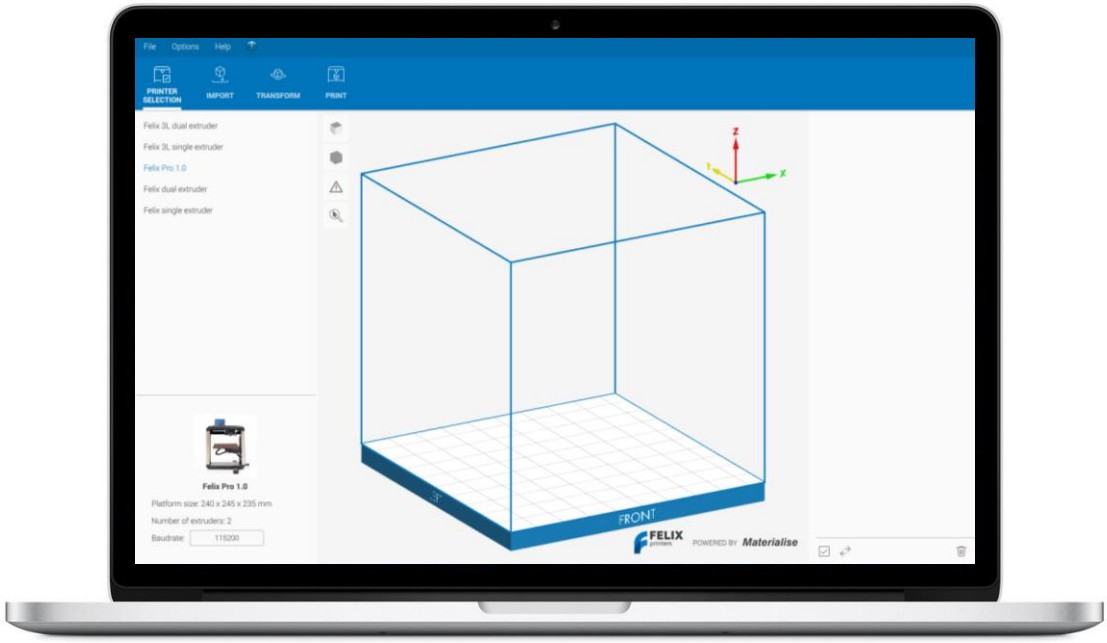
FELIXbuilder User Manual

FELIXbuilder 2.1

9 August 2016

Revision 1



FELIX printers

1 Table of Contents

2 Welcome to Felix Builder

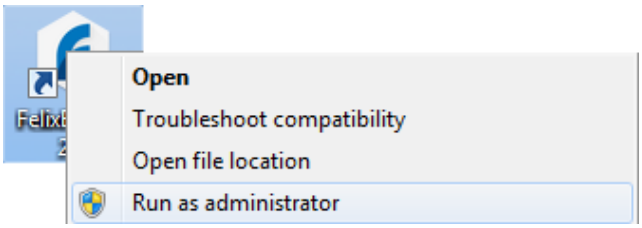
2.1 Install and registration

2.1.1 Application install windows

The preferred software for your FELIX 3D printer is FELIXbuilder from Materialise. This software package prepares your 3D model into machine instructions for your 3D printer. Besides sold separately FELIXbuilder is also bundled together with your 3D printer. So together with your 3D printer, you also received a voucher code which is valid for 1 year. You should use this voucher code to register the software.

1. Open the installer (.msi) and follow the wizard to install the Felix Builder software
2. Open the Felix Builder hyperlink in the start menu
3. Approve the end-user license agreement.
4. Enter the voucher code that you have received with the Felix 3D printer and press validate.
5. The software will be registered and you will get notified of the validity period of the voucher.

Note: if the voucher registration isn't working please run the application as administrator.

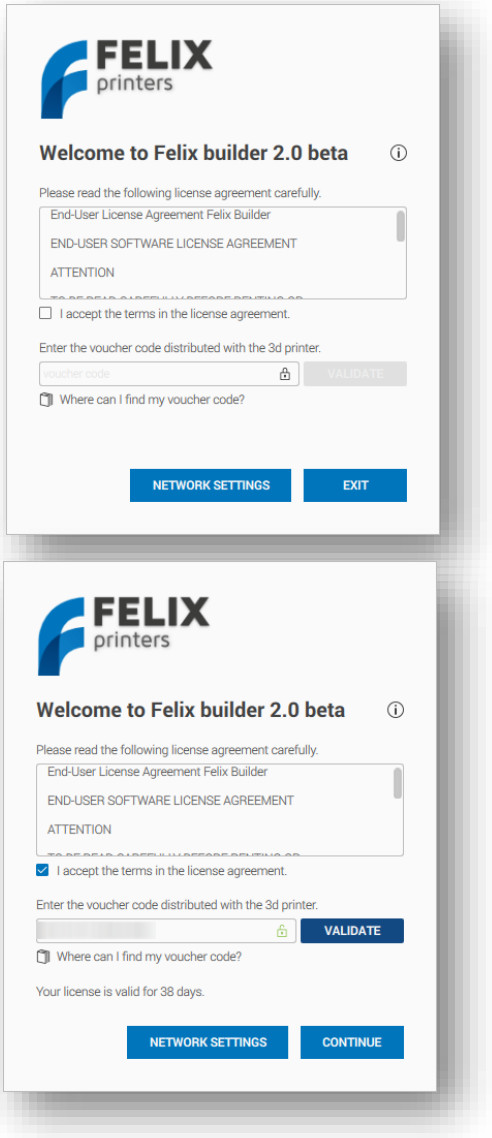


2.1.2 Application install mac

The preferred software for your Felix 3D printer is Felix Builder from Materialise. This software package prepares your 3D model into machine instructions for your 3D printer. Felix Builder is not sold separately but bundled together with your 3D printer. So together with your 3D printer, you also received a voucher code. You should use this voucher code to register the software.

1. Open the installer (.dmg) and drag and drop the application to your Applications folder.
2. After installing Felix Builder, right click the application and press open.
3. Click Open in the dialog box. If prompted, enter an administrator name and password (not able to open the application? See note)

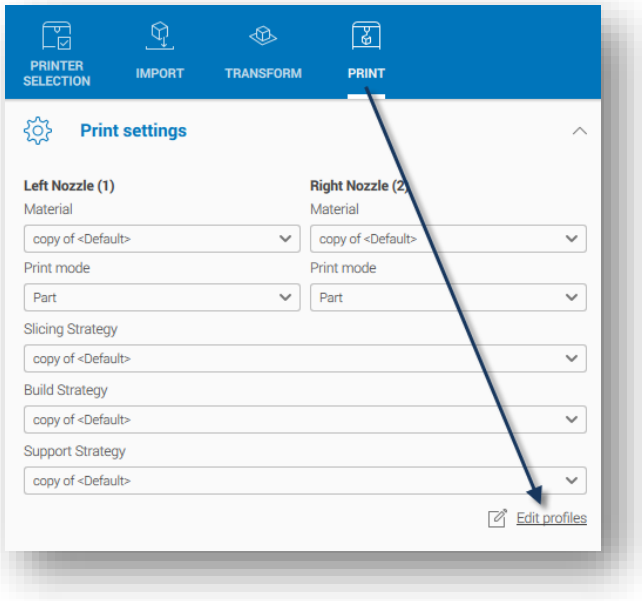
4. Approve the end-user license agreement.
5. Enter the voucher code that you have received with the Felix 3D printer and press validate.
6. The software will be registered and you will get notified of the validity period of the voucher.
7. Press **Finish** to complete the installation.



Note: for more info visit: https://support.apple.com/en-gb/HT202491

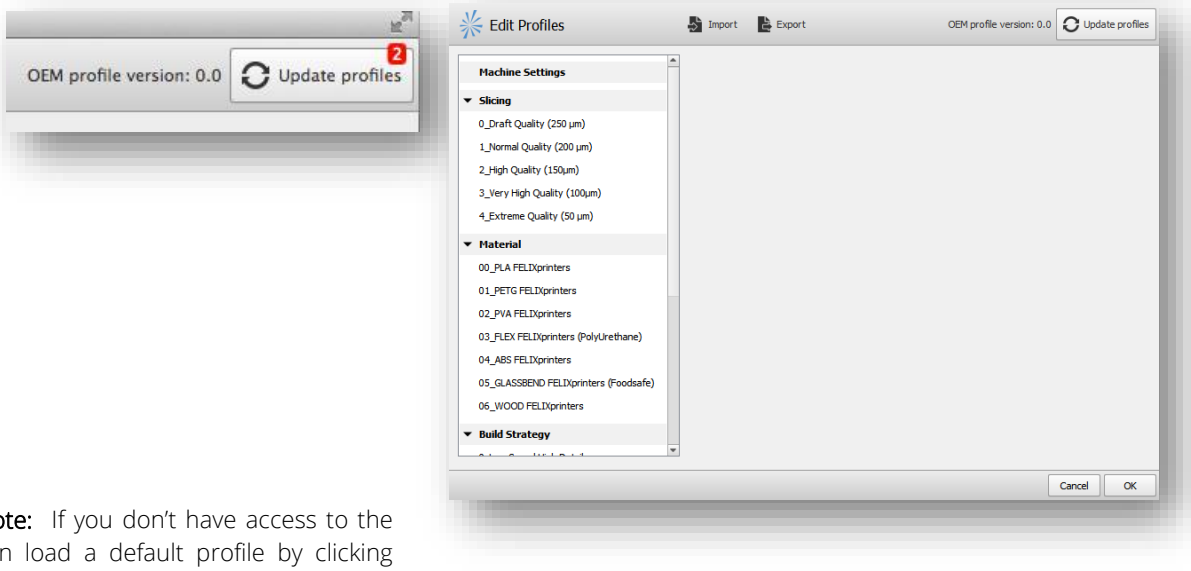
2.1.3 Load Felix build profiles

1. Navigate to Print → Edit profiles

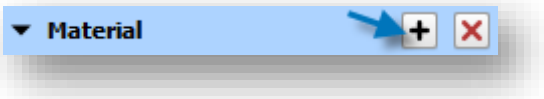


2. Profile editor window opens
3. Press **Update profiles** to get the latest Felix profiles (internet connection is needed)

