

Slic3r User Guide

By Lorenzo Cantini (Kent's Strapper) and Ivan Bortolin
Translation by Linda Anticoli

10/08/2012



Contents

Overview	i
1 Installation and first run	1
1.1 Installation	1
1.2 First run and wizard	1
2 Main window	5
2.1 <i>File, Window and Help</i> menus	5
2.2 Tabs	7
2.3 Load and export buttons.	7
2.4 File editing buttons.	8
2.5 Print settings buttons.	8
3 Print Settings	9
3.1 Save and load settings.	9
3.2 Layers and perimeter	10
3.3 Infill	11
3.4 Speed	13
3.5 Skirt and brim	14
3.6 Support material	16
3.7 Note	17
3.8 Output options	18
3.9 Multiple Extruders	18
3.10 Advanced	19
4 Filament Settings	21
4.1 Save and load settings	21
4.2 Filament	21
4.3 Cooling	22
5 Print Settings	23
5.1 General	23
5.2 Retraction	24
5.3 Custom G-Code	25
5.4 Extruder	25
6 Useful links	27

Overview

Premise

This document is user guide for Slic3r, a slicing software for 3D RepRap printers and similar.

This software has been made by Alessandro Ranelucci, who we wish to thank; another thank-you is for RichRap whose guide has been used as hint and starter.

This document has been realized by Lorenzo Cantidi (Ken's Strapper) and by Ivan Bortolin; if you find a errors in this guide, please send the details to kentstrapper@gmail.com or ivan.bortolin@gmail.com.

Transalition is by Linda Anticoli linda.anticoli@gmail.com.

The authors are not responsible for any error or harm to people, or other problems caused by the use of this guide.

Introduction

With Slic3r Alessandro Ranelucci gave his contribution to the development of 3D printing. This project, started as an experiment, has in only a few months experienced unexpected success, becoming an international reference for the 3D printing field.

A community with hundreds of active contributors and thousands of users was spontaneously born around this project; a forum dedicated to this project has been activated in the official site of the RepRap project and an official chatroom is born with about 80 users average in it. Slic3r is mentioned in the official RepRap blog and it is used by some of the main community members, from the university of Bath to the producers of printers and kits. Slic3r holds a record for the highest resolution 3D print (10 micron).

What exactly does Slic3r do? Slic3r transforms a 3D model (i.e. a file representing the object to print) in a set of instructions which guide the printer and allow it to put down layers through its engines and its extruder. The software uses a lot of mathematical algorithms working on model's geometry, and selecting the best path for extruding material that will compose single layers: perimeters, honeycomb infill to spare material with the same structural resistance, bridges, obliquous sides, temporary supports to sustain overhanging threshold. Moreover, Slic3r decides the amount of material to extrude, controls temperature, fans and cooling time. ¹

Slic3r is a core software for 3D printing and it's made in Italy.

We are going to analyze version 0.9.0 of the software. In this version there are many news like a better management of multiple extruders, the possibility to import files in .obj and .amf and the possibility to export files in .svg format.

¹<http://makerblog.it/post/18788200901/cose-slic3r>

Chapter 1

Installation and first run

1.1 Installation

Slic3r is available for Linux, Windows and Mac OS-X.

Please follow the links below to find the correct guide for your environment:

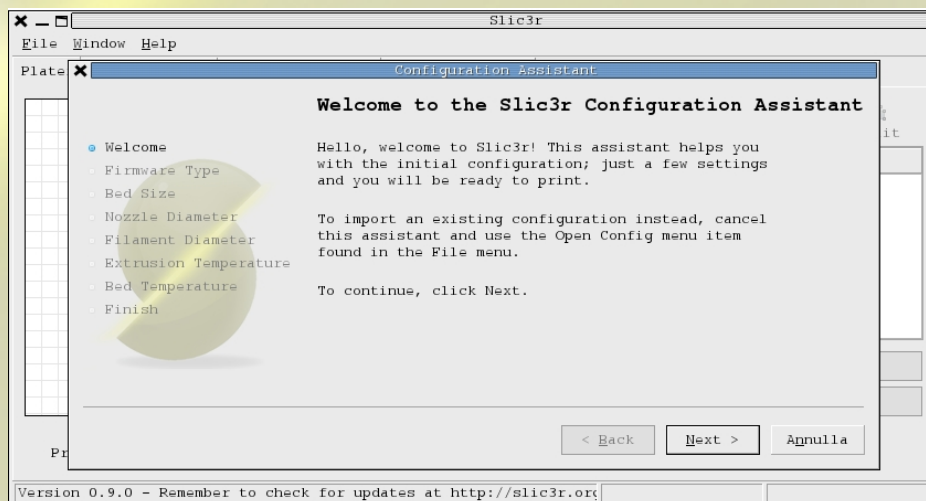
- Linux¹.
- Windows
- MacOs

1.2 First run and wizard

On the first run of the program, we can follow a small wizard to insert basic settings to obtain good G-Code.

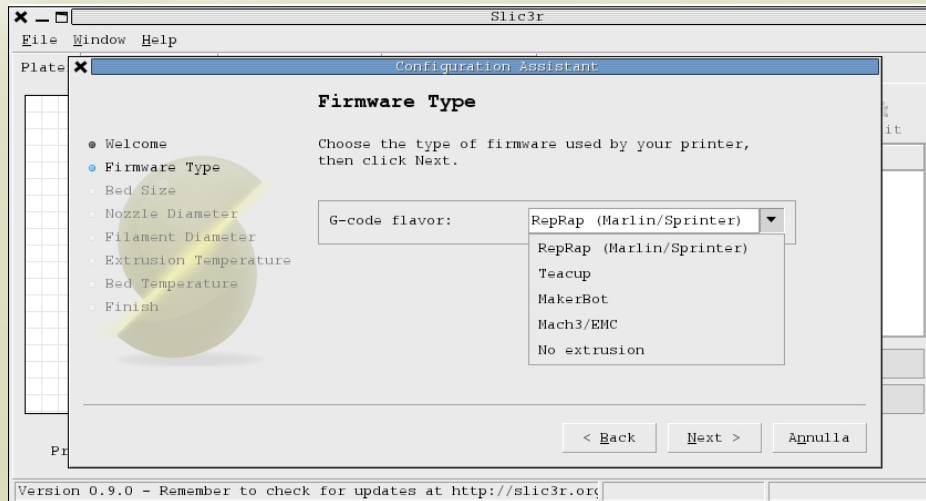
These settings are obviously present in the following sections of this guide.

Let's look the steps of this configuration.

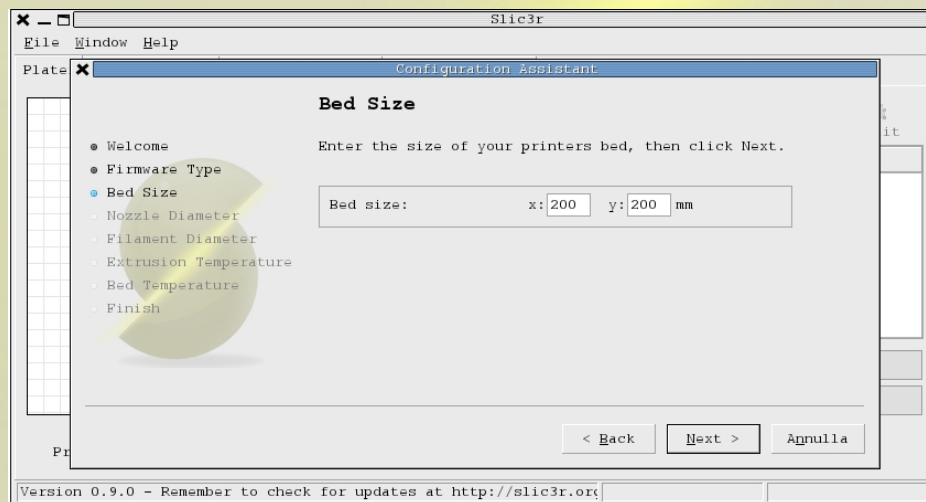


¹For versions of Ubuntu before 12.04 you may have dependency issues; we recommend to follow this guide

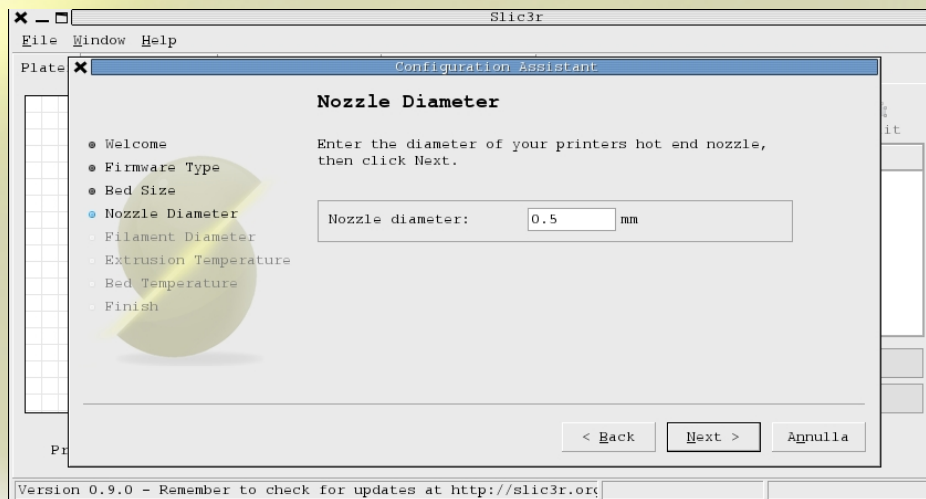
This is the welcome screen of the Wizard. Following this one, we have the settings screen for the firmware we have previously loaded on our electronic board.



