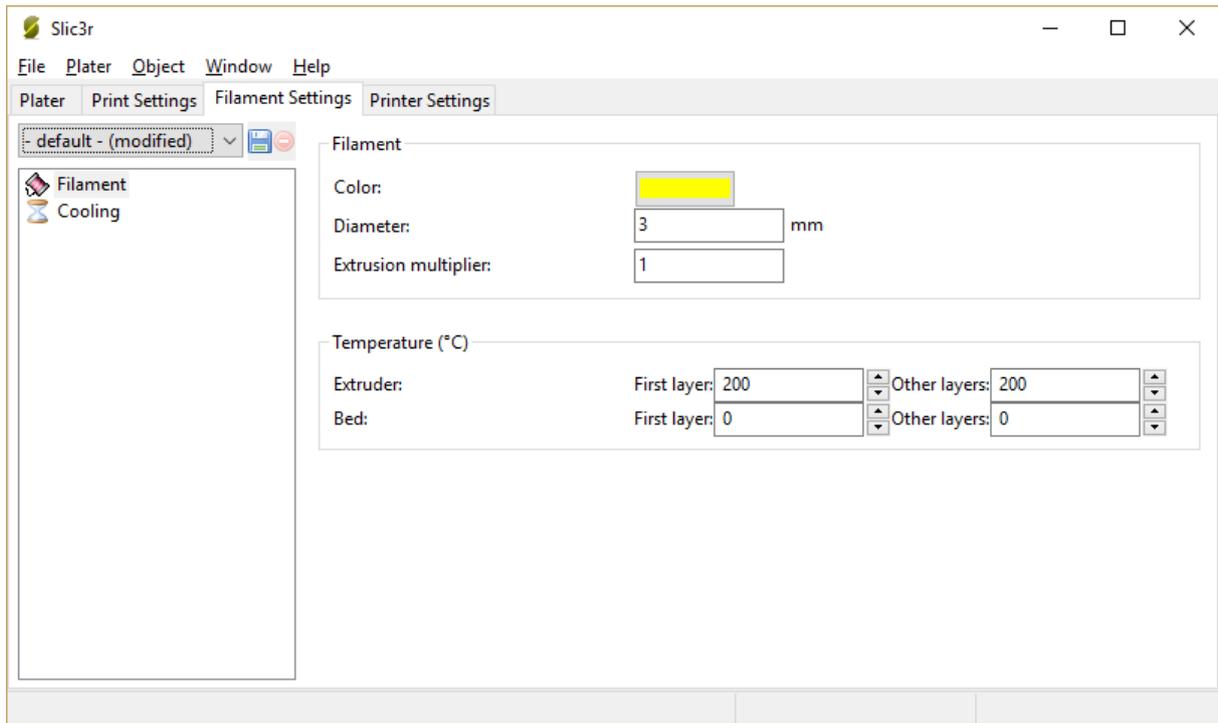
We keep the default settings.

The screenshot shows the Slic3r software interface with the 'Speed' settings panel selected. The panel is organized into several sections:

- Speed for print moves:**
 - Perimeters: 60 mm/s
 - Small perimeters: 15 mm/s or %
 - External perimeters: 50% mm/s or %
 - Infill: 80 mm/s
 - Solid infill: 20 mm/s or %
 - Top solid infill: 15 mm/s or %
 - Support material: 60 mm/s
 - Support material interface: 100% mm/s or %
 - Bridges: 60 mm/s
 - Gap fill: 20 mm/s
- Speed for non-print moves:**
 - Travel: 130 mm/s
- Modifiers:**
 - First layer speed: 30 mm/s or %
- Acceleration control (advanced):**
 - Perimeters: 0 mm/s²
 - Infill: 0 mm/s²
 - Bridge: 0 mm/s²
 - First layer: 0 mm/s²
 - Default: 0 mm/s²
- Autospeed (advanced):**
 - Max print speed: 80 mm/s
 - Max volumetric speed: 0 mm³/s