4. The latest Felix profiles are loaded in the software



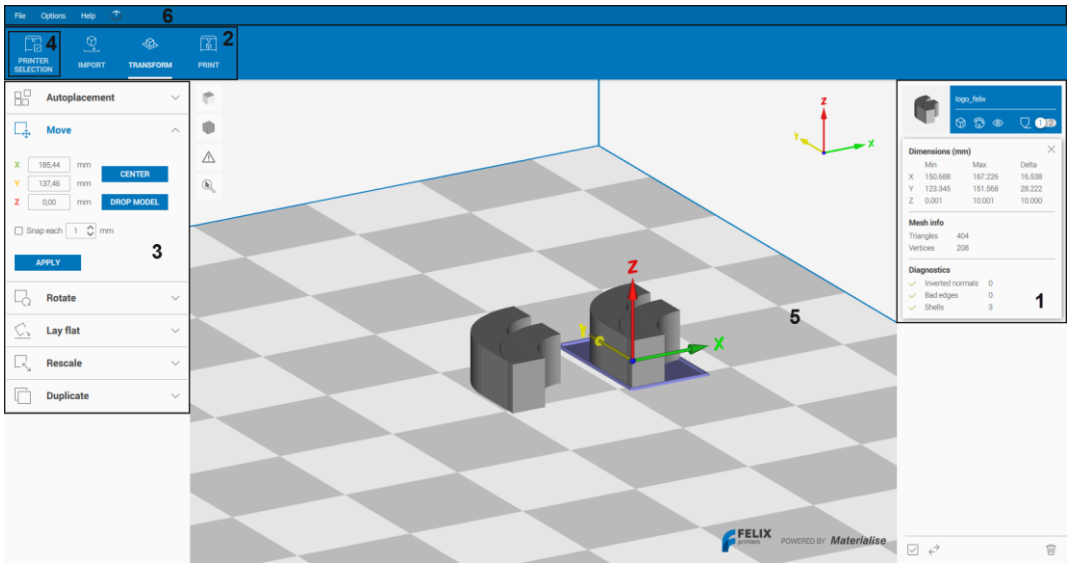
Note: If you don't have access to the internet you can load a default profile by clicking here:



2.2 Basic Interface

1. Part List
2. Flow toolbar
3. Function-toolbar
4. Printer selection
5. Workspace
6. Main toolbar

When you launch Felix Builder, you will directly see a platform from the selected printer.



2.2.1 The flow

A. Click the **Import** flow button to load your STL-files
B. Click the **Transform** flow button if you want to translate, rotate, rescale or duplicate your part(s)
C. Click the **Print** button if you want to build your part(s) on your 3D printer

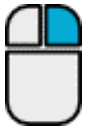
2.2.2 Navigate the workspace

Leftmouse button

Select objects.

Scrollwheel button

Use the scroll wheel to zoom in or out.



Rightmouse button

Hold and move the mouse to rotate the viewpoint around the 3D model.

Rightmouse button + shift or middle mouse button

Hold and move the mouse to pan the 3D view.

3 Main Toolbar

3.1 File

3.1.1 New Project

This function will remove all parts and scenes from the current project and generates a new, empty project. The user is asked to save the current project (parts and scenes) before closing. If you want to work in two (or more) parallel Felix Builder sessions, just open Felix Builder a second time.

3.1.2 Open Project

This function allows you to open an existing project in Felix Builder. The Felix Builder Project File (.builder) is a dedicated file format of Materialise and has the ability to save or load information other than STL data with a particular STL file or files (project). Due to the builder file, you can save a whole project at once instead of saving all parts (STL-files) separately.

3.1.3 Save Project

Saving STL-files in a project (*.builder) has the advantage of compressing the file and saving any other information such as platforms, scenes, measurements and annotations. The **Save** function will ask you to re-name your project while preserving the original project.

Use the *.builder file format if you want to continue your work another time. It will save everything as it is when you save and close Felix Builder.

3.1.4 Exit

By clicking on **Exit** you will exit the application. In case the last changes are not saved yet, you will be asked if you want to save them before closing the application.

3.2 Options

3.2.1 Smooth Shading

This option will influence how the part is rendered. The variations in color will be shown more gradually and no longer as separate triangles. Note that only the visualization of the part changes, the number of triangles and the accuracy of the STL are not changed.

3.2.2 Autoplacement on import

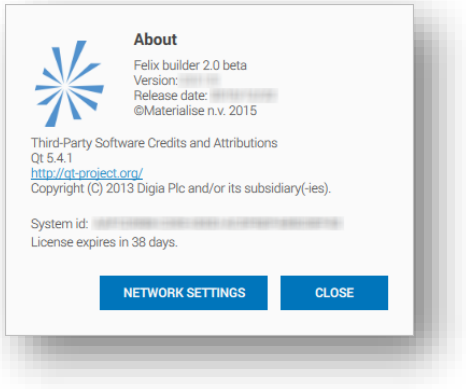
If this option is selected, the parts will be automatically placed around the center of the platform when importing them.

3.2.3 Communication Log

If you want to log the communication with your printer, select this checkbox or click Ctrl+L to display the communication log window.

3.3 Help menu

3.3.1 About



This button will open the about box of Felix Builder. In this dialog you can find the version number, copyright info and Third-Party Software Credits.

License info:

- System ID: unique identifier of your computer.
- License period: the time your license will stay valid.

3.3.2 Manual

This button will navigate to the support website of Felix Builder. There you can find the all info you need to use this software.

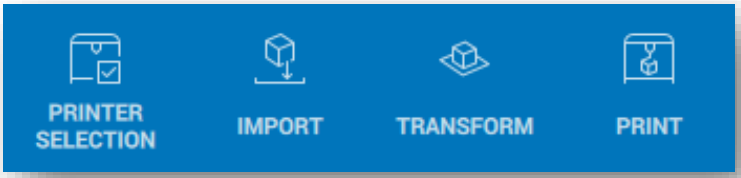
3.4 3DPrintCloud

A file that isn't printable? The Materialise 3DPrintCloud significantly simplifies the 3D Printing process, offering easy-to-use tools that allow you to prepare models for 3D printing in a few automated steps.

4 Printer selection

Felix Builder has a printer selection menu to change your selected printer.

5 Import part & Part list



Felix Builder is based on the STL-file format which is the 3D-Printing Industry's standard data format. An STL file is a triangular representation of a 3D object: it will describe surfaces as a collection of triangles, which makes them ideal for use in Rapid Prototyping or any environment that requires a triangulated file.

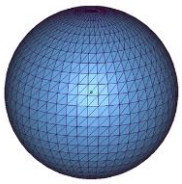
Unlike CAD, STL uses triangles and not entities to describe an object. Each triangle is uniquely defined by its normal and three points that represent its vertices.

Digital CAD

2 entities

- Center point
- Radius

STL-file



Triangle 1

Triangle 2

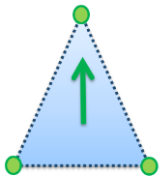
Triangle 3

Triangle 4

...

Triangle 1500

Single Triangle



1 Triangle
=
3 vertices
&
1 normal

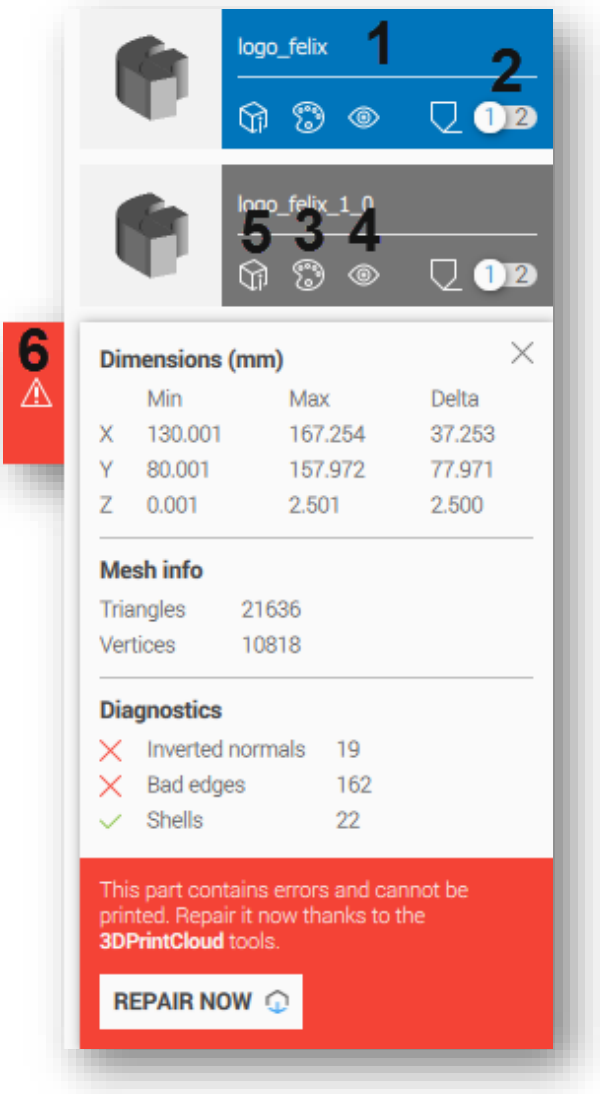
5.1 Import

Use this button in the flow toolbar to load STL-files on the platform.

After you opened your STL-files they will appear in the part list and on the platform.

5.2 Part List

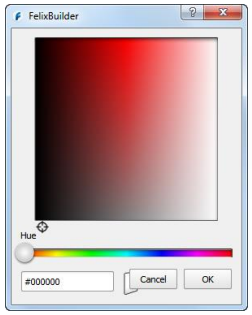
The Part list is a very useful and versatile tool for managing loaded parts. All loaded parts will be displayed in the list.



1. Selected/deselect part
   You can select/deselect a part by clicking on the tile in the part list or on the part on the build platform.