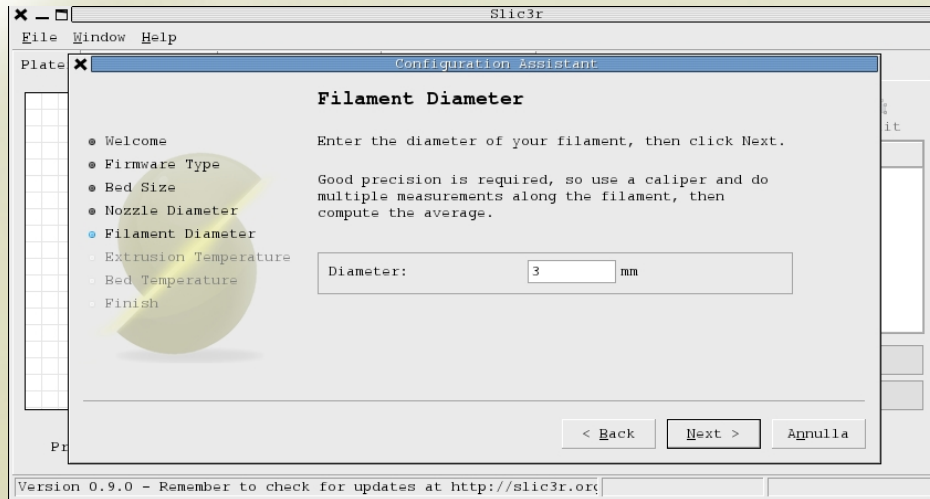
From the pull-down menu, select the firmware installed into your controller and then click *Next*.



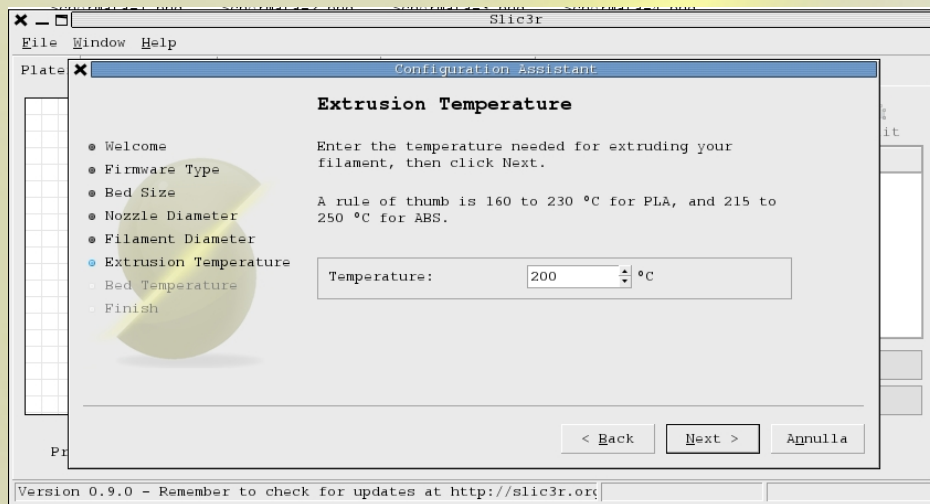
In this window we have to insert the size of the printer bed.



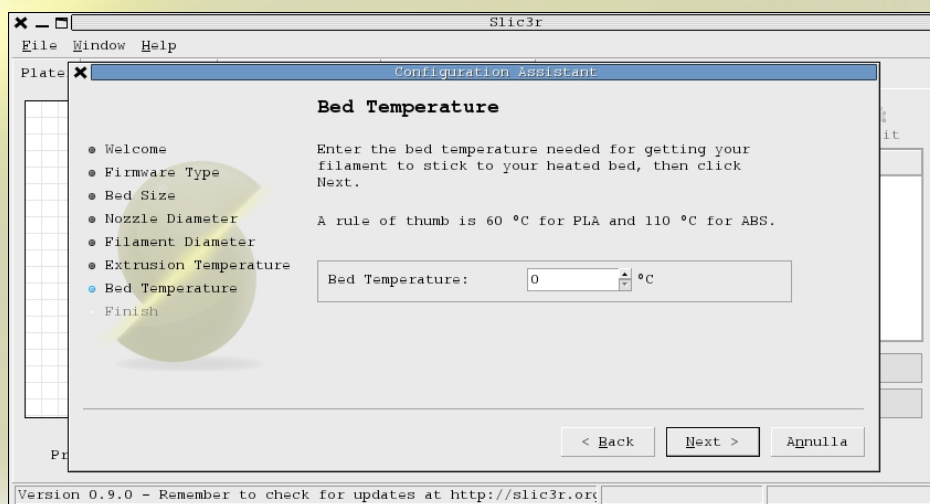
Here, we have to insert the nozzle diameter mounted on the *HotEnd*.



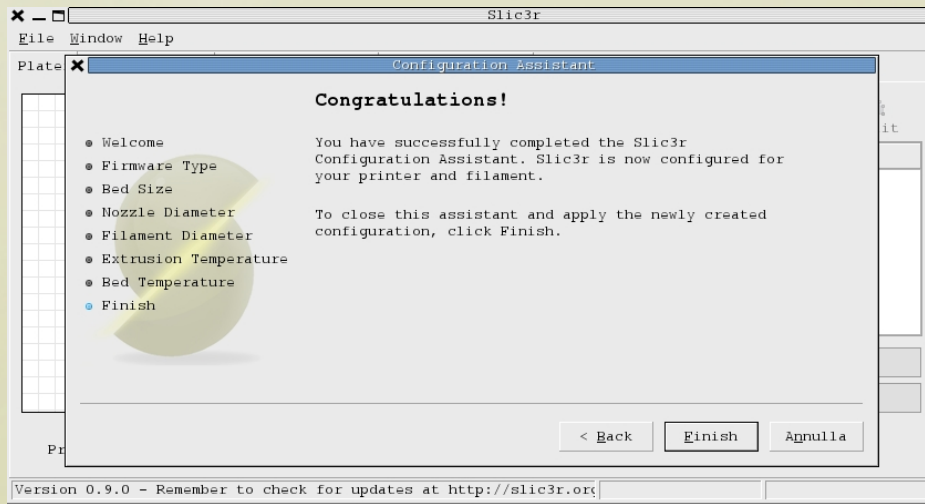
Here, we have to insert the filament diameter.



In this window, we have to insert the extrusion temperature for the *HotEnd*. Usually it's 230° for ABS and 190° for PLA.

