Slic3r User Manual

Various Contributors

Slic3r User Manual

by Sound, et. al.

Published by Free Software Folks

2013217

Contents

1 *Introduction* 7
- 1.1 Overview 8
- 1.2 Goals & Philosophy 8
- 1.3 Donating 8

2 *Getting Slic3r* 9
- 2.1 Downloading 10
- 2.2 Installing 10

3 *First Slice* 13
- 3.1 Calibration 14
- 3.2 Configuration Wizard 15
- 3.3 The Important First Layer 23
- 3.4 Simple Mode 25
- 3.5 Working with Models 34

4 *Configuration Tuning* 35
- 4.1 Infill Choices 36
- 4.2 Speeding Things Up 36
- 4.3 Cooling Things Down 36
- 4.4 Brimming 36
- 4.5 Other Things 36

5 *Configuration Organization* 37
- 5.1 Profiles 38

6 *Advanced Slicing* 39
- 6.1 Support 40
- 6.2 Multipe Extruders 40
- 6.3 Sequential Printing 40
- 6.4 Extrusion Widths 40

6.5 SVG Output 40

7 *Troubleshooting* 41

7.1 Common Issues 42

8 *The Cutting Edge of Slic3r* 43

8.1 Building from Source 44

9 *Slic3r Support* 45

9.1 Community 46

10 *Contact Information* 47

10.1 Support 48

10.2 Chat 48

10.3 Website 48

Index 49

List of Figures

3.1 Configuration Wizard: Welcome Screen 15
3.2 Configuration Wizard: Firmware Type 16
3.3 Configuration Wizard: Bed Size 17
3.4 Configuration Wizard: Nozzle Diameter 18
3.5 Configuration Wizard: Filament Diamter 19
3.6 Configuration Wizard: Extrusion Temperature 20
3.7 Configuration Wizard: Bed Temperature 21
3.8 Configuration Wizard: End 22
3.9 Simple Mode: Print Settings. 26
3.10 An example of insufficient top layers. 27
3.11 The most common infill patterns. 28
3.12 An example of support material. 29
3.13 A diagram depicting the clearance cylinder around an extruder. 30
3.14 Simple Mode: Filament Settings. 30
3.15 Simple Mode: Printer Settings. 32

Introduction

1.1 Overview

Slic3r is a tool which translates digital 3D models into instructions that are understood by a 3D printer. It slices the model into horizontal layers and generates suitable paths to fill them.

Slic3r is already bundled with the many of the most well-known host software packages: Pronterface, Repetier-Host, ReplicatorG, and can be used as a standalone program.

This manual will provide guidance on how to install, configure and utilise Slic3r in order to produce excellent prints.

1.2 Goals & Philosophy

Slic3r is an original project started in 2011 by Alessandro Ranellucci (aka. Sound), who used his considerable knowledge of the Perl language to create a fast and easy to use application. Readability and maintainability of the code are among the design goals.

The program is under constant refinement, from Alessandro and the other contributors to the project, with new features and bug fixes being released on a regular basis.

1.3 Donating

Slic3r started as a one-man job, developed solely by Alessandro in his spare time, and as a freelance developer this has a direct cost for him. By generously releasing Slic3r to the public as open source software, under the GPL license, he has enabled many to benefit from his work.

The opportunity to say thank you via a donation exists. More details can be found at: `http://slic3r.org/donations`.

Getting Slic3r

2.1 Downloading

Slic3r can be downloaded directly from: `http://slic3r.org/download`.

Pre-compiled packages are available for Windows, Mac OS X and Linux. Windows and Linux users can choose between 32 and 64 bit versions to match their system.

Source

The source code is available via github: `https://github.com/alexrj/Slic3r` (For more details see §8.1)

Manual

The latest version of this document, with LaTeX source code is presently at: `http://devel.lulzbot.com/Slic3r/`

2.2 Installing

Windows

Unzip the downloaded zip file to a folder of your choosing, there is no installer script. The resulting folder contains two executables:

- `slic3r.exe` - starts the GUI version.
- `slic3r-console.exe` - can be used from the command line.

The zip file may then be deleted.

Mac OS X

Double-click the downloaded dmg file, an instance of Finder should open together with an icon of the Slic3r program. Navigate to the Applications directory and drag and drop the Slic3r icon into it. The dmg file may then be deleted.

Linux

Extract the archive to a folder of your choosing. Either:

- Start Slic3r directly by running the Slic3r executable, found in the bin directory, or
- Install Slic3r by running the do-install executable, also found in the bin folder.

The archive file may then be deleted.

First Slice

3.1 Calibration

Before even attempting the first print it is vital that the printer is correctly calibrated. Skipping or rushing this step will result in frustration and failed prints later, so it is important to take the time to make sure the machine is correctly set up.

Each machine may have it's own calibration procedure and this manual will not attempt to cover all the variations. Instead here is a list of key points that should be addressed.

- Frame is stable and correctly aligned.
- Belts are taut.
- Bed is level in relation to the path of the extruder.
- Filament rolls freely from the spool, without causing too much tension on the extruder.
- Current for stepper motors is set to the correct level.
- Firmware settings are correct including: axis movement speeds and acceleration; temperature control; end-stops; motor directions.
- Extruder is calibrated in the firmware with the correct steps per mm of filament.

The point regarding the extruder step rate, is vital. Slic3r expects that the machine will accurately produce a set amount of filament when told to do so. Too much will result in blobs and other imperfections in the print. Too little will result in gaps and poor inter-layer adhesion.

Please refer to the printer documentation and/or resources in the 3D printing community for details on how best to calibrate a particular machine.

3.2 Configuration Wizard

Slic3r has two features to aid newcomers: the configuration wizard, and simple mode.