2. Change nozzle button
   By using this switch you will assign the part to a nozzle. This is only applicable when printing in part-part print mode with two nozzles.

3. Change color
   By clicking the button you can change the visualization color of the part.



4. Visible/invisible button
   By clicking on this button the part will turn invisible on the platform. Click it a second time to deactivate it.

5. Part info button
   When you press this button the tile in the part list will flip, and the part info will appear.

   Dimensions:
   The minimum, maximum and the bounding box XYZ-dimensions.

   Mesh info:
   Number of triangles and number of vertices.

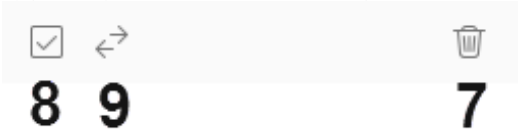
   Diagnostics:
   Information about the quality of the STL-file.
   Errors in STL-files can cause problems during slicing and building. Therefore you need watertight (and error-free) files. A good file contains one single Shell and zero Inverted Normals or Bad Edges.

6. STL error tooltip
   This warning button indicates if your STL-file contains errors. When you press it you will see the part info (same as 5. Part info button).

7. Remove Part
   This button will remove the part from the project. You can delete multiple parts by selecting multiple parts and pressing the backspace on your keyboard.

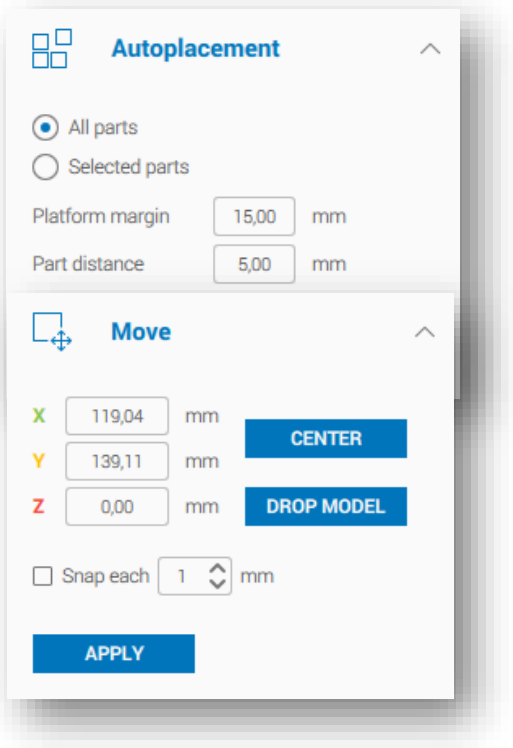
8. Select all parts
   By pressing this button all parts will be selected. Press it twice to deselect all parts.

9. Invert selection
   By pressing this button all deselected parts will be selected and vice-versa.



6 Transform

6.1 Autoplacement



This command will automatically place parts about the middle of the build platform. The part distance will define the space between the parts when placing the parts automatically. The platform margin will indicate how far the parts need to be placed from the borders of the platform.

The parts to nest selection will give the user the possibility to select all parts or just the selected ones.

6.2 Move

The move operation allows you to interactively move a part (or a group of selected parts) to another position.

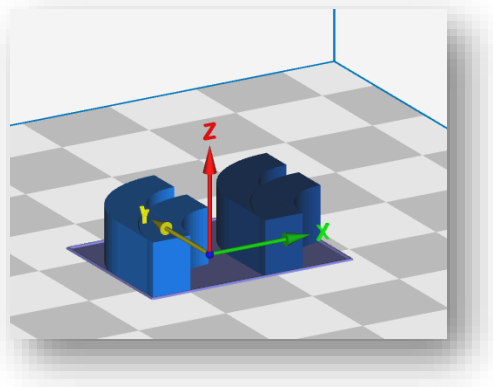
Grab an Axis or the blue plane on the platform to move the selected part(s) while visualizing in 3D view.

Use the Snap checkbox to move parts in defined increments. You can specify the step size yourself.

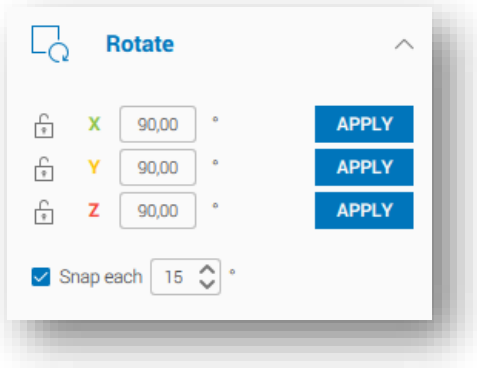
In addition, you can specify a relative movement: enter your desired value to move your part in XYZ directions.

You can use the Center button to move (in X, Y) the center of your selection to the center of the build platform.

Use the Drop Model button to move (in Z) the selected parts with their bottom plane onto the build platform.



6.3 Rotate

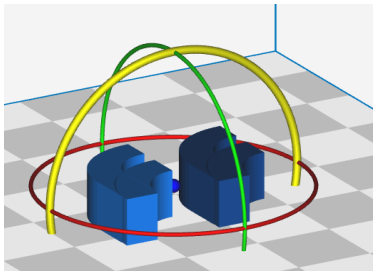


Grab one of the circles to interactively rotate about the chosen axis. You can also change your part selection during operation.

Use the **Snap** checkbox to rotate parts in defined increments. You can specify the step size yourself.

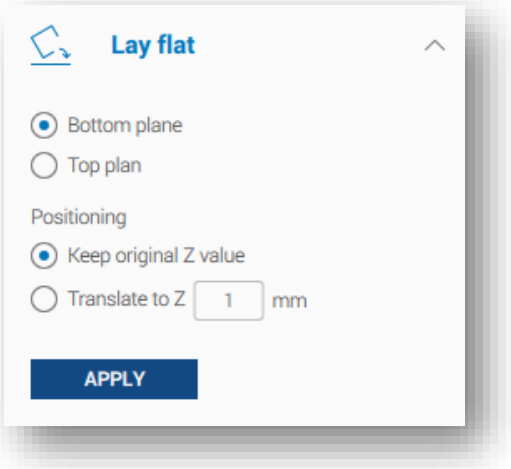
In addition, you can specify a relative rotation: fill in your desired degrees to rotate about the X, Y or Z axis. Press **Apply** to rotate.

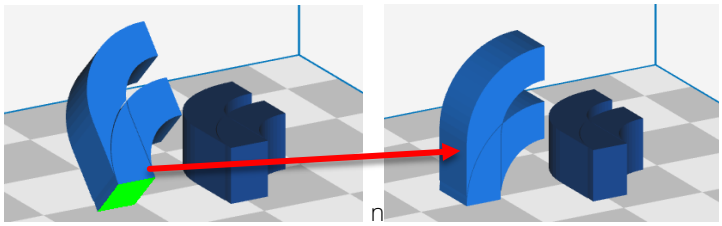
Use the lock icons to counter movement about the specified axis.



6.4 Lay flat

The Lay flat function allows easy orientation of the selected part by indicating a plane as the lay flat. This plane will automatically be oriented parallel to the platform (i.e. the XY-plane).





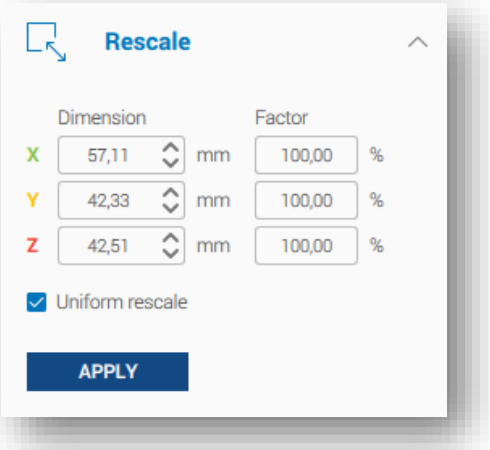
n

Indicate plane	Select you reference plane (the selected plane is indicated in green). The selected plane will be orientated parallel to the platform facing the upper or bottom plane (depending on your selection).

Positioning (after orientating the plane parallel to the platform)

Keep original z value	No translation is performed.
Translate to Z position	After orientation, the part is translated along Z to the entered minimal Z position.

6.5 Rescale

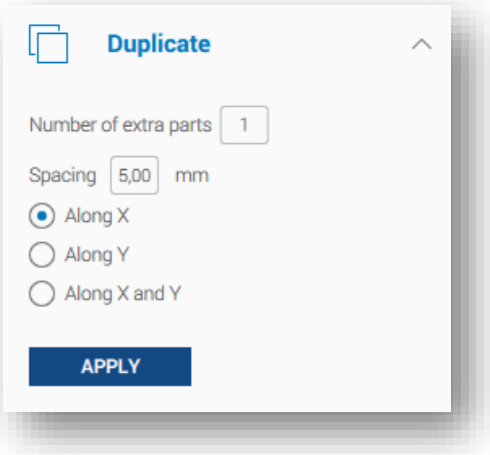


The Rescale function gives you the ability to resize the dimensions of a part as a whole or in only certain specified dimensions.

The *Factor* is a multiplying percentage for the dimensions in that direction. When the factor is 100%, no rescaling is performed, when the factor is 200%, size is doubled. A factor larger than 100 will enlarge the part, a factor smaller than 100 will shrink the part.

Enable *Uniform rescale* when you want to use an identical factor in all directions. Disable this option if you want to specify a different value in X, Y and Z.

6.6 Duplicate



This command automatically duplicates the selected part(s).

You can specify the number of extra parts you would like to create, the spacing between the duplicated parts and the direction you would like to duplicate in.

7 Print

7.1 Print Settings

To generate a file that is appropriate for your print job you will have to choose the right profiles and a print mode.