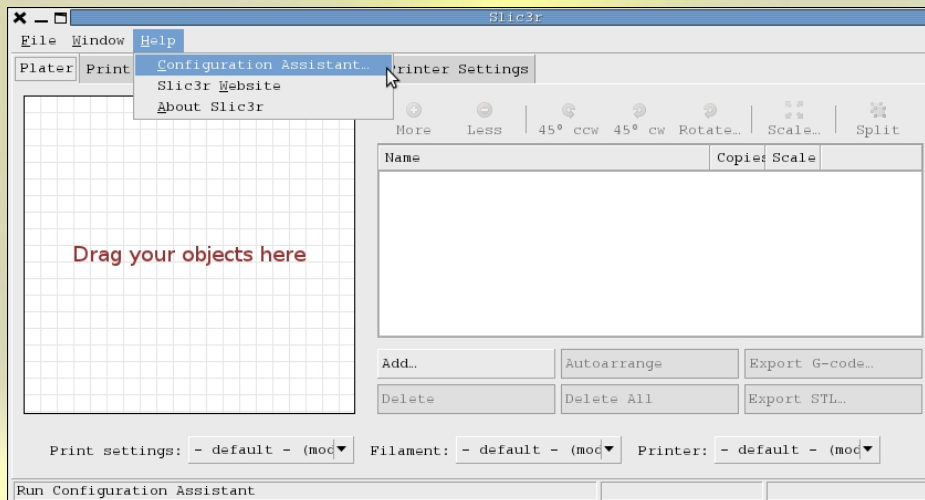


In this window, we have to insert the *HeatedBed* temperature. Obviously, if the bed is not heated, you should set this value to 0° C. For PLA it's not necessary to heat the printing bed, although it's recommended to heat it to 60° C. For ABS, in the case we are not using *Award*² it's necessary to heat it at least to 110° C.



This is the last window.

In the case you need to reset the Wizard, you can click on *Help* and then on *Configuration Assistant*.

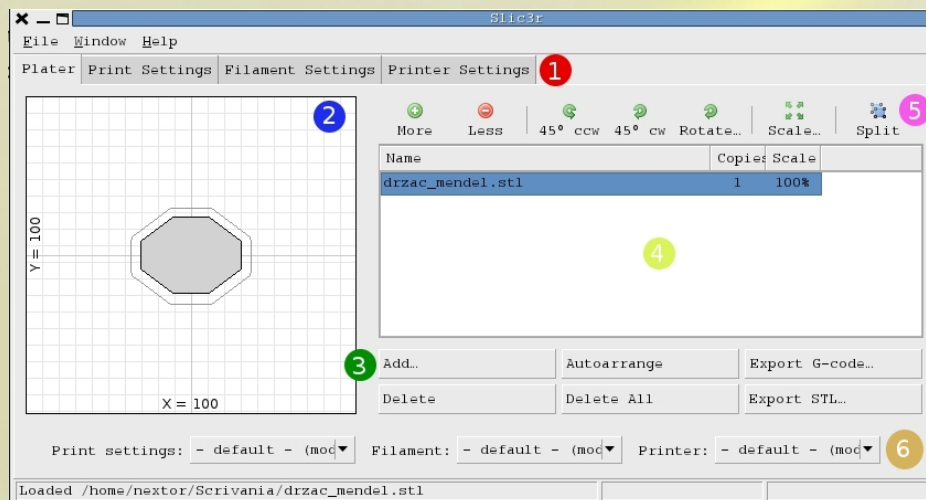


²http://reprap.org/wiki/AWARD_BED

Chapter 2

Main window

This window appears when Slic3r 0.9.0 is started.

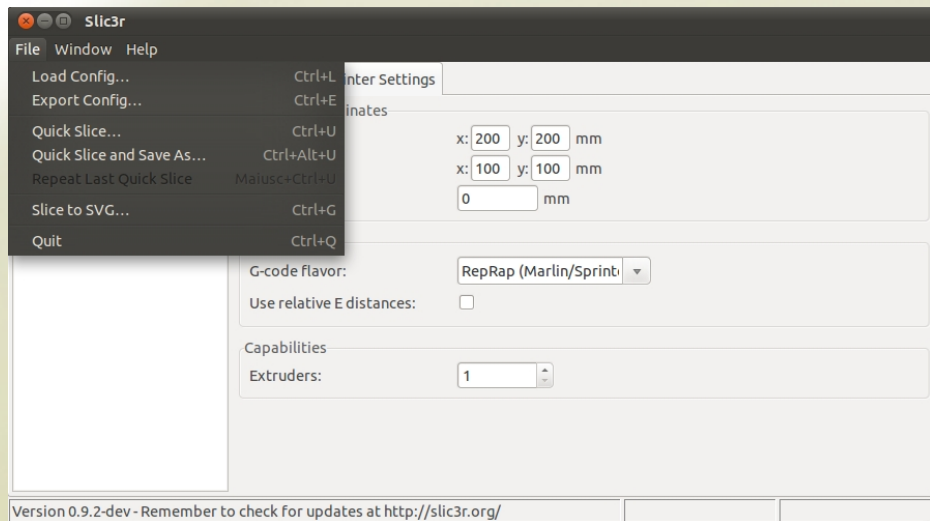


This window, is composed by the *File*, *Windows* and *Help* menus and:

- 1: a set of tabs;
- 2: a graphic area representing the printing bed;
- 3: a set of buttons, used to load and delete files in the program;
- 4: a list of loaded files;
- 5: a set of buttons, used to edit the project.
- 6: a set of buttons with printing settings.

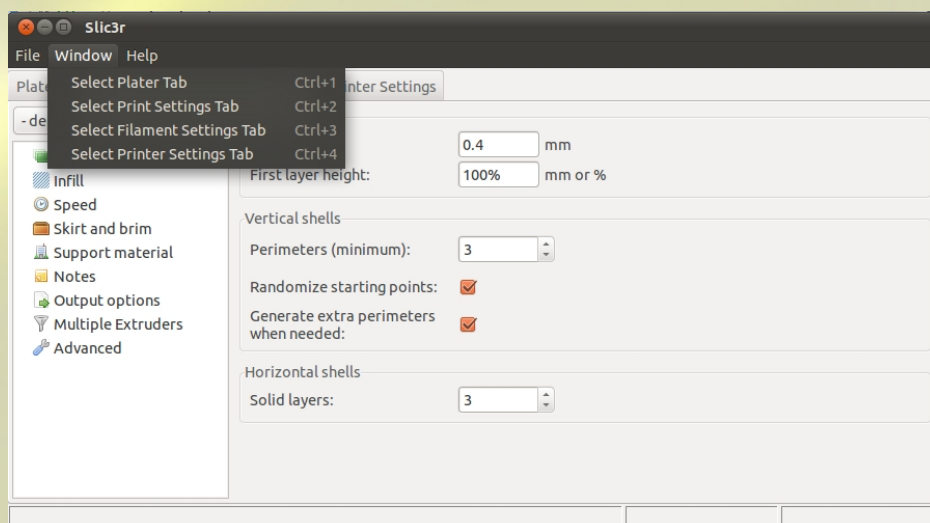
2.1 *File*, *Window* and *Help* menus

In the *File* menu we have the following options:

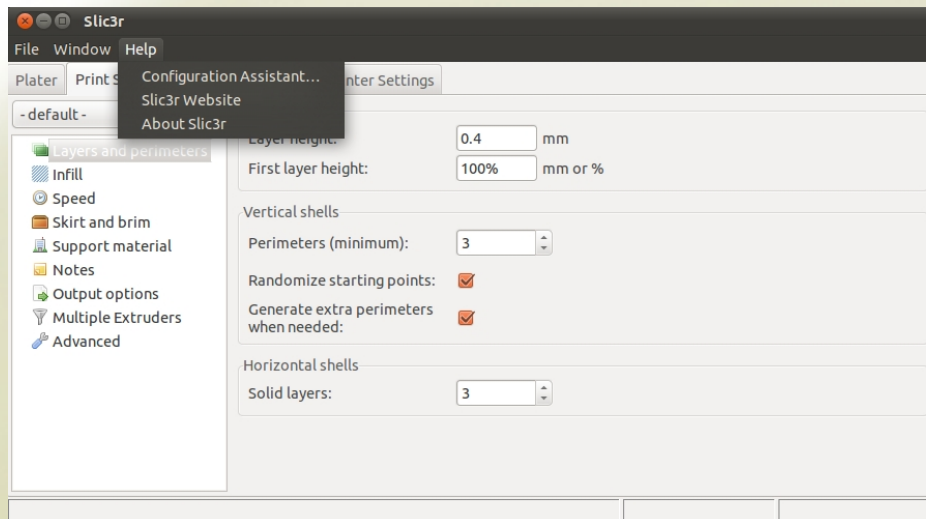


- **Load Config:** Loads previously saved configurations of the program, which are stored in a .ini file. In this version and onwards different configurations can be separated, as you can see in the following chapters.
- **Export Config:** Saves configuration in a .ini file.
- **Quick Slice:** Starts a G-Code processing with default settings.
- **Quick Slice and Save As:** Starts a G-Code processing with default settings and renames the file.
- **Repeat Last Quick Slice:** Repeats the last processing.
- **Slice to SVG:** Exports the project in a .svg file.
- **Quit:** Closes Slic3r.