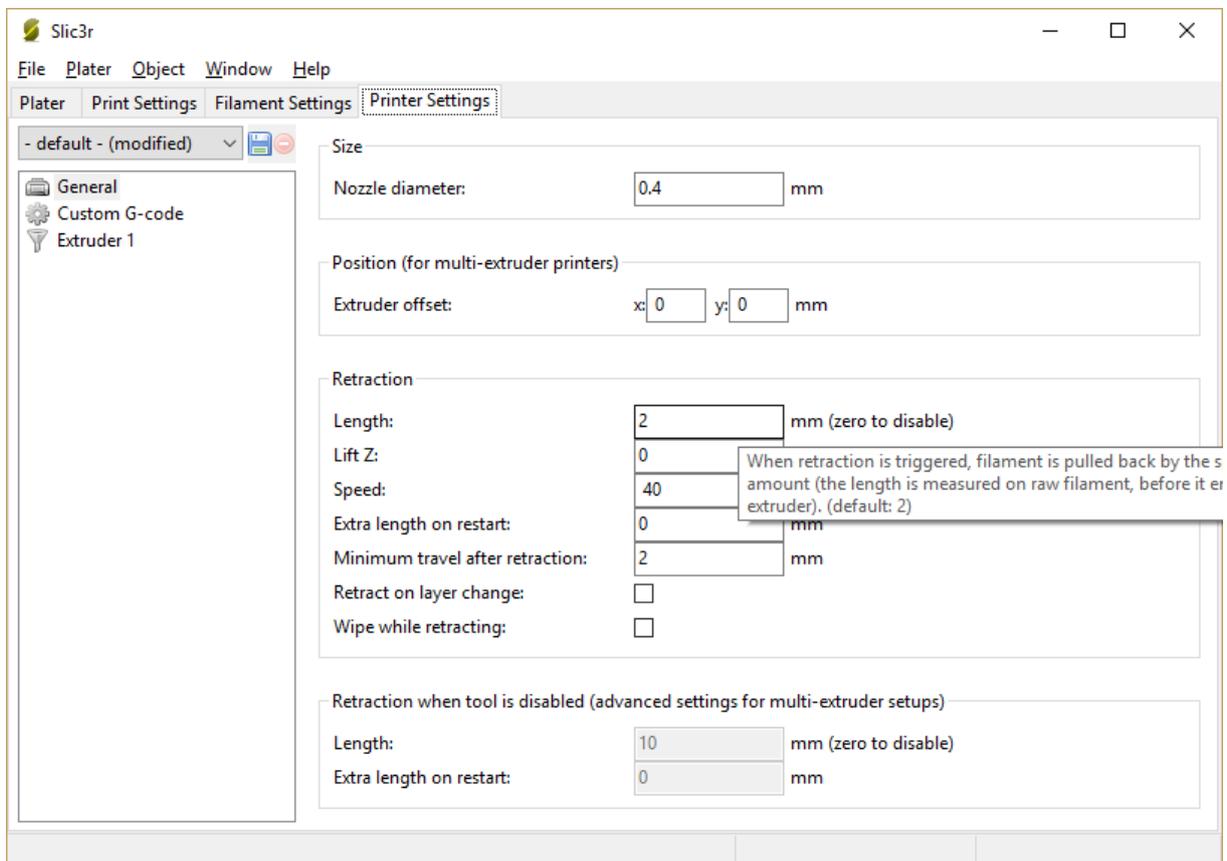
FILAMENT SETTINGS

You have to set the print string diameter.



PRINTER SETTINGS

Set the value - nozzle diameter





It is not necessary to set the Print settings each time, you can use the print profile save instead. (File/Export config).