Sometimes it is nice to have a helping hand when starting out with new software. The configuration wizard asks a series of questions and creates a configuration for Slic3r to start with.

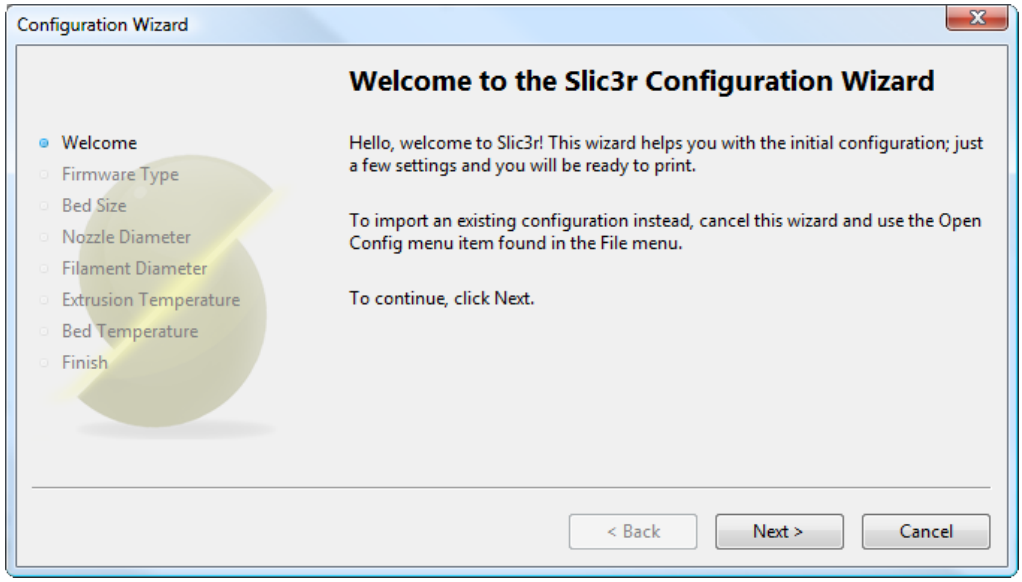


Figure 3.1: Configuration Wizard: Welcome Screen

1. Firmware Type

The gcode produced by Slic3r is tailored to particular types of firmware. The first step prompts for the firmware that the printer uses. This should have been specified when the printer was built or configured. If unsure then contact the supplier.

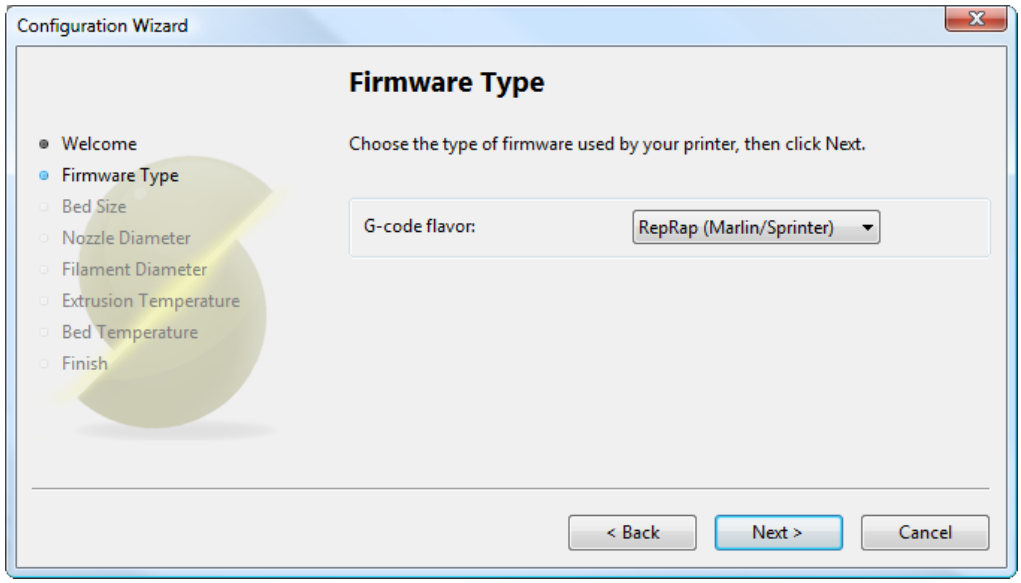


Figure 3.2: Configuration Wizard: Firmware Type

2. Bed Size

This setting defines the maximum distance the extruder may travel along the X and Y axis. If the dimensions are not readily available for the printer then it can be easily measured.

Be sure to measure from the lower left corner where the extruder nozzle rests when are the home position to the maximum distance the nozzle can travel in each direction. Take into account that the X carriage may touch the frame before the nozzle reaches it's full distance, this will depend on the printer make and model.

Also remember to check any firmware end-stop settings which may limit X/Y movement.