The *Window* menu allows the selection of working areas, which we are going to analyze in the next chapters.



In the *Help* menu we can find:



- **Configuration Assistant:** starts the wizard with basic configuration settings.
- **Slic3r WebSide:** Opens the Slic3r's web page in your web browser.
- **About Slic3r:** general information on Slic3r.

2.2 Tabs

The following tabs (1) are available:

- **Plater:** Main tab.
- **Print Settings:** Printing settings as perimeter management, layer distance, support material, engine speed, and so on;
- **Filament Settings:** Filament parameters, extrusion and heated bed temperature regulation, cooling fan settings;
- **Printer Settings:** Printing area settings, nozzle diameter, G-Code personalization.

2.3 Load and export buttons.

- **Add:** Loads one or more files in .stl, .obj or .amf format.
- **Delete:** Deletes selected files from the project.
- **Autoarrange:** Automatic correction of the file.
- **Delete All:** Removes all files from the program.
- **Export G-Code:** Exports the 3D file in G-Code.
- **Export STL:** Exports the project in a .stl file. Useful when you've edited the project.

2.4 File editing buttons.

The button group (5) over the table are automatically **activated** when you load a file with the *Add* button:

- **More:** Adds another copy of the same file into the project. The copy is visible in the printing area.
- **Less:** Deletes a copy of the file from the project.
- **45° ccw (counter-clockwise):** Rotates the object 45° counter clockwise.
- **45° cw (clockwise):** Rotates the object 45° clockwise.
- **Rotate:** Rotates the object by an user-defined value.
- **Scale:** Scales the object by an user-defined value.
- **Split:** Splits the object in different parts. The file must be composed by more than one part.

2.5 Print settings buttons.

With the new pull-down menus (6) it's possible to load default or personalized settings:

- **Printing Settings**
- **Filament Settings**
- **Printer Settings**

Chapter 3

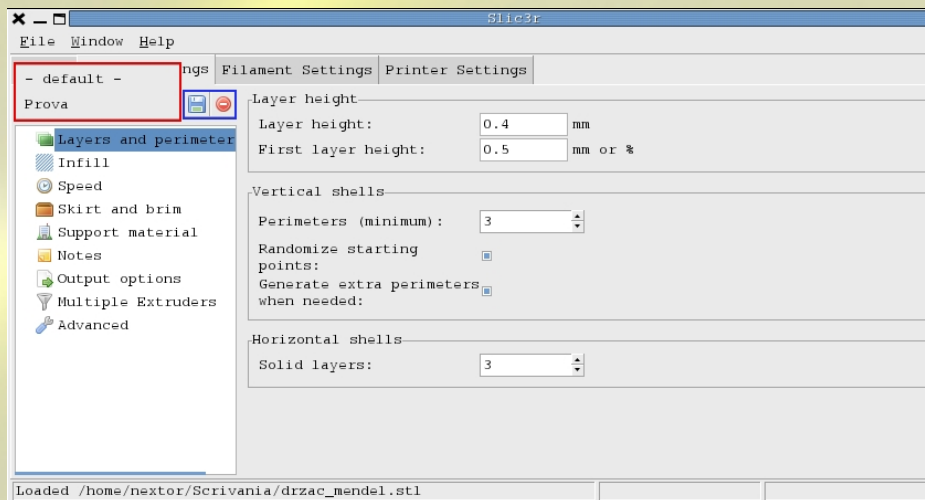
Print Settings

3.1 Save and load settings.

As we said before, we're now able to load previously saved configurations. This is very useful because allows us to set up the software more quickly and avoids distraction errors.

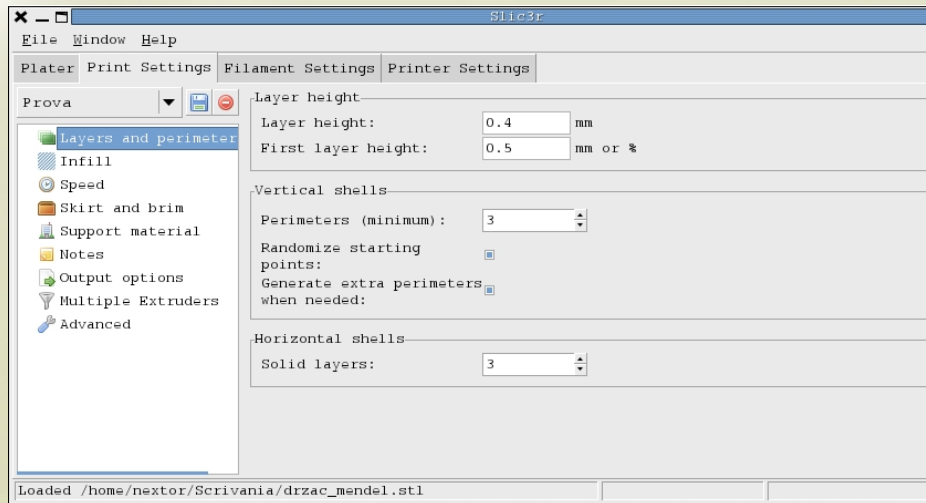
These settings can now be saved or loaded faster, while the operation was significantly longer in the previous version.

This simplicity has been obtained by adding two buttons and a pull-down menu.



The buttons highlighted in the blue rectangle are respectively used to save settings (as *default* settings or customized settings) or to delete settings. The pull-down menu highlighted in the red rectangle allows us to select the settings we want to edit or delete.

3.2 Layers and perimeter



In this section the have the following settings:

- **Layer height**

- Layer height. The offset over the Z axis between two layers. It's an important setting, as it must be compatible with the nozzle diameter. You can use a 0.3 mm value for a 0.5 mm nozzle. We recommend not to use values above 0.4 mm; the best quality is obtained with lower values. You can use values down to 0.1 mm.
- First layer height. You can set it in mm or in %. We recommend tusing the same Layer Height value defined above or adding at most 25% of the Layer Height value.

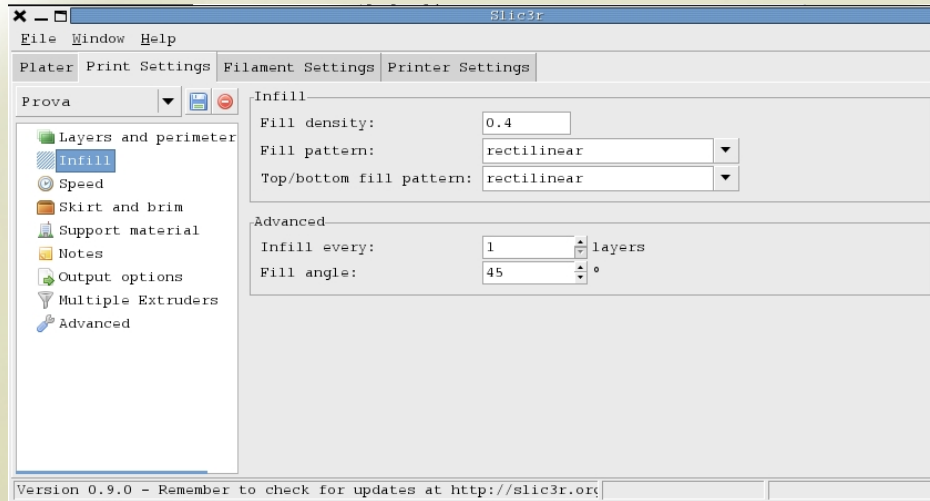
- **Vertical shells**

- Perimeters (minimum): Defines the lowest numbers of solid contours for the object perimeter. For very thin objects, the program may generate only one or two contours. You may want to use less contour and more infill density.
- Randomize starting point: self-explanatory.
- Generate extra perimeters when needed: augments the quality of the 3D print generating more perimeters, but decreases speed.

- **Horizontal shells**

- Solid layers: the quantity of solid layers that are made on the top flat surfaces. A good value for this setting is 3; we recommend not to use values above 6.