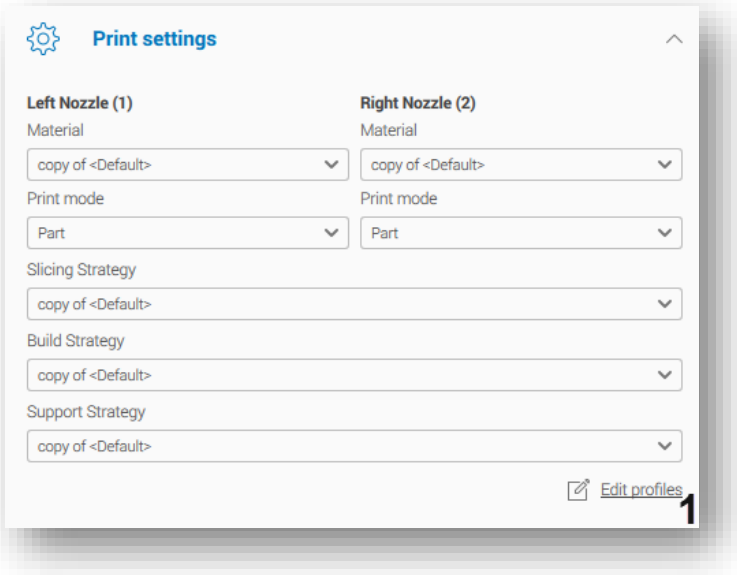
7.1.1 Profiles

Nozzle dependent

- Material: filament related parameters such as filament diameter, extrusion temperature and print speed, etc.

Nozzle independent

- Slicing Strategy: parameters of slice thickness and accuracy.
- Build Strategy: parameters for printing contours (or perimeters), upskin and downskin layers and infill (fill density of a part).
- Support Strategy: parameters for support material generation. This section also includes settings of raft, brim and skirt.



7.1.2 Print mode

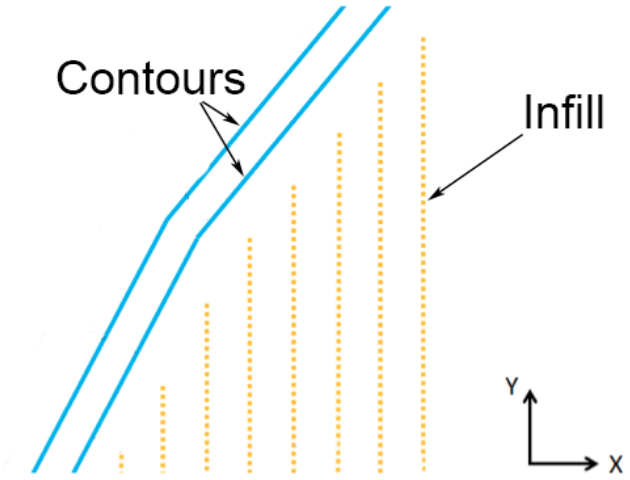
The print mode is a specific option for printing with two nozzles to indicate for which purpose you would like to use the nozzle.

Options:

Part - Part: the printer will use both nozzles to print parts. You can assign a part to a nozzle with the switch button in the part list. (See part list section)

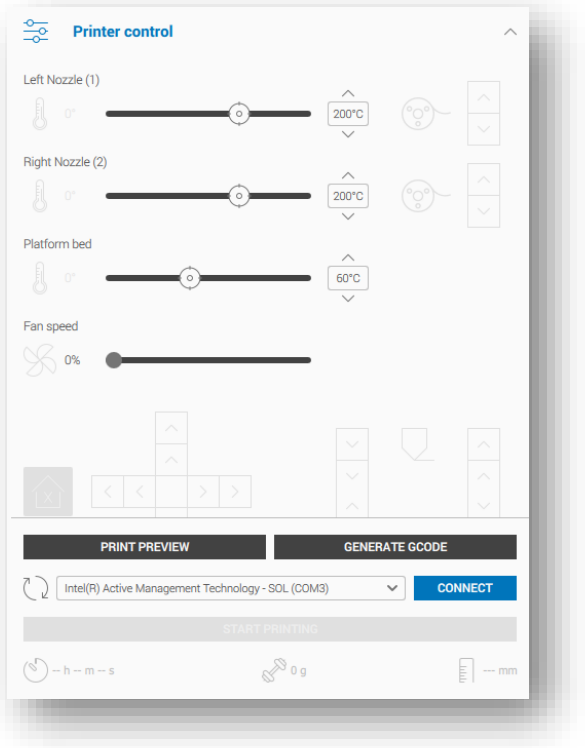
Part - Support: the nozzle that is in part-mode will print the actual part, the other nozzle will be in support mode and will print the support material.

Contour - Infill: the nozzle that is in contour mode will print the contours of a layer and the other nozzle will print the infill.



7.2 Printer Control

The Printer Control functions make it possible to directly send your print job to the 3D printer via USB connection. The printer can also be directly controlled by performing manual printer movements and adjusting temperature settings.



First you will have to select the COM port where your 3D printer is connected to. To be sure that it is the right one you can insert the USB cable and press the refresh icon. The port that appears should be connected to your 3D printer.

- **Port selection:** select the COM port where your 3D printer is connected to. Press refresh if the COM port you would like to connect with isn't in the list.
- **Connect/disconnect button:** By pressing this button you are going to setup a serial connection with your 3D printer. If you are connected the button will change to a disconnect button.

Note: If the port isn't appearing in the list please check if you have the right drivers installed.

To submit a print job that is appropriate for your 3D printer you will have to choose the right profiles and a print mode in the print settings section described above.

7.2.1 Start printing

When you are connected you can press Start Printing, the button changes to a print progress.

While printing a process bar appears to indicate the printing status. The first 12% of the bar is indicating the slicing progress, from that point on the bar moves by the progress of the print job.

The nozzle temperature will change to the ones that are set in the g-code. During a print you can monitor and manually change the temperature.

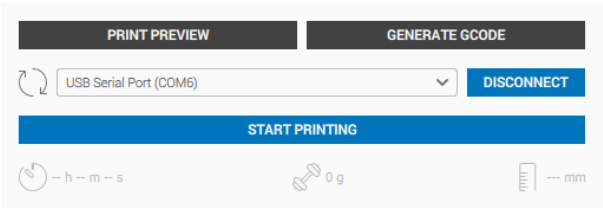
The pause button will pause a print job. On pause the printer will move up for 10 mm and to X/Y home.

The pause button becomes a resume button, the resume button will result the printer to resume the print where it was paused.

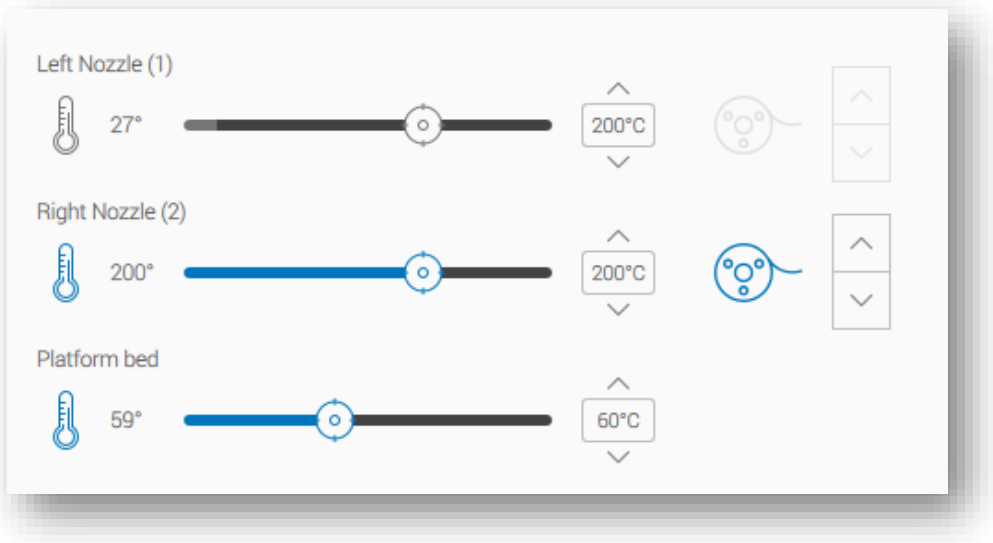
After the software performs a slicing operation it also calculates the estimated time of arrival the amount of mm filament that will be extruded in the job and the weight of the filament that will be used. The numbers will be displayed until you overwrite them by performing another slicing operation.

Slicing is done when generating a print preview, generating g-codes or when you start a print job from on the application.

When the printer is printing the numbers will also decrease when the print job is evolving.



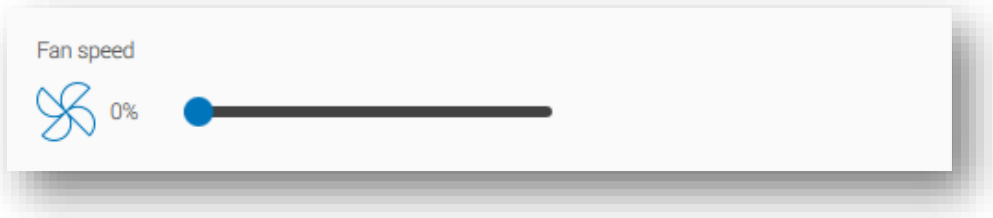
In the Printer Control you are able to control the hardware of your 3D printer directly.