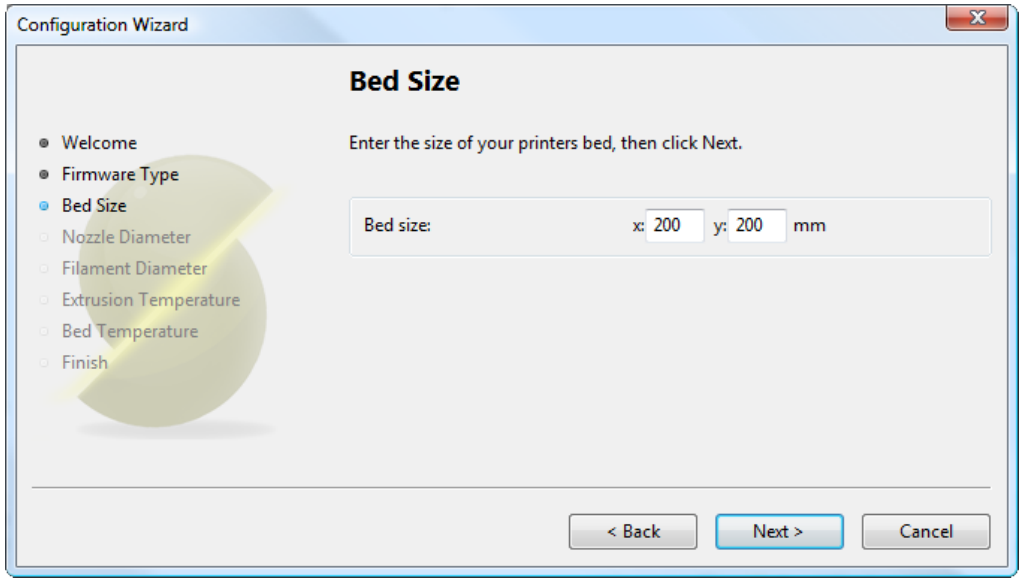


Figure 3.3: Configuration Wizard: Bed Size

3. Nozzle Diameter

The diameter of the hot-end nozzle is usually clearly displayed either in the description of the hot-end, or in the associated documentation, when the hot-end is purchased. Common values are 0.5mm and 0.35mm.

If the nozzle was home-made, or came from a source without a diameter given, then carefully measure the aperture as accurately as possible. One way of determining nozzle size is to very slowly (1mm/s) extrude some filament into free air and measure the thickness of the resulting extrusion[1]. This has the benefit of taking die swell into account, and consequently may be a useful thing to do even if the diameter is known.

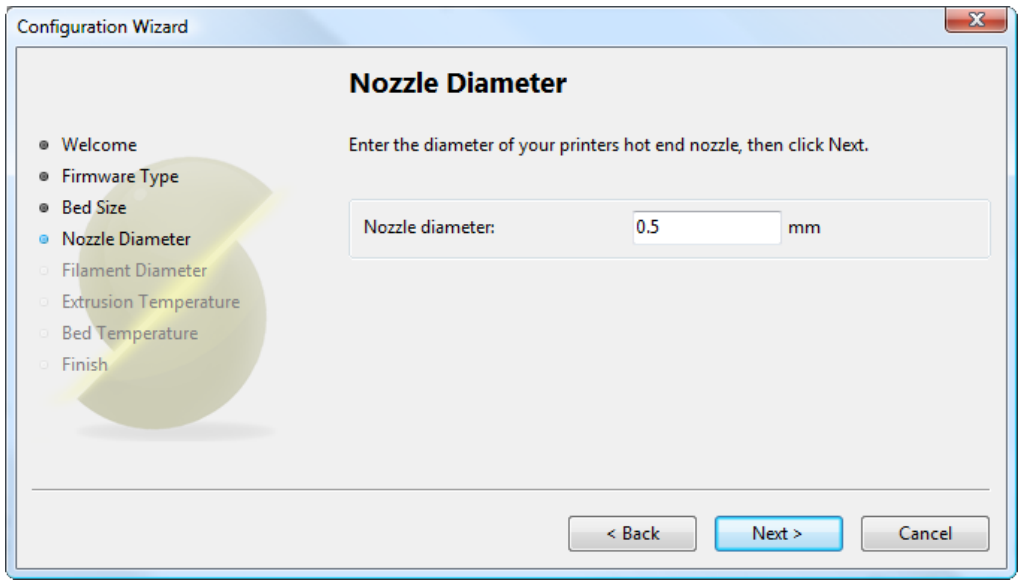


Figure 3.4: Configuration Wizard: Nozzle Diameter

[1] `http://forums.reprap.org/read.php?1,113374,113953`

4. Filament Diameter

For Slic3r to produce accurate results it must know as accurately as possible how much material is pushed through the extruder. Therefore it is vital to give it as precise a value as possible for the filament diameter.

Although the filament used in FDM printers is sold as being either 3mm or 1.75mm this is only a general guide. The diameter can vary between manufacturers and even between batches. Therefore it is highly recommended to take multiple measurements from along a length of the filament and use the average. For example, measurements of 2.89, 2.88, 2.90 and 2.91 would yield an average of 2.895, and so this would be used.