3.3 Infill



- **Infill**

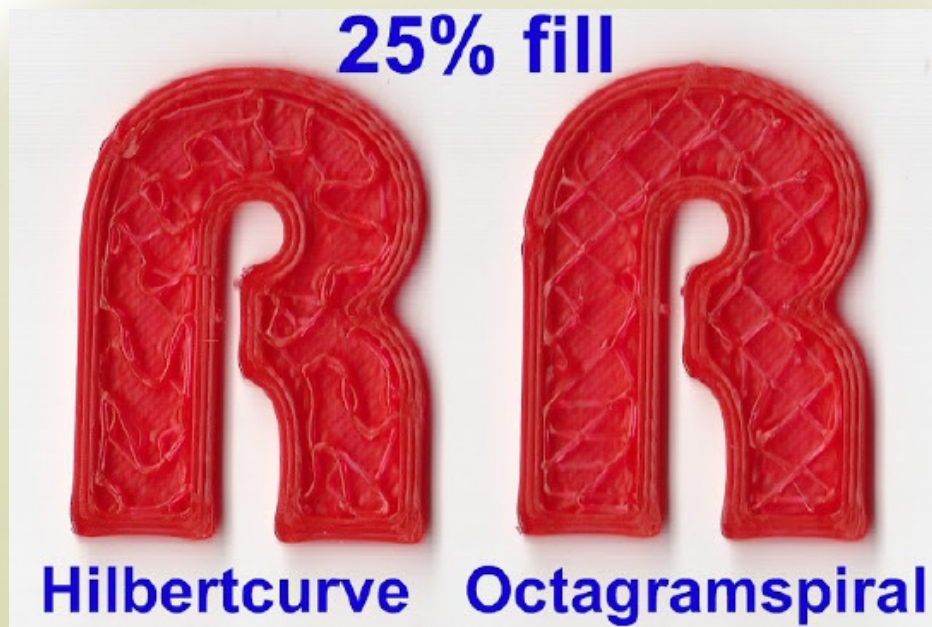
- Fill density: Set the infill amount which will be used for the 3D print to fill the internal empty space. For example, a value of 0.4 means a 40% filling. We recommend to use a value between 0.25 and 0.5.



- Fill pattern. These fill patterns are available:
 - * Rectilinear
 - * Line
 - * Concentric
 - * Honeycomb
 - * Hilbertcurve (slow)
 - * Archimedeananchor (slow)
 - * Octagramspiral (slow)

You can see some examples below.





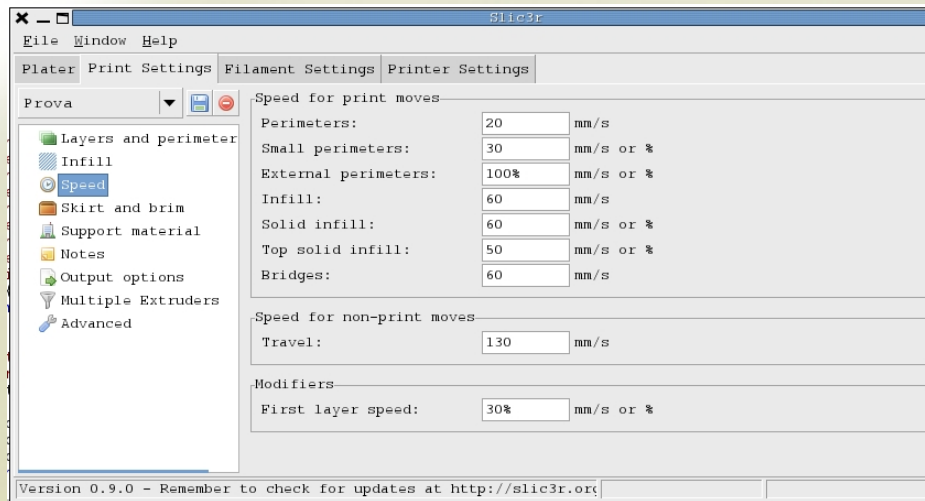
- Top/bottom fill pattern: as the previous setting, but only for first and last layer. The available patterns are:
 - * Rectilinear
 - * Line
 - * Concentric
 - * Hilbertcurve (slow)
 - * Archimedean chord (slow)
 - * Octagramspiral (slow)

- **Advanced**

- Infill every: Sets the printing height depending from the perimeter height. For example, when set with *Layer Height 0.2*, *Infill every Layer 2*, RepRap will print with 0.2 perimeter height and 0.4 fill height. It's used to decrease printing time without decreasing the perimeter quality.
- Fill angle: sets the angle between X and Y axis, which will be used for the object filling.

3.4 Speed

In this section, motor speeds are set.



The customizable speeds for the choosing are:

- **Speed for print moves**

- Perimeters: Object’s perimeter printing speed. In order to have high quality in projects it is better to have a low perimeter printing speed value.
- Small perimeters: small perimeters printing speed. It is better to keep this value low to avoid deformations caused by the fast passage of the HotEnd upon the same area.
- External perimeters: External perimeter printing speed. It is better to keep this value the same as “Perimeters” or lower.
- Infill: object’s infill speed.
- Solid infill: “solid” infill speed, i.e. layers before the top layer.
- Top solid infill: top layer “solid” infill speed.
- Bridges: “Bridges” production speed, “Bridges” are span filaments across a gap in free-air with or without support.

- **Speed for non-print moves**

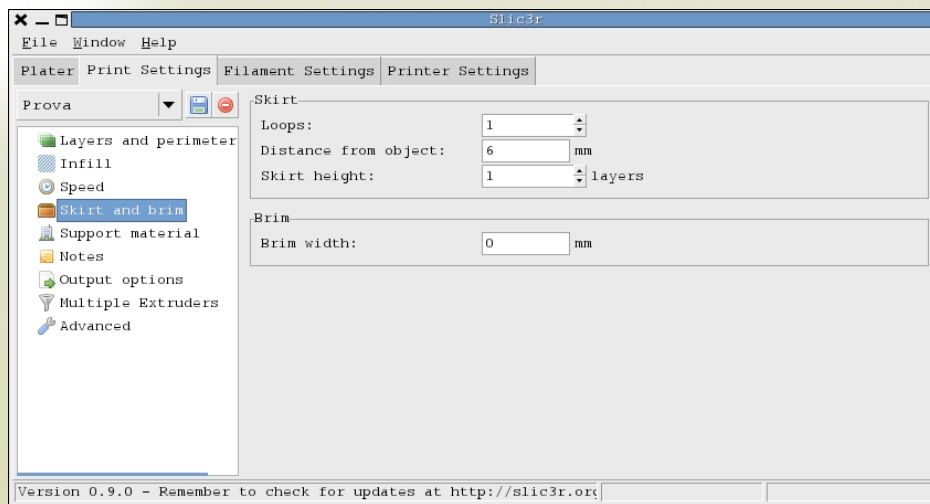
- Travel: movement speed of the carriage from one area to another when there is no extrusion.

- **Modifiers**

- First layer speed: First layer production speed. It is better to keep this value low to permit the best adherence of the plastic to the printing bed.

3.5 Skirt and brim

This parameter draws a line around the printed object’s perimeter. It permits a good flow before the print of the object, avoiding “first layer adherence” issues. It also helps the user to see if the extruder is set at the proper height. Moreover it has been added the *Brim* that realizes a first layer perimeter increase, in order to enlarge the adherence surface and avoid deformations. It is similar “Raft” in Skeinforge.



The parameters are:

- **Skirt**

- Loops: number of lines we need to realize before our first real print of the object. For small objects it is appropriate to have at least 2 lines.
- Distance from object : distance between the line we want to draw and the object.
- Skirt height: Printing height of the Skirt.



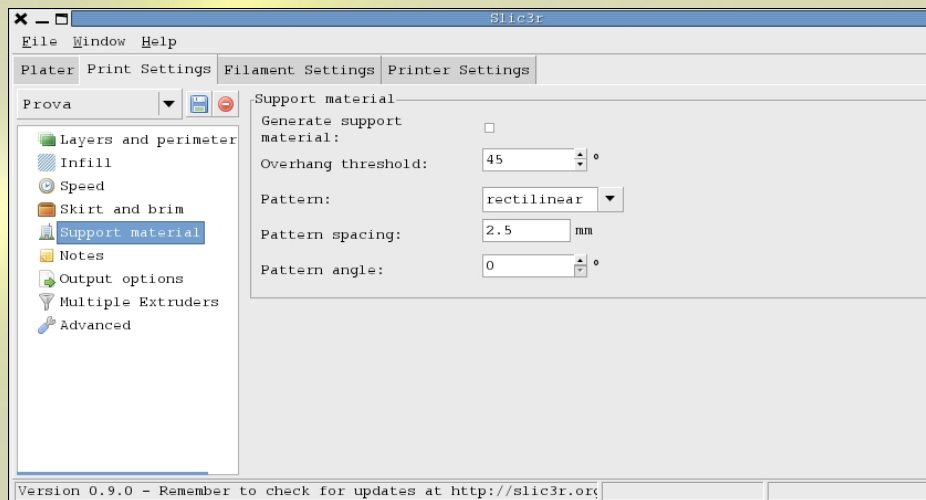
- **Brim**

- Brim width: Brim dimension setting. It is better to set it up to 3 when we use ABS to print, or to 6 when we have objects with huge perimeters.