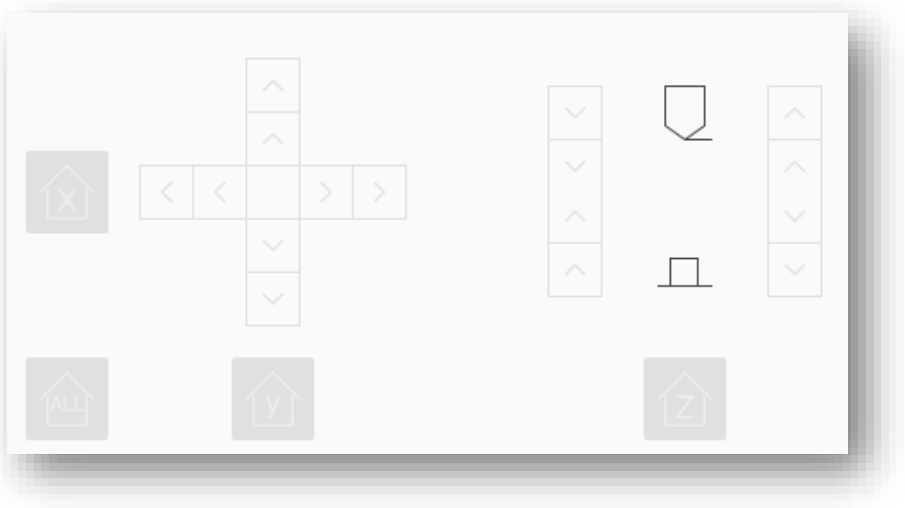
The nozzle(s) and platform bed blocks allow you to change the temperature. The temperature can be set either by changing the value on the right text field or by dragging the slider to a desired temperature. If you change it in the text field, you need to press return or leave the field to set the value. By clicking the left On/Off icon button the extruder or the bed is deactivated or reactivated. Next to the On/Off button the actual temperature is displayed. Next to that there is also a clear temperature indicator in the slider.

Note: You are able to change the temperature while printing, but if a temperature is set in the machine control file it will overwrite your manual set temperature.

The extrude filament controls next to the nozzle temperature controls are designed to let you manually extrude material out of the nozzle or retract it in the opposite direction. These controls are only active when the nozzle is heated to a temperature higher than 150°C. You can press and hold one of the two arrow buttons to manually extrude filament out of the nozzle or retracting it inside. The icon will spin in the way that the extruder motor is spinning to give a clear indication.



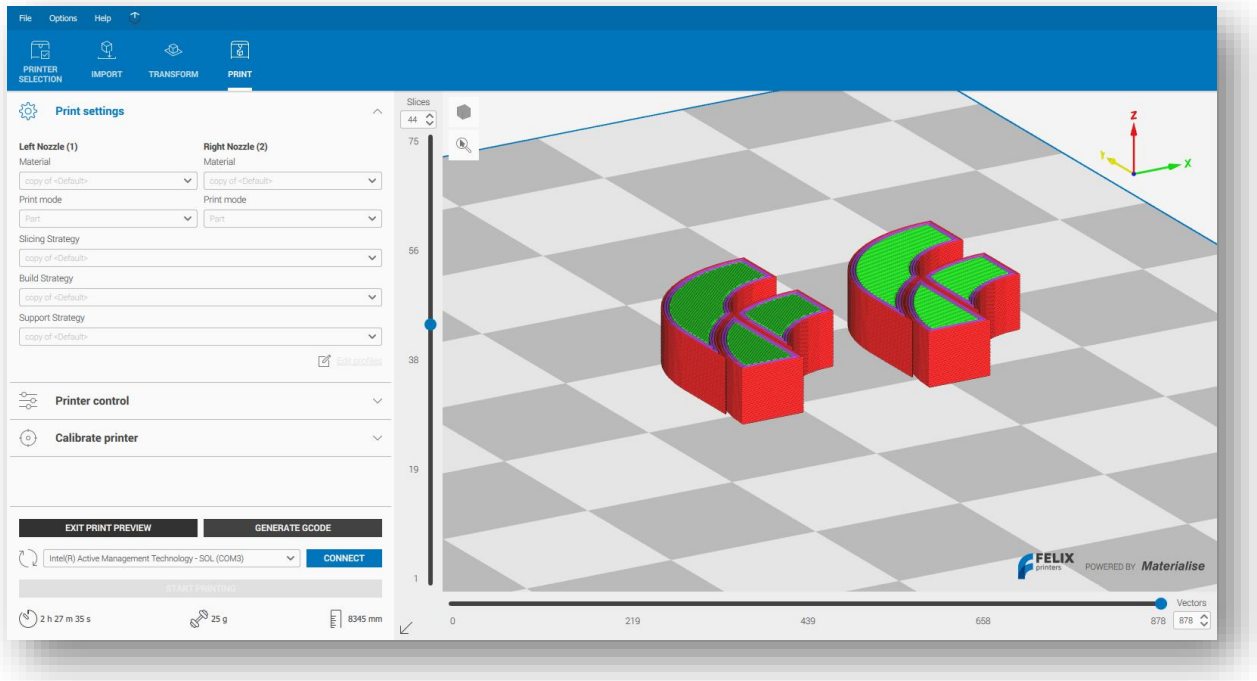
The fan speed control slider is designed to give feedback on the fan speed during a print job. It can be used to change the fan speed during a print job or in idle mode.



The next block controls the positioning of the extruder. With the arrow keys you move the extruder relative in any direction. You can perform big jumps by clicking on the button on the most outer part or do small jumps by pressing on the inside. The home buttons per axis will move the nozzle to home. The home all button will move all axis to home.

7.3 Print Preview

The print preview function can be used to check the slicing output in 3D. The slicing progress will be displayed and afterwards the slices are displayed.



The slider on the side can be used to drag through the slices, the bottom slider can be used to navigate through the vectors of the active slice.

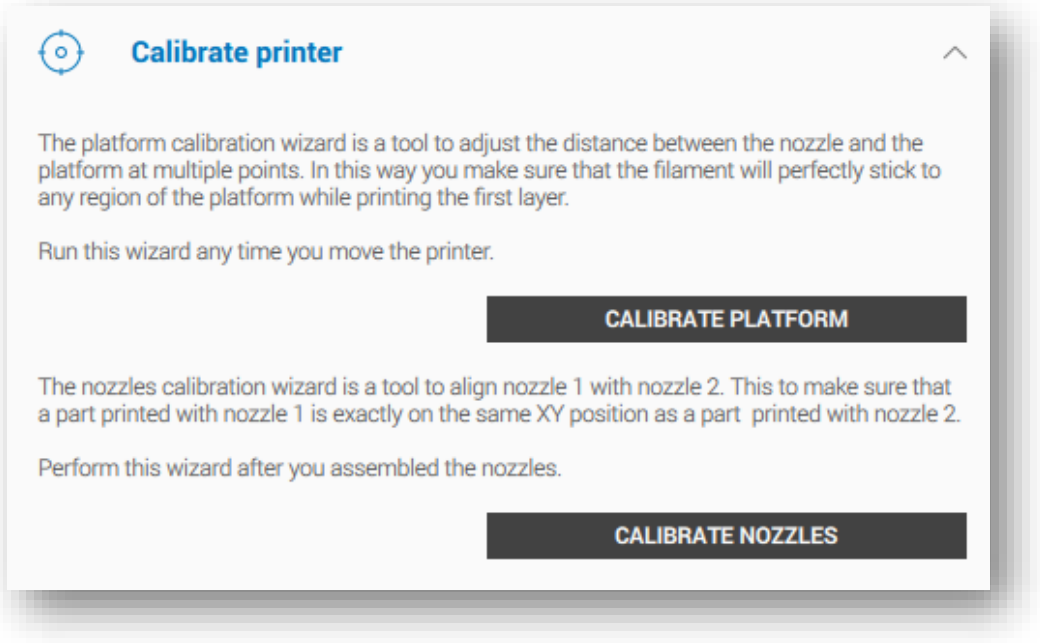
The estimated time of arrival is shown in the bottom left corner.

In the print preview mode you are not able change the selected print profile. You can exit the print preview mode by pressing the **Exit print preview** button or by pressing on another mode in the flow toolbar.

7.4 Generate g-code

If you want to generate a g-code file and transfer it for instance through an USB stick to your printer, you can click on the **Generate g-code** button. You will first be asked to specify a name for the g-code file that will be generated. After that, the part(s) will be sliced and the slicing progress will be displayed.

8 Calibrate printer



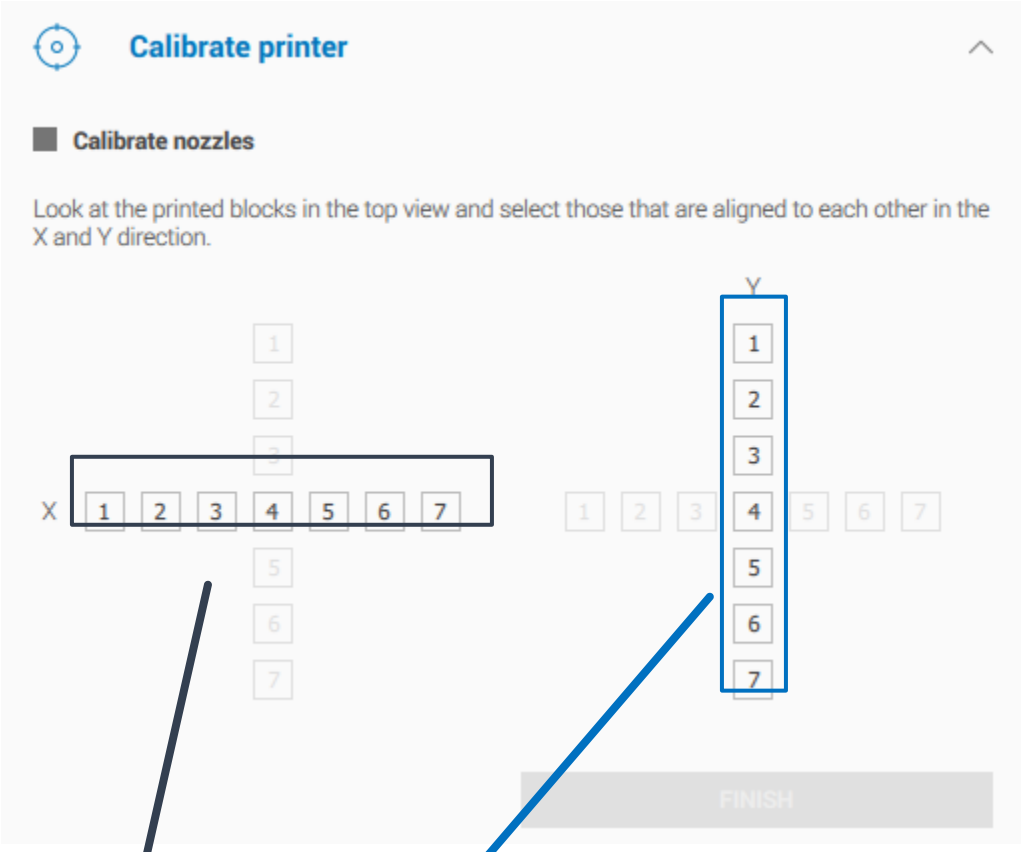
This wizard consists out of two functions: calibrate platform or calibrate nozzles.