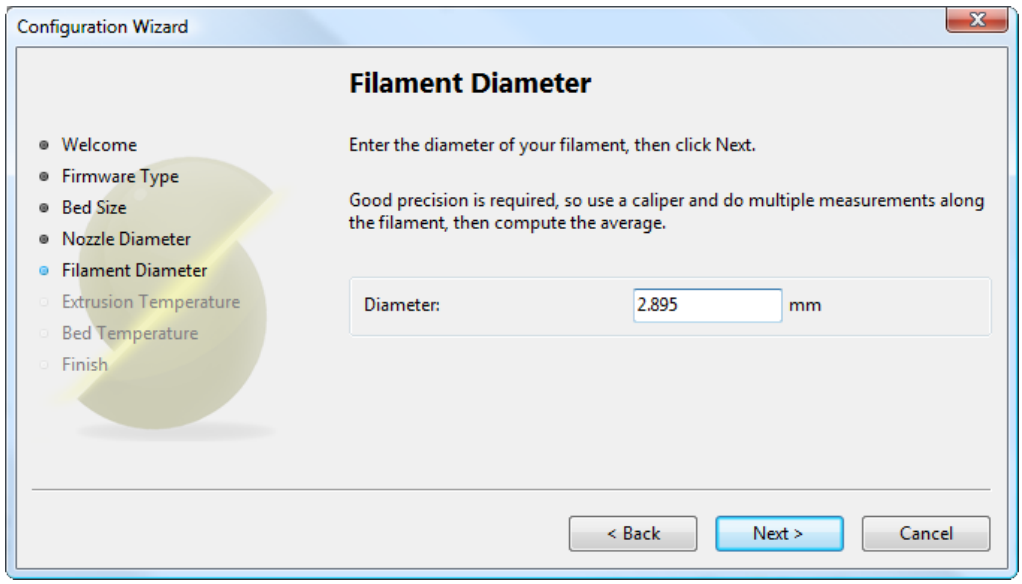


Figure 3.5: Configuration Wizard: Filament Diamter

5. Extrusion Temperature

The extrusion temperature will depend on the material, and most can operate over a range of temperatures. The supplier should provide guidance as to which temperatures are suitable. A very general rule of thumb is that PLA lies between 160°C and 230°C, and ABS lies between 215°C and 250°C. More exotic materials will have a different range.

This is one parameter which you will want to fine tune when you start producing prints. The optimal temperature can vary even between colours of the same material. Another factor which may affect the chosen temperature is how fast the extrusion is, where generally faster extrusion runs hotter.

Note: One may choose to control the extruder temperature manually from the printer controller. In this case the temperature can be set to zero.

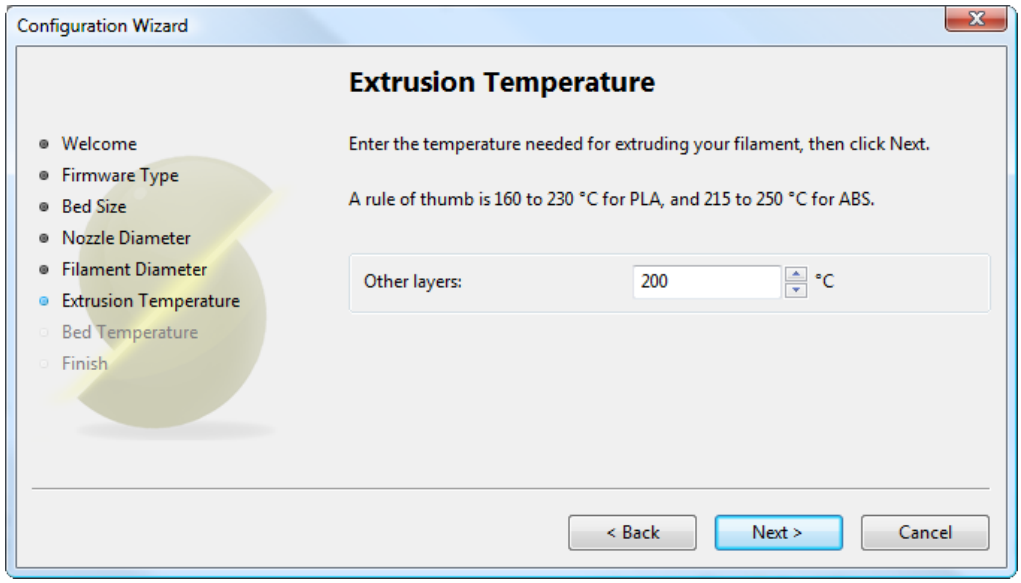


Figure 3.6: Configuration Wizard: Extrusion Temperature

6. Bed Temperature

If the printer has a heated bed then this parameter may be set. As with the extruder temperature, the value will depend on the material used. A rule of thumb is that PLA requires 60°C and ABS requires 110°C.

Note: One may choose to control the bed temperature manually from the printer controller. In this case the temperature can be set to zero.

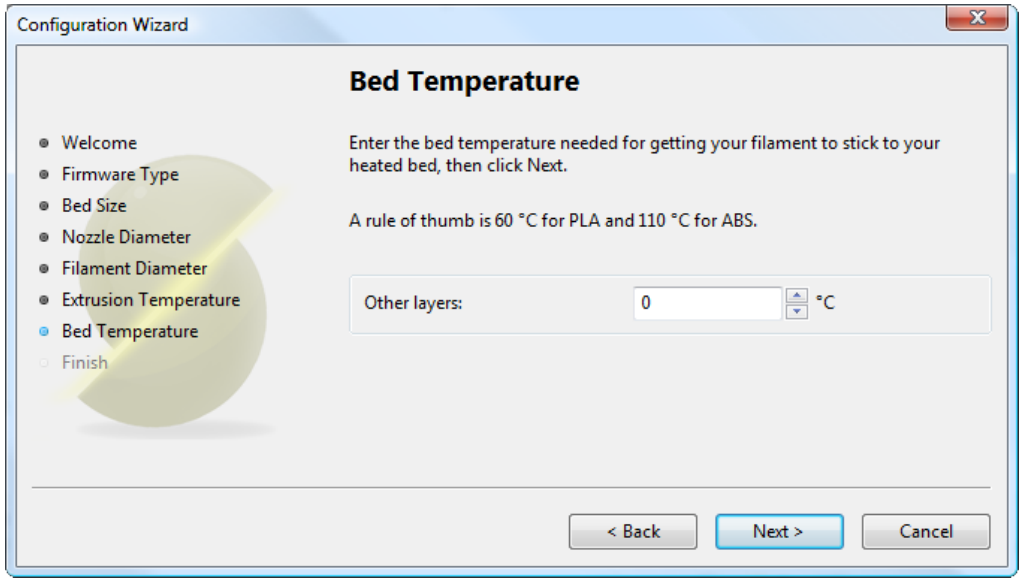


Figure 3.7: Configuration Wizard: Bed Temperature

At this stage the wizard is complete and the basic configuration is defined.