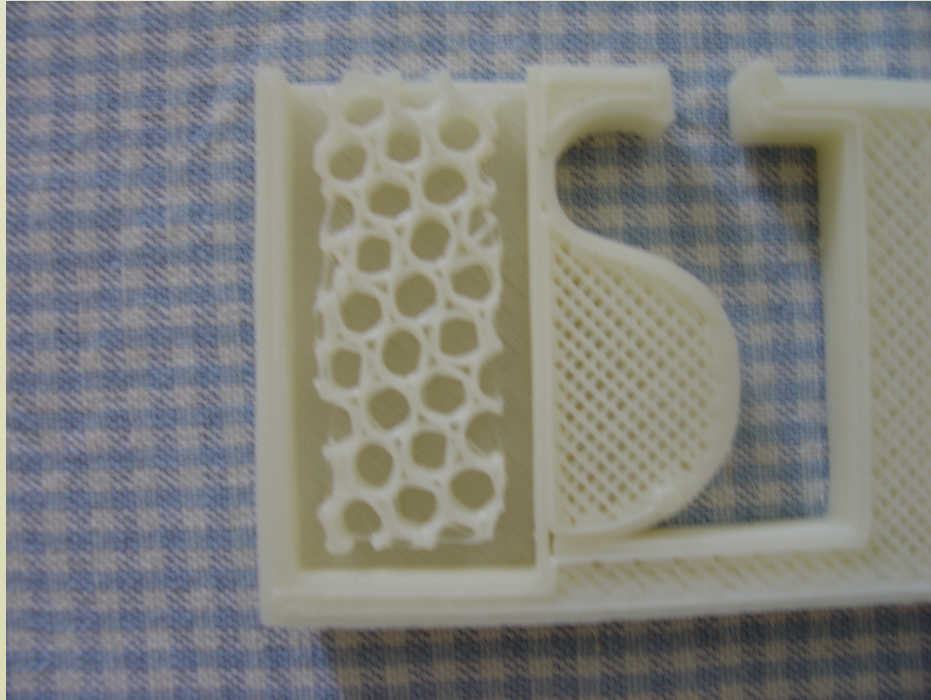
3.6 Support material

In this section it is possible to enable supports configuration and generation. When you enable it you will create a layer that will support back draft areas.



- Generate support material: checking it you will enable supports generation.
- Overhang threshold: identifies the back draft angle sustaining the support.
- Pattern: defines support geometry that can be:

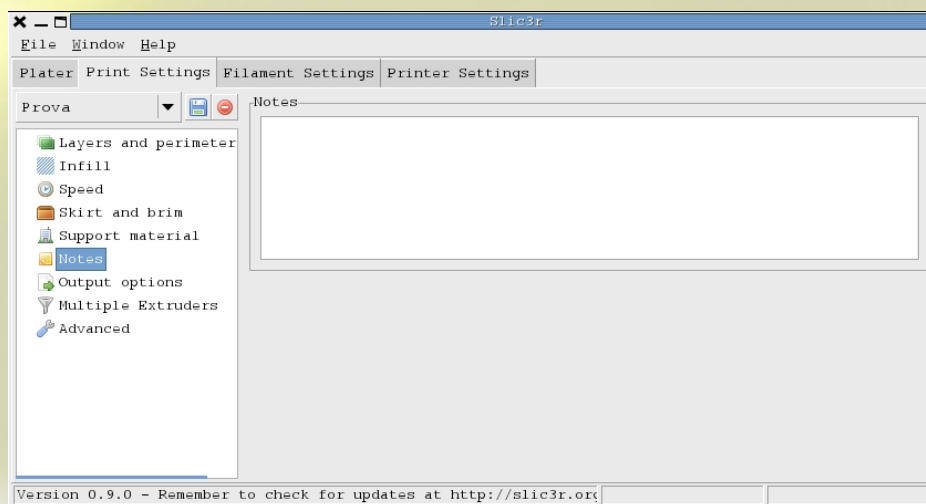
- rectilinear.
- honeycomb.



- Pattern spacing: identifies the area (in mm) between support layer and the other layer from an horizontal point of view.
- Pattern angle: identifies the support layer production angle.

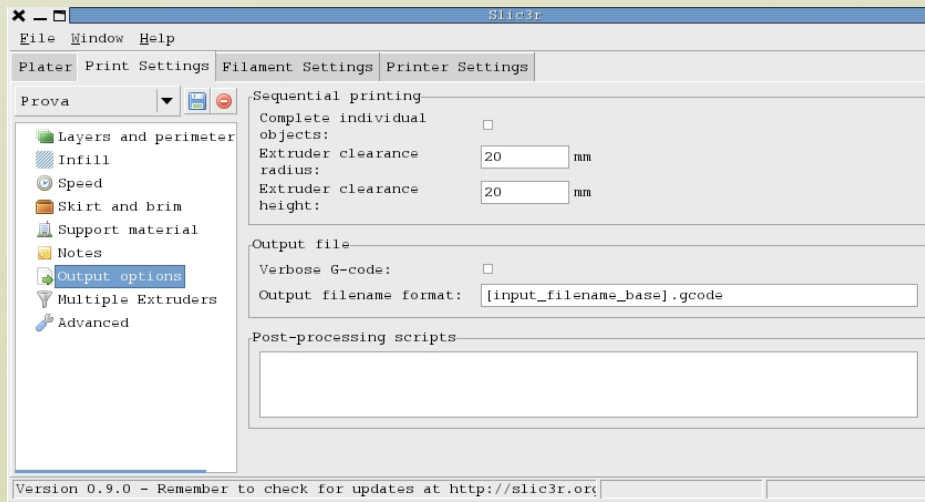
3.7 Note

Here you can add comments about your settings.



3.8 Output options

Here you can decide how to print multiple objects and you can add scripts for G-Code processing.



- **Sequential printing**

- Complete individual object: checking it, when the printer is printing multiple objects or copies, it completes one single object before carrying on the others, checking possible collisions. This option is disabled by default.
- Extruder clearance radius: defines the radius (in mm) within which the extruder won't collide with other objects.
- Extruder clearance height: defines (in mm) the height in order to avoid collisions between the extruder and other already printed objects.

- **Output file**

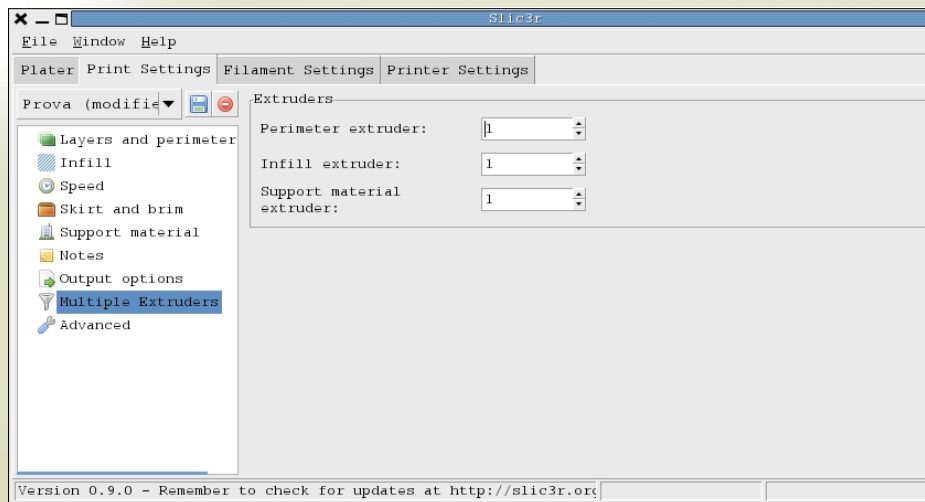
- Verbose G-Code: near a G-Code line will appear an explanation about the performed command.
- Output filename format: here it is possible to rename and change the extension of the file processed by Slic3r.

Post-processing scripts: generated G-Code will be processed by the “script” inserted in this box. ¹

3.9 Multiple Extruders

Here it is possible to configure multiple extruders.