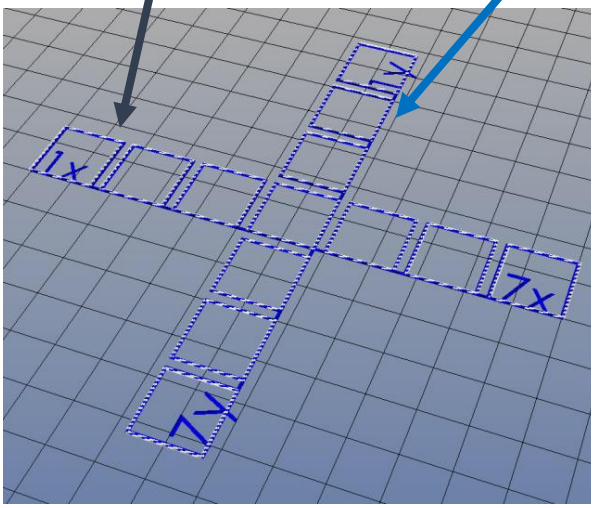
8.1 Calibrate platform

For a successful print it is important that the platform is properly calibrated. The distance between the nozzle and platform should be the same at every X, Y position. The leveling mechanism of the platform can be adjusted by turning the nuts underneath the heated platform bed. It is important to check it first, because the leveling can change during shipment.

Move the nozzle to all the points and check that the distance between the nozzle and the platform is equal across all the points. If it isn't adjust the nuts underneath the platform.

8.2 Calibrate nozzles (extra calibration for dual nozzle printers)





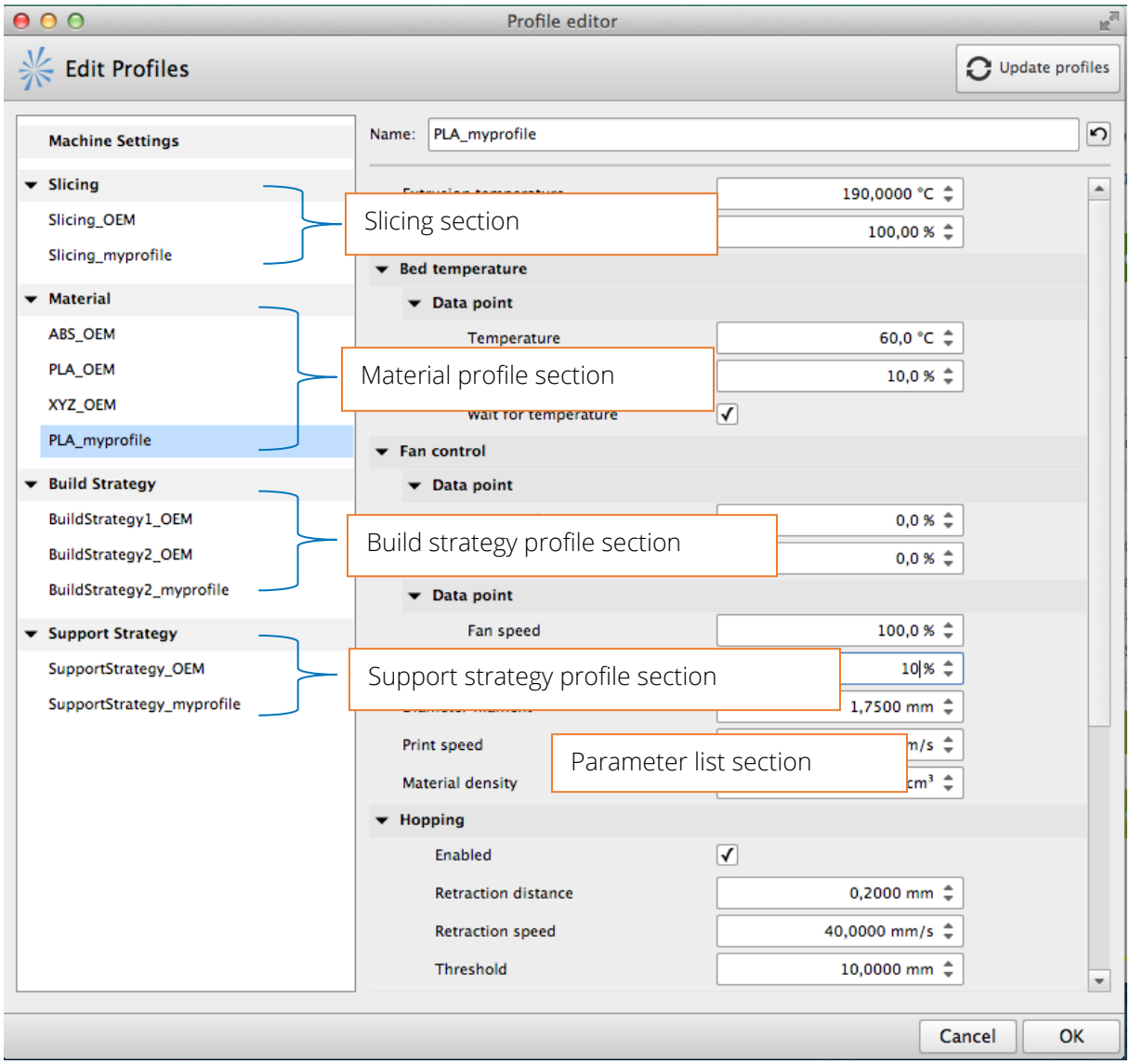
1. If both separate extruders have proven to print well, it is time to calibrate the distances between them. Print out the calibration parts by pressing the Print Parts button. In this print two layers of squares are printed. Layer one is printed with extruder 1 and layer two is printed with extruder 2 and shifted compared to layer one.

2. After you have selected the squares that align on top of each other, you will send these settings to the eeprom of the printer by pressing the Finish calibration button.

3. You finished the calibration and you should be able to print successfully with your dual head printer.

9 Profile editor

The profile editor allows you to setup and manage the print and printer parameters. These parameters are grouped into logical sections called profiles. Each profile has a parameter list attached to it that allow to make settings to the way the build processor behaves.



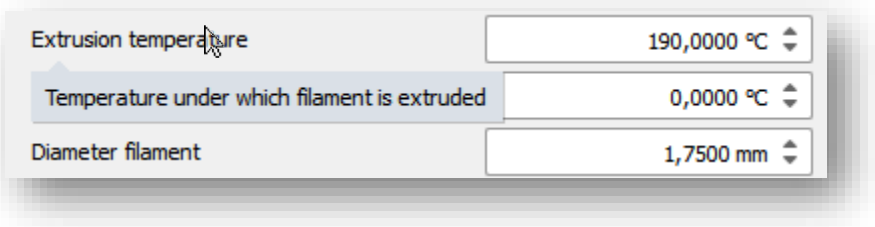
9.1 General behavior


This button will allow the user to make a copy of the profile that is select or it will allow the user to add an additional parameter in the parameter list section

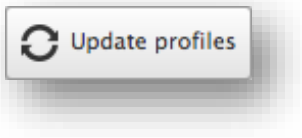

This button will allow the user to delete the profile that is selected. This will only appear when you hover over the parameter or profile.


This button will give the user additional info about the parameters it stands beside. This will only appear when you hover over the parameter or profile.

All parameters also have a tooltip it will appear when putting your cursor on the parameter name



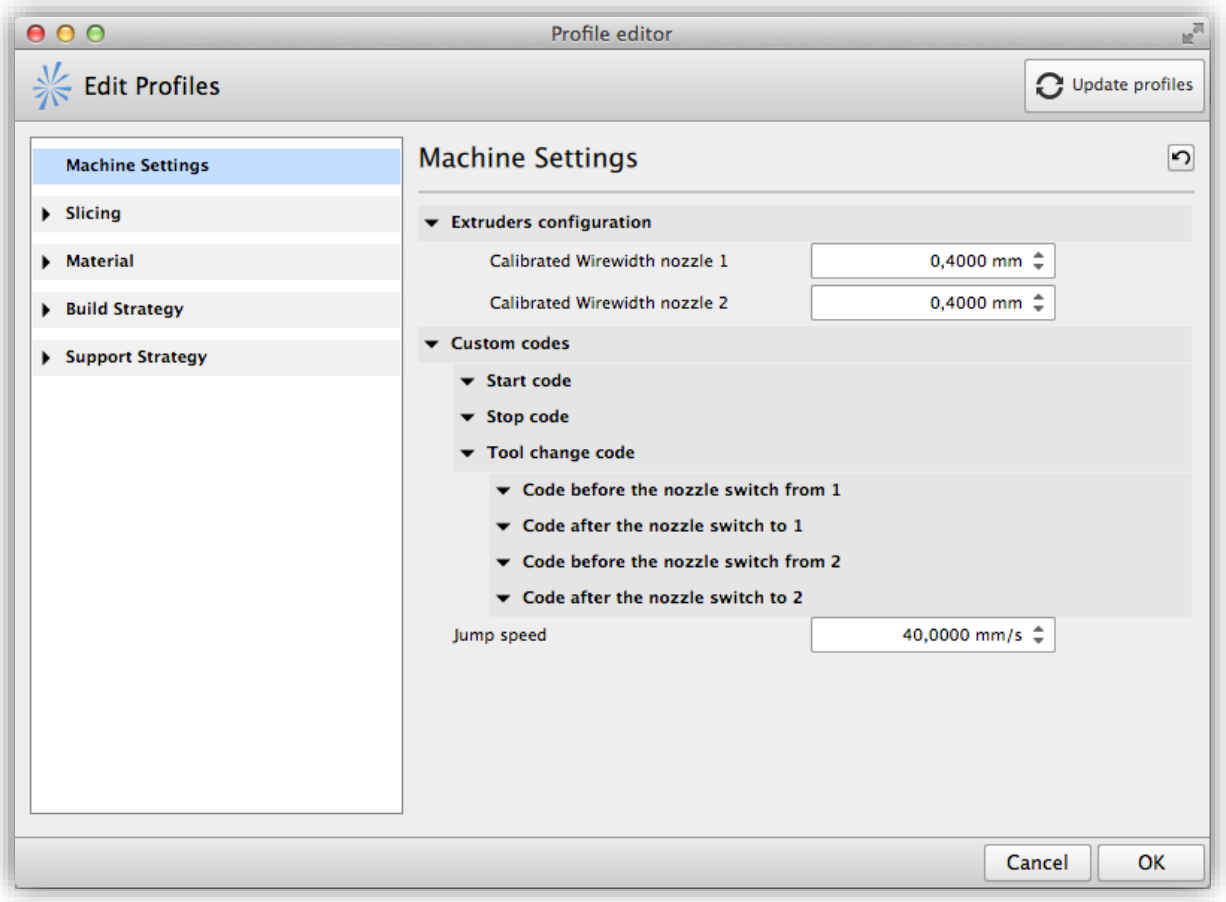
The update button will allow the user to download predefined profiles from the manufacturer's server.