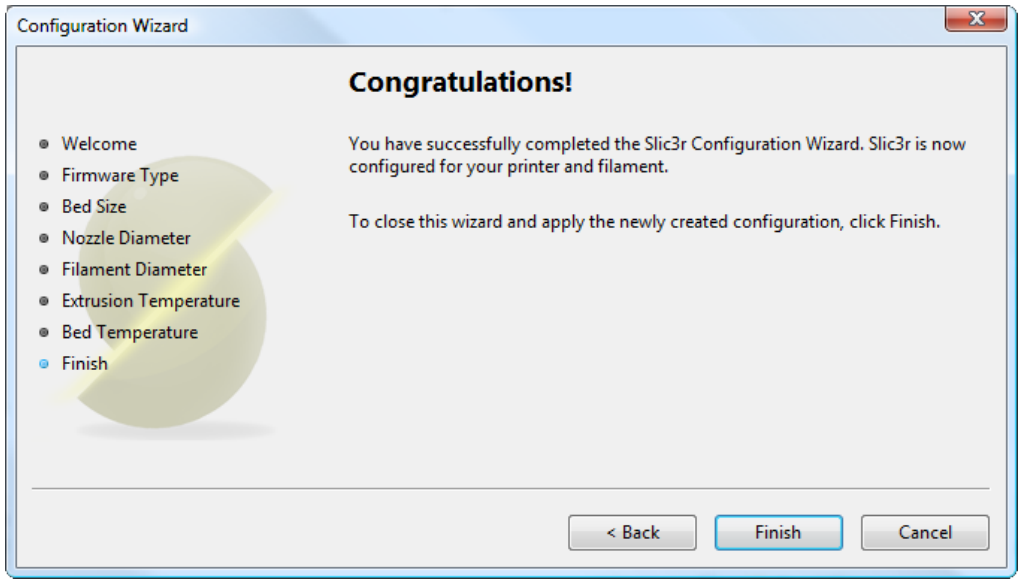


Figure 3.8: Configuration Wizard: End

3.3 The Important First Layer

Before delving into producing the first print it is worthwhile taking a little detour to talk about the importance of getting the first layer right. As many have found through trial and error, if the first layer is not the best it can be then it can lead to complete failure, parts detaching, and warping. There are several techniques and recommendations one can heed in order to minimise the chance of this happening.

Level bed. Having a level bed is critical. If the distance between the nozzle tip and the bed deviates by even a small amount it can result in either the material not lying down on the bed (because the nozzle is too close and scrapes the bed instead), or the material lying too high from the bed and not adhering correctly.

Higher temperature. The extruder hot-end and bed, if it is heated, can be made hotter for the first layer, thus increasing the viscosity of the material being printed.

No cooling. Directly related with the above, it makes no sense to increase the temperature of the first layer and still have a fan or other cooling mechanism at work. Keeping the fan turned off for the first few layers is generally recommended. Of course, some models may need direct cooling due to their size, but this would be an exception.

Lower speeds. Slowing down the extruder for the first layer reduces the forces applied to the molten material as it emerges, reducing the chances of it being stretched too much and not adhering correctly.

Correctly calibrated extrusion rates. If too much material is laid down then the nozzle may drag through it on the second pass, causing it to lift off the bed (particularly if the material has cooled). Too little material may result in the first layer coming loose later in the print, leading either to detached objects or warping. For these reasons it is important to have a well-calibrated extrusion rate as recommended in §3.1).

Wider extrusion width. The more material touching the bed, the more adhesion it will have. There are several ways to achieve this:

- Reduce the height of the first layer, either by a percentage or a fixed amount. A value of approximately 60% is usually recommended.
- Increase the extrusion width of the first layer, either by a percentage or a fixed amount. A value of approximately 200% is usually recommended.

Note: These options are available in the advanced mode.

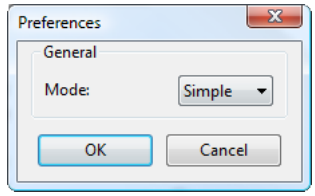
Bed material. Many options exist for the material to use for the bed, and preparing the right surface can vastly improve first layer adhesion.

PLA is more forgiving and works well on PET, Kapton, or blue painters tape.

ABS usually needs more cajoling and, whilst it can print well on PET and Kapton, there are reports that people have success by applying hairspray to the bed before printing. Others have reported that an ABS slurry (made from dissolving some ABS in Acetone) thinly applied can also help keep the print attached.

3.4 Simple Mode

Slic3r has two modes of operation, Simple and Advanced. These may be chosen from the `Preferences` window (found under the `File` menu).



As is expected, the simple mode offers a cut-down set of options, enough for the beginner to get started with. The advanced options give more control over how Slic3r produces the gcode and will be looked at later.

Printer Settings

The `Print Settings` tab provides the opportunity to change settings related to the actual print. Whereas the other tabs are changed rarely, the settings on this tab will be modified regularly, possibly for each model printed.

Layer height. There are several factors that influence how high each layer should be:

- **Desired resolution** - Lower layer height should result in prints with less noticeable ribs or bands, as each layer is smaller. Aesthetics plays a role here, but also the type of model, for example, a mechanical part may not need such a high resolution finish, whereas a presentation piece may do so.
- **Print speed** - Shorter layers will result in smoother prints but each print will take longer, simply because the extruder must trace the pattern more times. A later goal will be to strike a balance between layer height, the speed of the printer, and the quality of the resulting print.