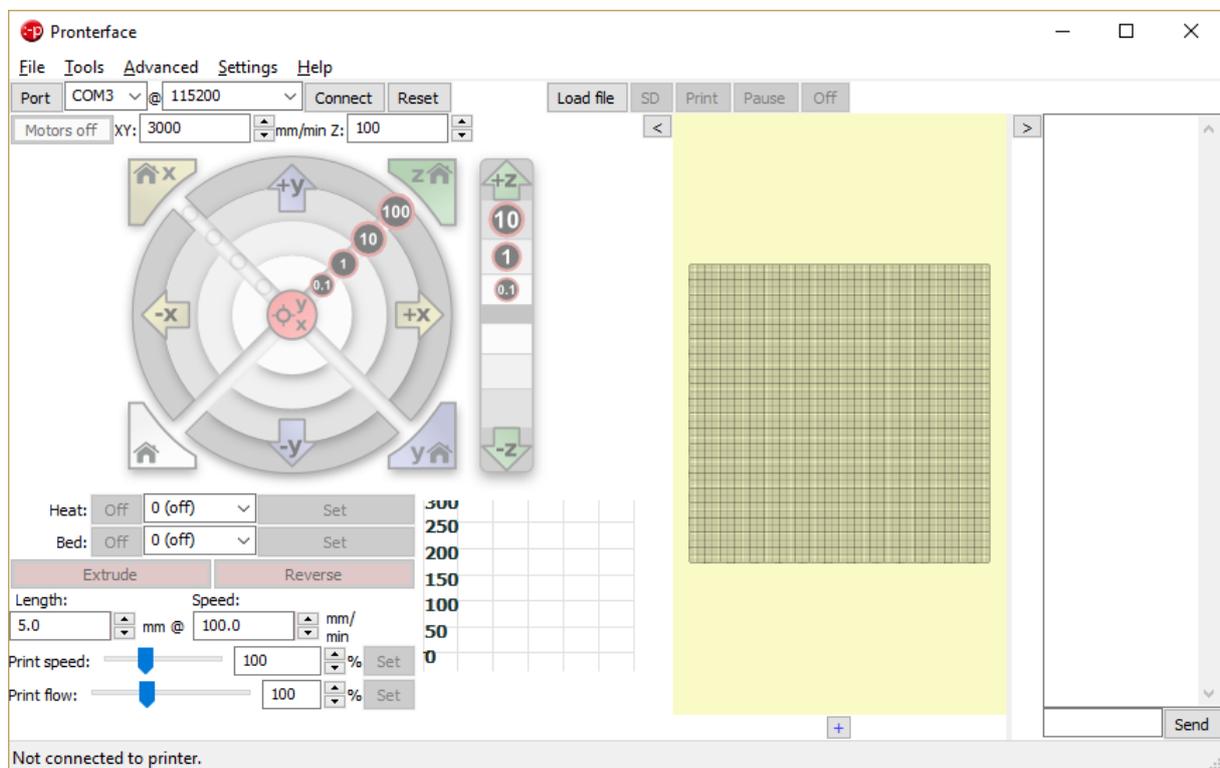
A detailed manual describing all details of the settings can be found on the website of the program.

HOW TO PRINT

You need to download a program for your own printing. The Pronterface can control the printer, visualizes individual print layers, and contains the integrated slicing program.

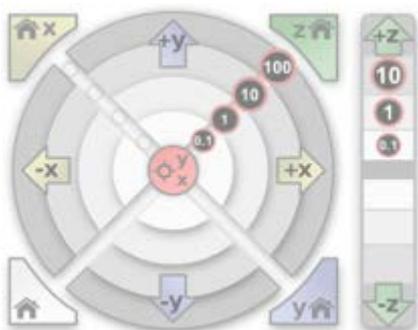
Pronterface can be downloaded directly from the address <http://www.pronterface.com/#download>.

After launching the application, you need to set Port (port and speed). The port number can be found in the administration of the computers. Then, connect to the printer using the Connect button.



If the connection has been made correctly, we should see the information about the connection in the console.

You run 3D printer by clicking on the image buttons.





Use the Load File button to upload a g-code model that was previously generated.

Here are the temperatures of extruder and heat bed carried out. For ABS and PET, the extruder temperature is set at about 285 °C, with a 90 °C heat bed. In the case of PLA, temperatures are somewhat lower, namely from 195 to 245 °C for the extruder and 50 °C for the heat bed. Confirm the setting by the click on the SET button.

After the extruder and heat bed temperatures reach the target values, you can start printing using the Print button. Firstly, the print head automatically moves to the base position and then begins to create the model layer by layer. Both monitored temperatures will be automatically maintained/adjusted to the desired values. Slight variations of their values do not indicate a defect.

Once the printing is completed, the printer automatically stops heating the extruder. The heat bed and the print head move to the beginning of the x-axis. Before removing the model from the heatbed, it is necessary to wait until the heatbed is fully cooled. A knife is the best tool for removing "well" stuck models.