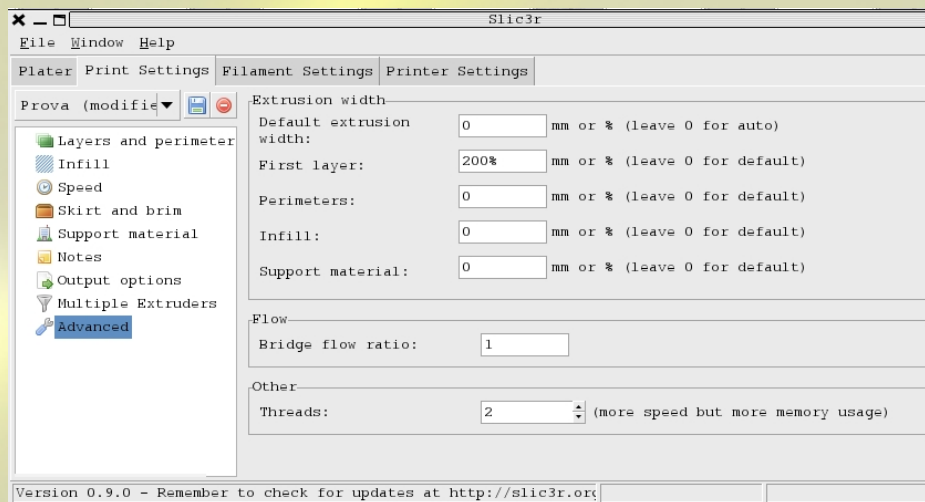
¹For further informationi <http://forums.reprap.org/read.php?263,121863>



- Perimeter extruder: selects the extruder for perimeters.
- Infill extruder: selects the extruder for infill.
- Support material extruder: selects the extruder for support.

3.10 Advanced

In this section it is possible to manually set up some parameters, to fine tune your print by selectively increasing or decreasing extrusion widths, adjusting the extrusion rate for bridging, and increase the slicing speed by use of processing threads (CPU & Memory dependant).



- **Extrusion width.**
 - Default extrusion width: extrusion flow value in mm or %.
 - First layer: first layer extrusion flow value in mm or %.
 - Perimeters: perimeter extrusion flow value in mm or %.
 - Infill: infill extrusion flow value in mm or %.

- **Flow**

- Bridge flow ratio: Bridges' extrusion flow value.

- **Other**

- Threads: sets up the parallelization value of G-Code generation. By increasing this value the processing time decreases with disadvantage for memory usage. If you do not have a particularly powerful computer it is better to set up the default value ².

²wikipedia

Chapter 4

Filament Settings

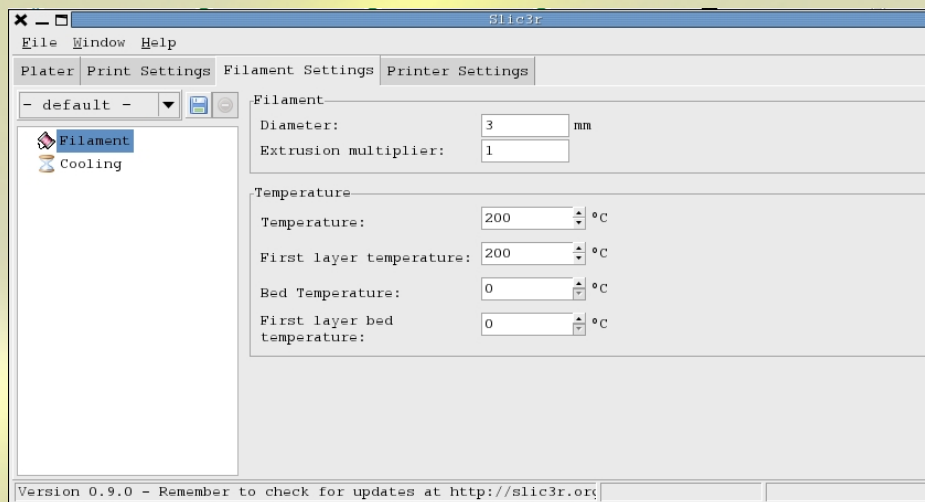
In this section we can find all the settings related to filament dimension, nozzle diameter and cooling management.

4.1 Save and load settings

As *Printing Setting* these settings can be customized and saved.

4.2 Filament

Here we have filament diameter and temperature management.



- **Filament**

- Diameter: sets filament diameter.
- Extrusion multiplier: sets extruder flow multiplication value.

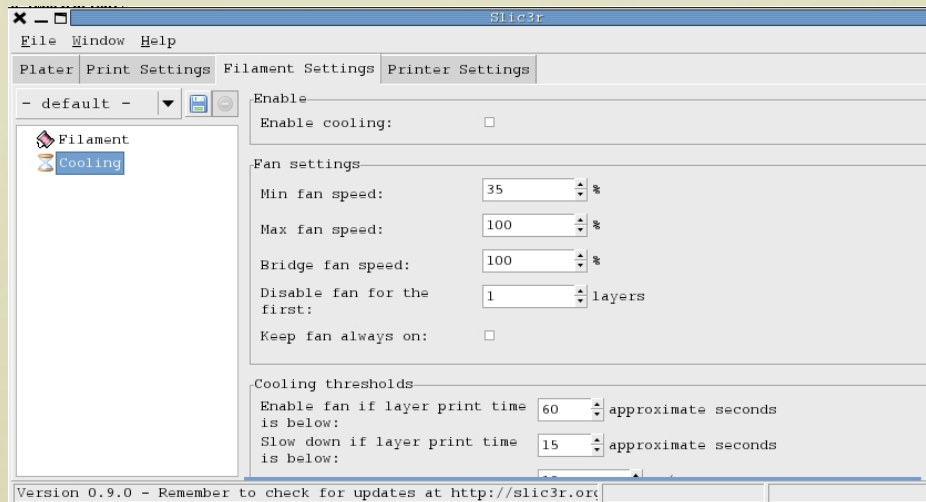
- **Temperature**

- Temperature: sets extruder temperature.
- First layer temperature: sets extruder temperature for the first layer.

- Bed Temperature: sets *HeatedBed* temperature.
- First layer bed temperature: sets *HeatedBed* temperature for the first layer

4.3 Cooling

In this section we can find the cooling system management during the print.

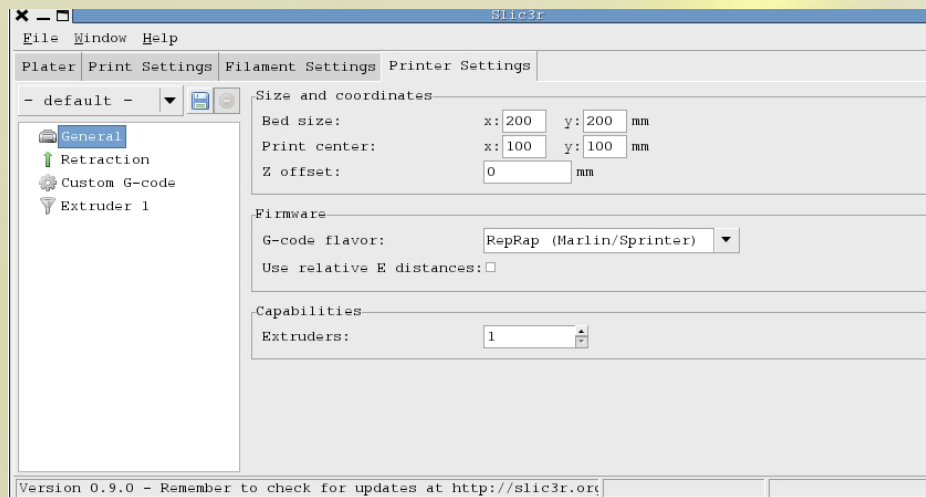


- **Enable:** by checking this box you will enable the cooling system and its management system.
- **Fan setting:**
 - Min fan speed: sets in % the minimum fan speed.
 - Max fan speed: sets in % the maximum fan speed.
 - Bridge fan speed: sets in % the minimum fan speed during *bridge* printing.
 - Disable fan for the first: sets the amount of layers before the fan starts.
 - Keep fan always on: by checking this box the fan will always be enabled.
- **Cooling thresholds:**
 - Enable fan if layer print time is below: enables fan if printing time is less than the previously setted one.
 - Slow down if layer print time is below: disables fan if printing time is less than the previously setted one.
 - Min print speed: sets the minimum printing speed that keeps the fan off.

Chapter 5

Print Settings

In this section we find printing area management, multiple extruders management and G-Code personalization.



5.1 General

Here you can manage the size of the printing area, the number of extruders and the firmware.

- **Size and coordinates**

- Bed size: sets the dimension of the printing area in mm. This settings will affect the graphic area in the *Plater* tab view too.
- Printer center: sets the center of the printing area in mm. The object will be positioned according to this parameter.