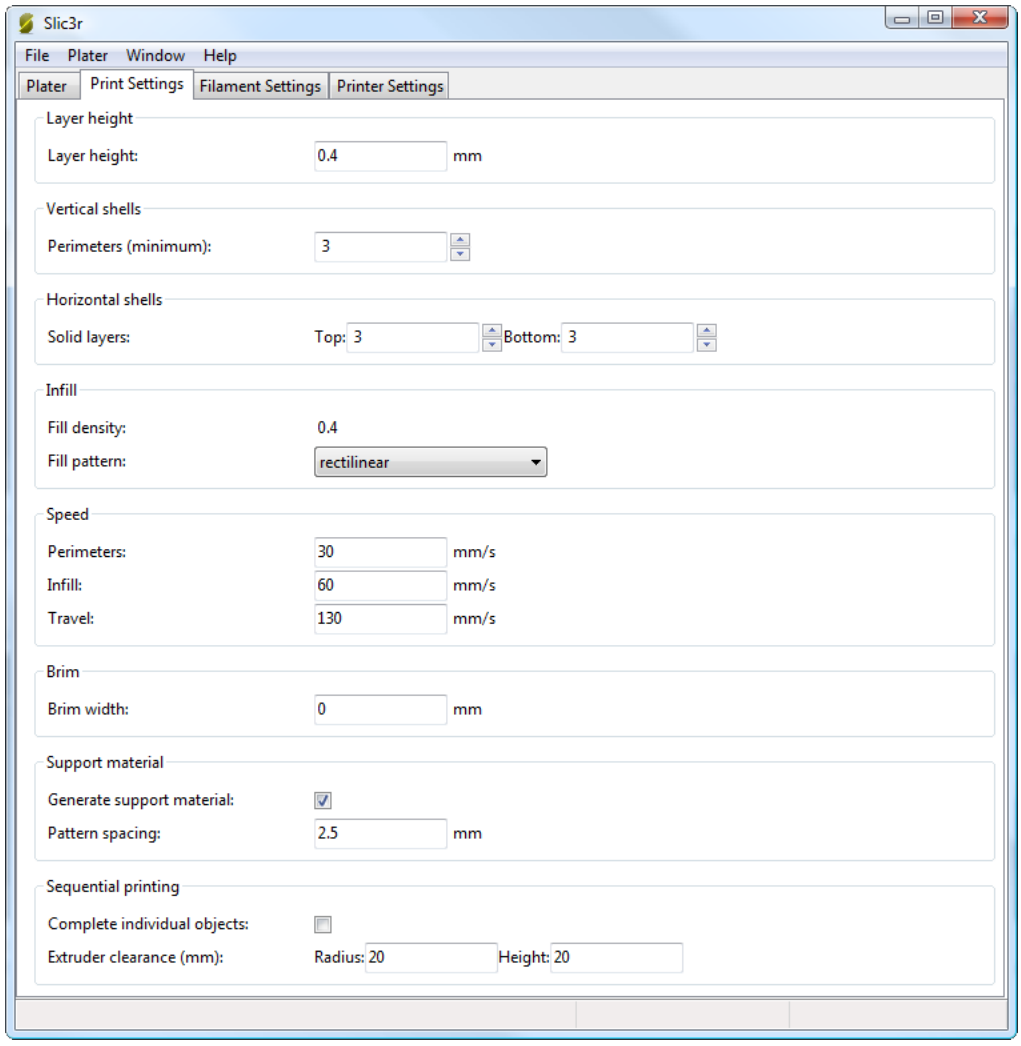


Figure 3.9: Simple Mode: Print Settings.

Vertical shells. Defines the number of perimeters a print will have. Unless the model requires single width walls it is generally recommended to have a minimum of two perimeters as this gives some insurance that if a section of the perimeter is not printed correctly then the second perimeter will help cover it.

Horizontal shells. The upper and lowermost layers that sandwich the model are filled with a solid pattern. For the bottom layers the important factor to consider is how the surface will look should there be a mistake whilst laying down the first layer, and for this reason it is recommended to have at least two bottom layers. Of course, once the printer is reliably producing excellent results this can be reduced, if so desired.

A similar consideration is required for the top layers. Because the intermediate layers are likely to be filled with a pattern set less than 100% then the covering layers will have to bridge this pattern and this can require more than one pass to cover completely.

Figure 3.10: An example of insufficient top layers.

Infill. For the majority of cases it makes no sense to 100% fill the model with plastic, this would be a waste of material and take a long time. Instead, most models can be filled with less material which is then sandwiched between layers filled at 100% (see `Horizontal Shells` above).

Slic3r offers several fill patterns, and these will be discussed in more depth later. Choosing a pattern will depend on the kind of model and personal taste. The more exotic fill methods are usually too slow and unnecessarily complex for most use cases, and so most of the time the infill pattern is either `rectilinear`, `line`, or `honeycomb`.

Figure 3.11: The most common infill patterns.

Speed. In simple mode there are only three speed settings to consider:

- **Perimeters** - The outline of the model may benefit from being printed slightly slower so that the outside s̈kinöf the print has fewer blemishes.
- **Infill** - As the infill is hidden this can be extruded a little faster. Take care though not to go too fast as higher speeds results in thinner extrusions, and this may affect how the extrusions bond.
- **Travel** - The jump between the end of one extrusion and the next should usually be performed as quickly as the printer will allow in order to minimise any mess caused by material oozing from the nozzle.

Brim. A relatively new parameter, `Brim` is used to add more perimeters to the first layer in order to provide more surface area for the print to stick (see §3.3). The brim is then removed once the print is finished and removed from the bed.

Support material. Printing a model from the bottom up, as with FDM, means that any significant overhangs will be printed in the air, and most likely droop or not print at all. Choosing support material will add additional structures around the model which will build up to then support the overhanging part. The `Pattern spacing` option determines how dense the support material is printed.

Figure 3.12: An example of support material.