The profiles generated by the manufacture are not removable or editable, you can duplicate the profile to make generate your own profile based on the one the OEM has created.

9.2 Machine settings

The machine settings define the parameters necessary to setup the software to work with a specific machine.



Extruder configuration

Here the user can define the nozzle diameters on his machine.

- *Calibrated Wirewidth nozzle 1:* sets the width of an extruded line with nozzle 1
- *Calibrated Wirewidth nozzle 2:* sets the width of an extruded line with nozzle 2

Note: For nozzle position, count from your right when standing in front of the machine

Custom codes

Here the user can insert custom define g-code commands into the output to change the behavior during special events of the build process. Those special events are:

- During the start of a build → start code
- During the end of a build → stop code
- During the change of a tool → before tool change code & after tool change code

Start code: Here the user can insert additional g-code commands that will be executed during the startup of a build

Stop code: Here the user can insert additional g-code commands that will be executed during the end of a build.

Before tool change code: here the user can insert additional g-code commands that will be executed before a tool change but after the build for that tool is finished

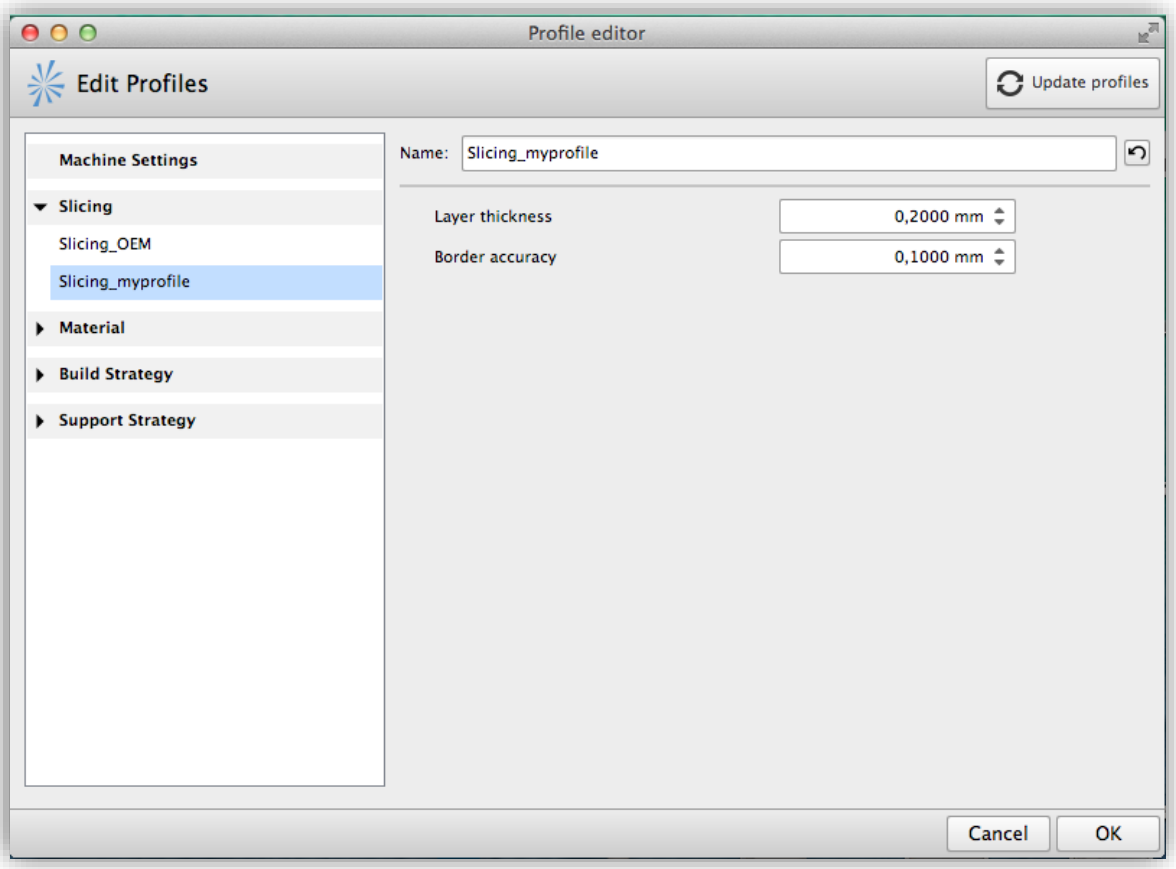
After tool change code: here the user can insert additional g-code commands that will be executed after a tool change but before the resume of the build for that tool.

Note: the build processor provide variables that can be used into the custom start and stop codes.

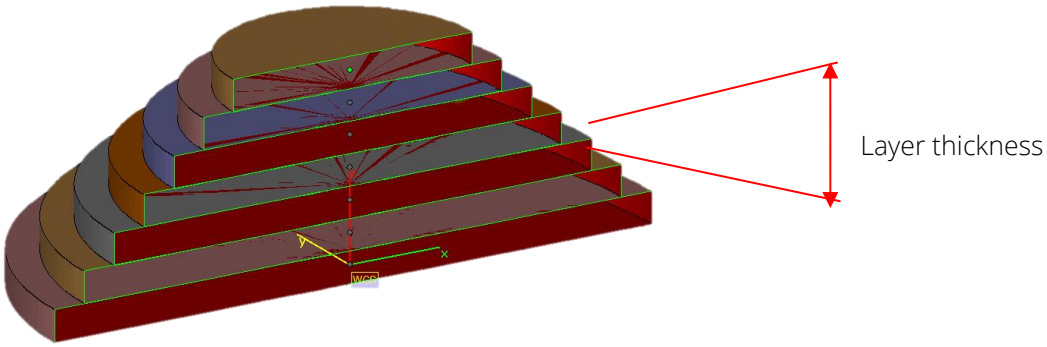
- ***%%Zstart%%:*** will give information about the start Z-level
- ***%%TemperatureNozzle1%%:*** will give a value corresponding with the set temperature for nozzle 1 like set in the corresponding profile
- ***%%TemperatureNozzle2%%:*** will give a value corresponding with the set temperature for nozzle 2 like set in the corresponding profile
- ***%%TemperatureBed%%:*** will give a value corresponding with the set temperature for the bed like set in the corresponding profile.
- {IF_NOZZLE1} If you enter this before start or stop code the following g-code will only be inserted when nozzle 1 is active in the print job.
- {IF_NOZZLE2} If you enter this before start or stop code the following g-code will only be inserted when nozzle 2 is active in the print job.
- *{IF_BED} If you enter this before start or stop code the following g*-code will only be inserted when the heated bed is active in the print job.

9.3 Slicing

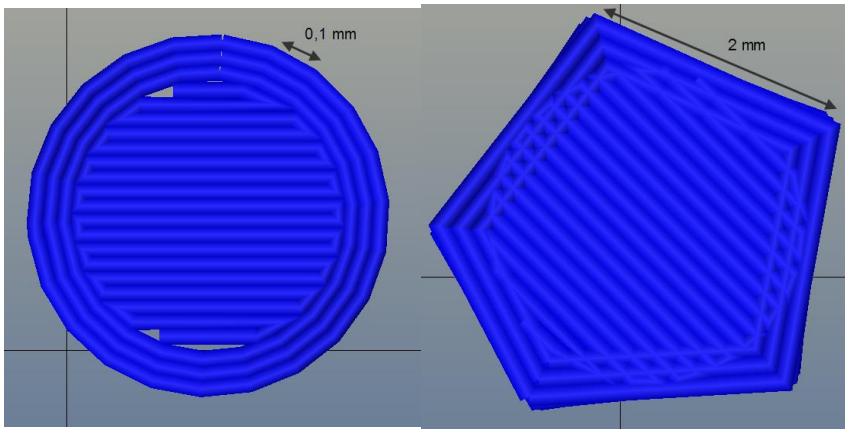
The slice profile section will allow the user to edit, manage and define the profiles that influence the slicing behavior of the build processor.