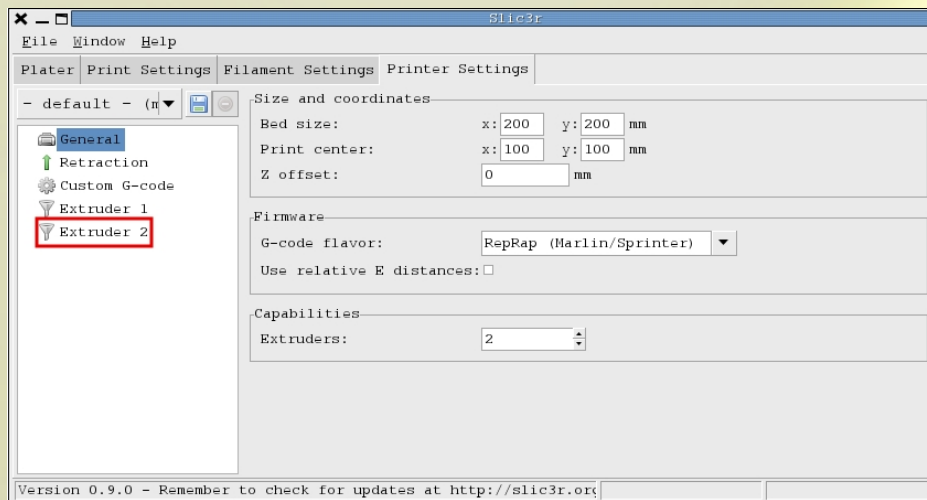
- **Firmware**

- G-code flavor: with this combo box you can select the firmware to use between:
 - * Marlin-Sprinter.
 - * Teacup.
 - * MakerBot.

- * Mach3/EMC.
- * No extrusion
- Use relative E distances: if checked, modifies the extruder movement according to the previous position instead of absolute position.

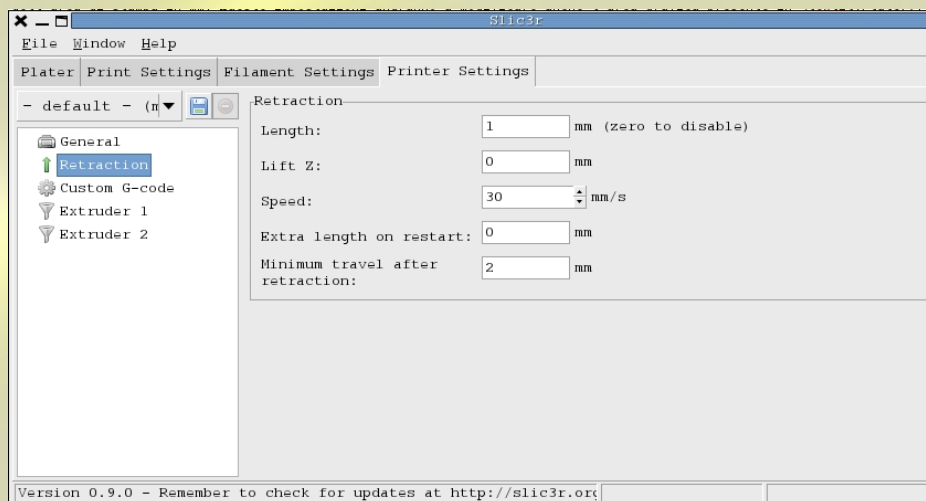
- **Capabilities**

- Extruders: sets the number of extruders. The values in *Printer Settings* will be automatically updated.



5.2 Retraction

The retraction in mm done by the extruder between two printing points. You may have to try a few settings before you find the best setting that guarantees no plastic filament or drop dirt.

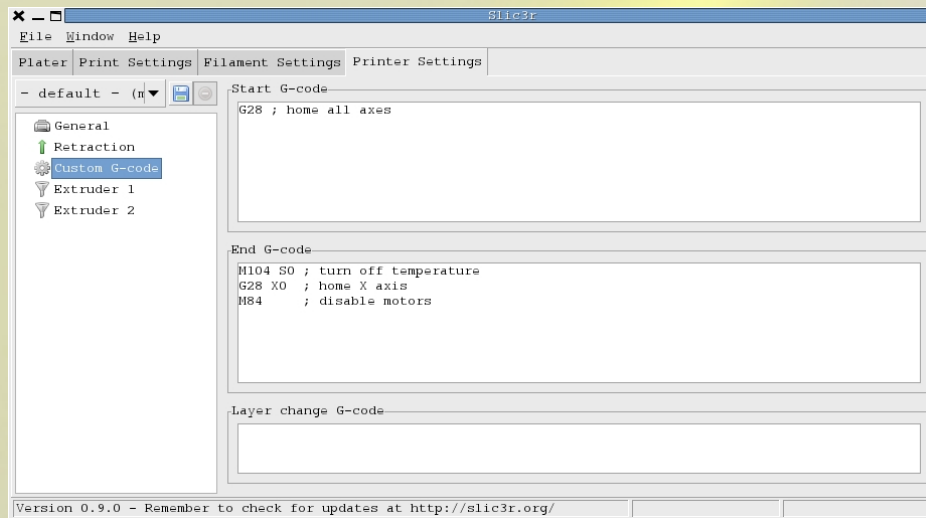


- Length: sets the amount of filament that will be retracted during movements. Depends from hot end and pushing mechanism. It's recommended to use the default value.
- Lift Z: sets the z axis offset used on fast movement in order to not damage already printed material with the hot end.

- Speed: sets the speed of filament retraction. With Marlin, you can use up 30 mm/s value; with Sprinter, 16 mm/s is recommended.
- Extra length on restart: indicates the excess extruded mm while resuming a printing after a retraction affected offset. It is better to keep this value to 0.
- Minimum travel after retraction: it is an important value, it is useful to avoid retraction if the point you're going to print next is near.

5.3 Custom G-Code

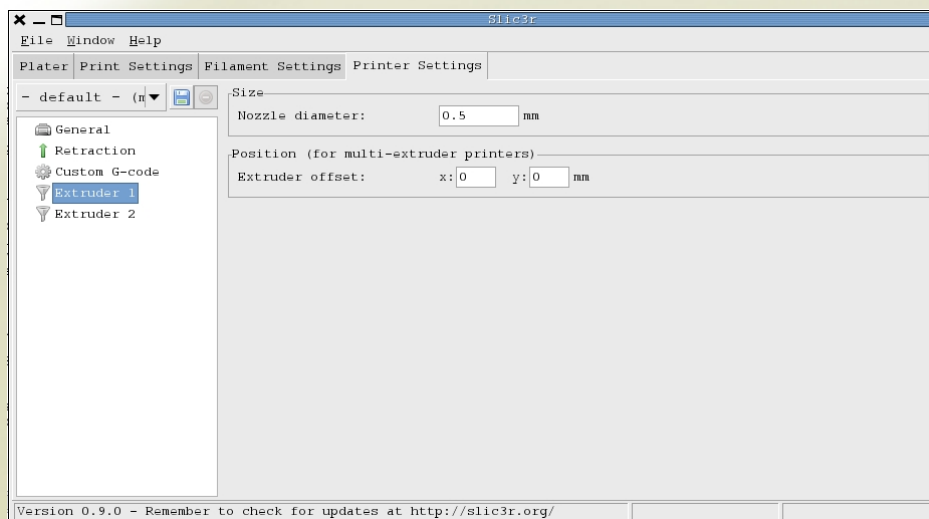
In this section we can manually insert G-Code before or after file body.



- Start G-Code: in this box you can write lines of code that will be executed at the beginning of the file .
- End G-Code: in this box you can write lines of code that will be executed at the end of the file .
- Layer change G-Code: set the execution of a Gcode at each change of layer.

5.4 Extruder

Here we have extruders' settings.



- **Size**

- Nozzle diameter: sets nozzle diameter.
- Extruder offset: sets the position (in mm) of the extruder. It is useful when we use multiple extruders.

Chapter 6

Useful links

- RepRap: <http://www.reprap.org>
- Slic3r: <http://slic3r.org>
- RichRap: <http://richrap.blogspot.it>
- Printron: <http://reprap.org/wiki/Printron>
- Kent's Strapper: <http://kentstrapper.blogspot.it>
- Ivan Bortolin: <http://www.ivanbortolin.it>

Chapter 7

License

This work is licensed under a Creative Commons Attribution-NonCommercial 2.5 License.