Tip: It is sometimes worth considering altering the orientation of the model in order to possibly reduce overhangs.

Sequential printing. When printing several objects at once it can be useful to print each one separately as this will minimise oozing and strings running between the prints. Care has to be taken that the nozzle and extruder does not interfere with already printed parts. This is the reason for the `Extruder clearance` parameters:

- **Radius** - The clearance that should be given around the extruder. Take care if the extruder is not mounted centrally - take the largest safe value.
- **Height** - The vertical distance between the nozzle tip and the X axis rods, or lowest part which may interfere with a finished print.

Figure 3.13: A diagram depicting the clearance cylinder around an extruder.

Filament Settings

The `Filament Settings` will normally be used infrequently, for example on receipt of a new roll of filament.

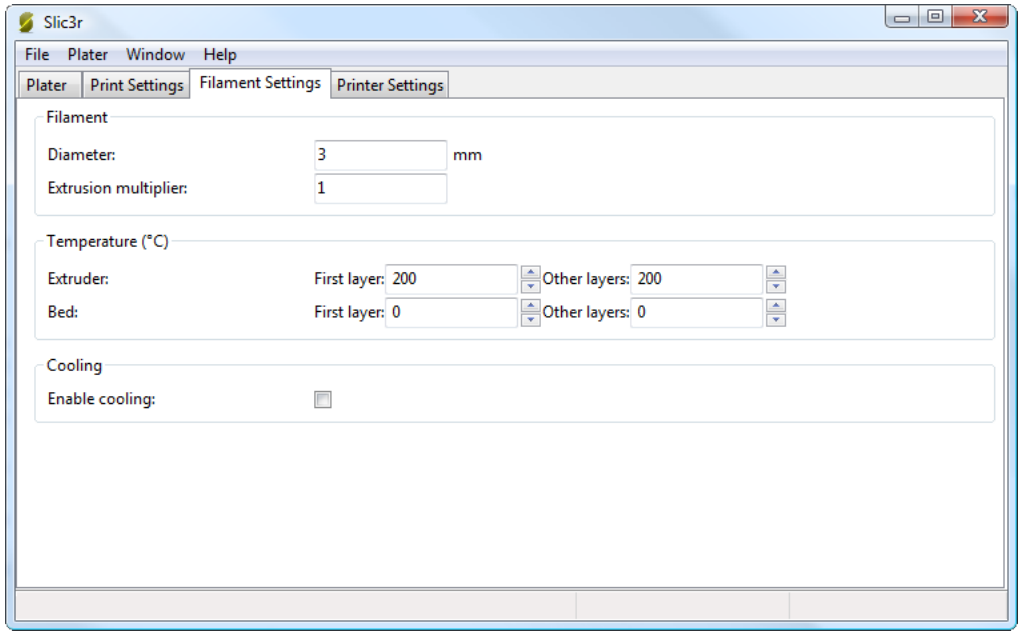


Figure 3.14: Simple Mode: Filament Settings.

Filament. The `diameter` setting will already have been filled from the value given during the wizard (see p.19), but can be updated here.

The `Extrusion multiplier` setting allows the fine tuning of the extrusion rate. Whilst the value should ideally be set in the firmware it can be useful to test slight changes to the rate by altering this value.

Temperature. These values are also filled from the wizard, but here the opportunity exists to set the temperature for the first layer (see p.23).

Cooling. The simple mode offers only one option for cooling: on or off. The program will decide when to turn on the fan and when to slow down as necessary.

Printer Settings

The `Printer Settings` will be updated the least, unless Slic3r is going to be used for many printers, for example, in a 3D printer farm.

Size and coordinates. The `Bed size` setting is taken from the wizard (see p.17), and the `Print center` is simply the mid-point of these values. `Z offset` can be used to compensate for an incorrectly calibrated Z end-stop. If the nozzle stops slightly too far from the bed, then adding a negative value will offset all layers by that amount. The real solution however is to fix the end-stop itself.

Firmware. As selected in the wizard (see p.16), this defines the dialect of gcode generated.

Extruder. As selected in the wizard (see p.18), this defines the nozzle diameter.

Retraction. Unless the material being extruded has a very high viscosity it will ooze between extrusions due to gravity. This can be remedied against by reducing the time between extrusions, and by actively retracting the filament between extrusions. Setting the `Length` parameter to a positive value will cause the filament to be reversed by that many millimeters during travel.

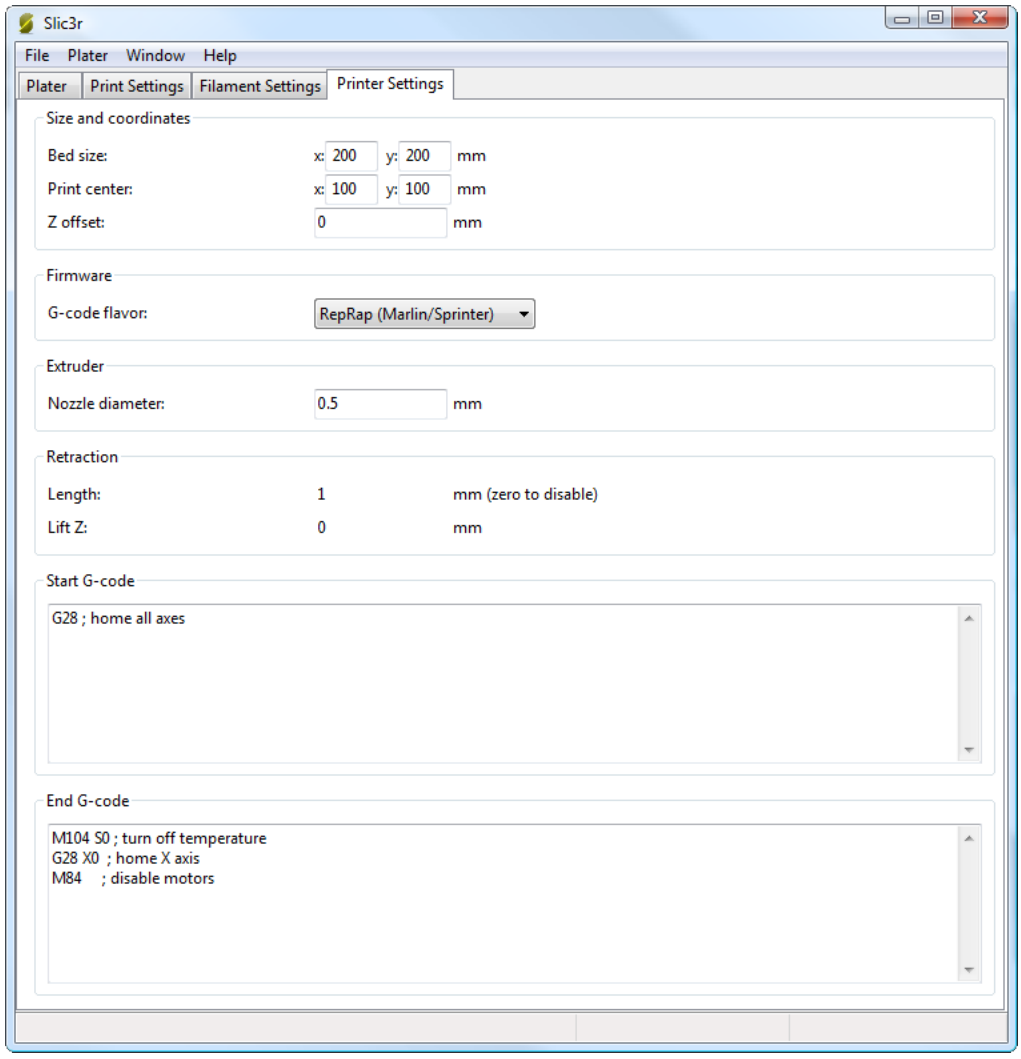


Figure 3.15: Simple Mode: Printer Settings.

Setting the `Lift Z` parameter to a positive value will raise the entire extruder on the Z axis by that many millimeters during each travel. This can be useful to ensure the nozzle will not catch on any already laid filament.

Start G-code. Custom gcode commands that run before the print run starts. Some common gcodes to use are:

- **G28** - Homes all the axes.

The RepRap wiki is a good resource to learn about the variety of gcodes available: `http://reprap.org/wiki/G-code`.

Note: Be sure to check that a given gcode is valid for your firmware.

End G-code. Custom gcode commands that run after the print run ends. Some common gcodes to use are:

- **M104 S0** - Sets the extruder temperature to zero.
- **M84** - Disables the motors.

3.5 Working with Models

Working with models. Not that kind!

Cleaning STLs

Use MeshLab or FreeCAD to clean up your model.

Working with Plater

You can put multiple objects on the plate with Plater.

Configuration Tuning

Configuration Tuning is the best.

4.1 Infill Choices

- line
- rectilinear
- concentric
- honeycomb
- hilbertcurve
- archimedeanchords
- octagramspiral

4.2 Speeding Things Up

Run at 250mm/s.

4.3 Cooling Things Down

Get a fan or go slow.

4.4 Brimming

Use a brim and it'll really stick. Hairspray is better though.

4.5 Other Things

To be covered:

Configuration Organization

Configuration organization–get your stuff in order.

5.1 Profiles

Creating Profiles

Save as...

Exporting and Importing Profiles

Load Config...

Export Config...

Advanced Slicing

Advanced slicing, for pros.

6.1 Support

Put something on the other.

6.2 Multipe Extruders

Print with multiple extruders for multiple color or material prints.

6.3 Sequential Printing

Print one thing after another.

6.4 Extrusion Widths

Make that extrusion fat or skinny...

6.5 SVG Output

Output to SVG for importing into Inkscape and other vector applications.

Troubleshooting

Troubleshooting

7.1 Common Issues

Clean up that STL.

The Cutting Edge of Slic3r

8.1 Building from Source

Follow these steps to build from source.

```
$ git clone git://github.com/alexrj/Slic3r

$ cd Slic3r

$ sudo perl Build.PL

$ sudo cpan Wx
```

Slic3r Support

9.1 Community

Community Support and Resources

- IRC chat rooms on the `irc.freenode.net` server.
  - `#reprap`: Highly active community chat room where help can easily be found
  - `#slic3r`: Slic3r chat room where Slic3r developers and users can give help
- RepRap.org forums: `forums.reprap.org`

Contact Information

10.1 Support

10.2 Chat

#slic3r on irc.freenode.net

10.3 Website

`www.slic3r.org`

Index

A
archimedeanchords, 36

B
binaries, 10
brim, 36

C
calibration, 14
community support, 46
concentric, 36
configuration, 38
cooling, 36

D
download, 10

E
export config, 38
extruders, 40
extrusion, 40

F
forums, 46
Freenode, 46

G
github, 10

H
hilbertcurve, 36
honeycomb, 36

I
infill, 36
IRC, 46

L
latex, 10
line, 36
load config, 38

O
octagramspiral, 36

P
profiles, 38

R
rectilinear, 36
RepRap, 46

S
sequential, 40
source, 10
source code, 10
speed, 36
support, 40
SVG, 40

W
width, 40

Colophon

Created with 100% Free/Libre Software

GNU/Linux

LaTeX Memoir