- *Layer thickness:* defines the thickness of the slicing process in mm

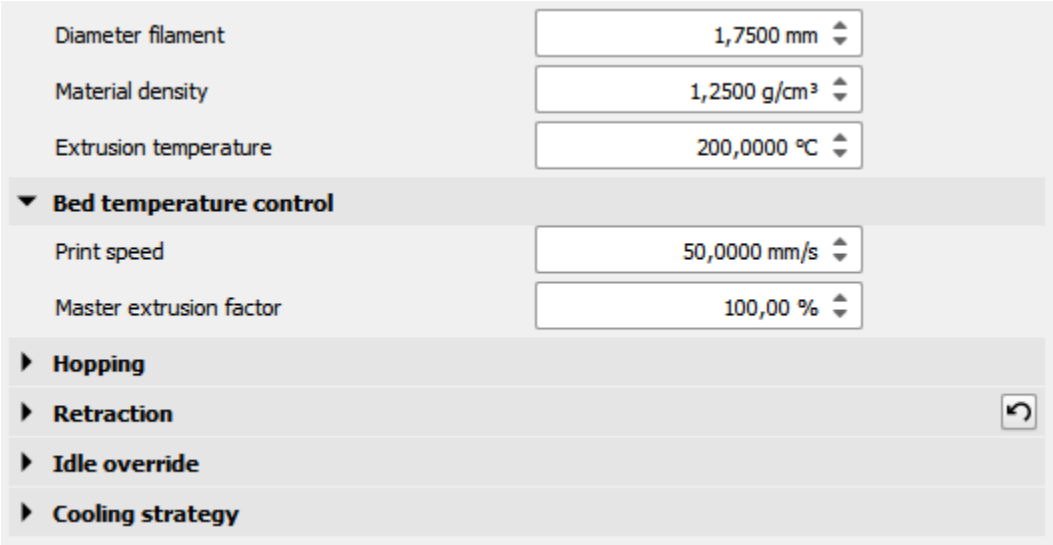


- *Border accuracy:* defines the maximum deviation in mm a toolpath is allowed to have towards the real toolpath. The higher the border accuracy the more deviation a toolpath is allowed to have and the more reduction of the amount of describing points we will have on curved paths. The lower the border accuracy the fewer deviation is allowed towards the real toolpath and the more describing points we will have in
curved paths.



9.4 Material

The material profile section will allow the user to edit, manage and define the profiles that define a material filament used for building a part.



Name: here the user can set the name of the material profile the parameter list corresponds to.

Diameter filament: here the user can set the diameter of the filament.

Material density: here the user can set the density of the filament in order to calculate the consumed material during a print.

Extrusion temperature: here the user can set the temperature the nozzle needs to have to process the filament.

Bed temperature control: here the user can set the temperature of the bed necessary to have the extruded material attached to the build platform. With the datapoints you are able to control the temperature of the bed during the print. By inserting different temperatures for another % of the total build height.

Print speed: Moving speed of the nozzle when filament is extruded. This is the reference speed for the other speed settings in Build Strategy and Support Strategy.

Master extrusion factor: Amount of filament will be extruded during the print. If this is set as the default value (i.e. 100%), the print path will be closer to the wirewidth set in Machine setting. A smaller value (i.e. <100%) will result in less extrusion and thinner print path, and vice versa if the value is larger than 100%.

Note: the extrusion factors in other profile settings are all relative values to this master extrusion factor

9.4.1 Hopping

Enabled: with this checkbox the user can enable hopping during printing

Distance: here the user can set the retraction distance when hopping

Speed: here the user can set the speed of retraction when hopping

Threshold: here the user can set the threshold at which hopping will occur. When a jump is larger than the threshold hopping will occur when it is lower than the threshold the jump will be executed in the normal way.

9.4.2 Retraction

Retracting is used to avoid oozing material when not actively extruding, avoiding blobs, especially on the surface. Pulling back the filament lowers the pressure in the nozzle, pulling in any molten filament.

Retraction Distance: length of filament being retracted. Should not be too much for direct drive extruders. Bowden-style extruders need more.

Retraction Speed: speed of the retraction. As fast as possible, but take not too strip your filament.

Prime Distance: the length of filament pushed into the nozzle before resuming printing (priming). Should be close to the retraction distance for direct drive extruders.

Prime Speed: speed at which the filament will be pushed into the nozzle when priming.

Threshold: Minimum distance to move before engaging retraction.

9.4.3 Idle override

When printing with two nozzles, the idle (non-extruding) nozzle can influence the printed part by heat or oozing. The idle override feature helps to solve this issue by retracting the filament and cooling the idle nozzle down to the idle temperature when switching nozzles. The print head will be parked away from the print until the idle nozzle has cooled down.

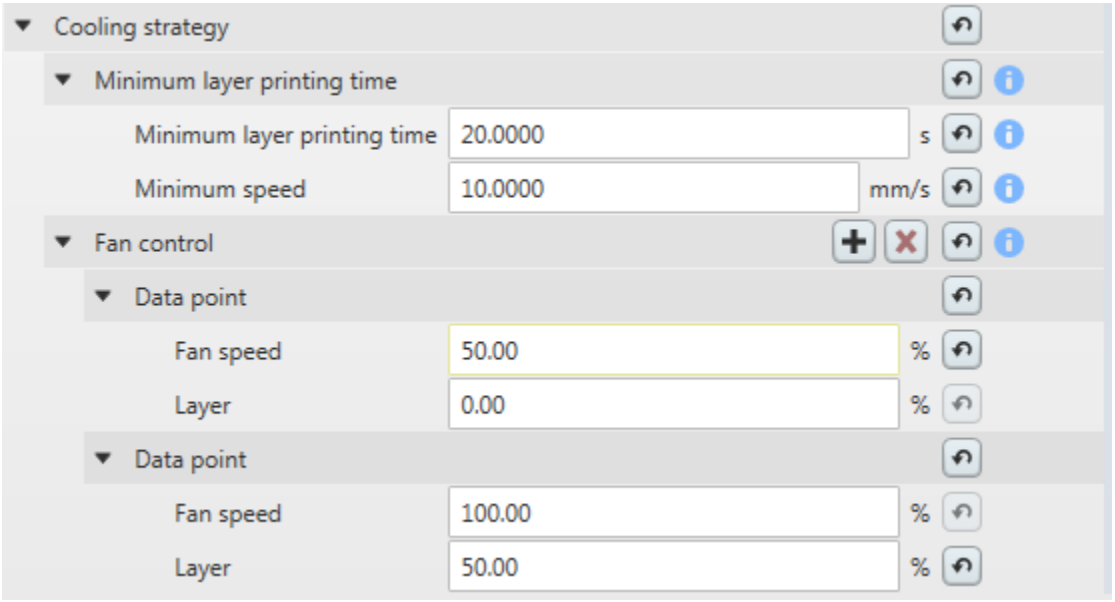
Enabled: this checkbox will enable to place a nozzle in the idle state when it becomes idle in dual nozzle printing.

Retraction speed: here the user can set the speed of retraction when a nozzle goes into idle state.

Retraction distance: here the user can set the distance of retraction when the nozzle goes into idle state.

Idle temperature: the temperature a nozzle will be cooled down to when it enters idle mode. This temperature will be maintained until the nozzle becomes active again.

9.4.4 Cooling strategy



FDM technology has difficulties achieving a smooth surface quality when working with small areas. This is usually caused by a too fast print speed that doesn't allow the previous layer to cool down and solidify sufficiently. Lowering the printing speed is an option, but will slow down the whole print. OpenFDM can slow down print speed when the layer print time is too short.

Minimum layer printing time: Minimum time used to print a layer. If printing a layer takes less than this threshold, the printing speed in this layer will be decreased till the minimum printing time is reached.

Minimum speed: the print speed will never drop under this. When a layer still takes less than the minimum layer printing time, the nozzle will wait before starting the next layer.

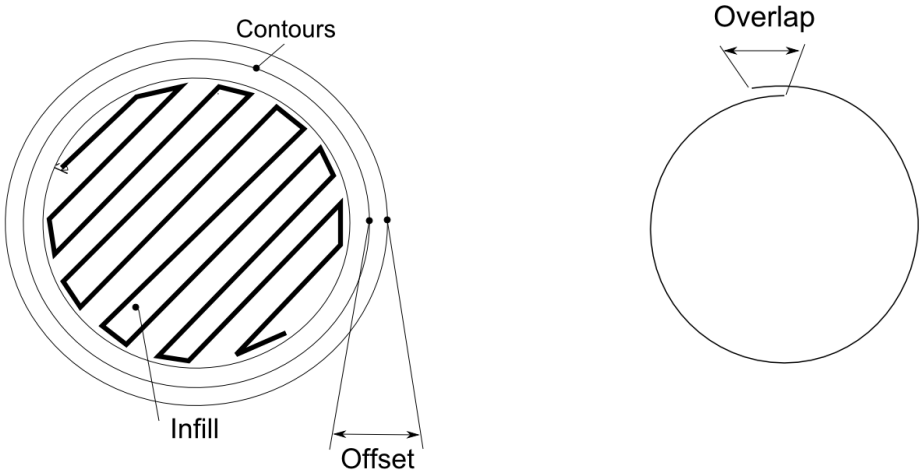
Fan control: The cooling fan behavior can be finely tuned to the layer height. Each datapoint marks a change in fan speed. Keep the fan low for the first few layers.

The deer pictured is printed at 25mm/s, but the antler was printed much slower, automatically.

9.5 Build strategy

Name: here the user can set the name for the build strategy profile

9.5.1 Contours



Number of contours: the number of boundaries printed each layer. More will make your part stronger, but slow down the print

Contour overlap: to fill the potential gap which may occur between the start and end points of a contour so as to produce a better surface quality. A negative value is allowed.

Contour before infill/Infill first: hatching order in every layer. Usually, the surface quality is better when printing the contours first, but in some cases, e.g. steep overhangs, reversing this can yield good results.

Offset: distance between the adjacent boundaries. Contours too close or too far influence adhesion and print quality. This value is defined as the percentage of the default width of the print path, and is only applied when more than one contour is printed.

Hatching order: Defines if the contours are printed inside-out or outside-in. Only relevant when more than one contour is printed, obviously. Inside out can sometimes increase print quality.

Print speed factor: nozzle moving speed when the contours are printed. Defined as a relative value (%) of the print speed in Material settings. Slower results usually in better quality.

Extrusion factor: amount of filament being extruded to print the contours. Defined as a relative value (%) of the default. Changing this value can influence the width of the printed path.

Thin walls: when parts of the model are thinner than the wirewidth, a contour is generated anyway. Double thin walls guarantees that any part of the model has two contours, further reinforcing the thin parts.

Start point relocation: defines the starting point of a contour. Use this to reduce 'stitching', where a seam is visible when the start and endpoints consecutive layers align.

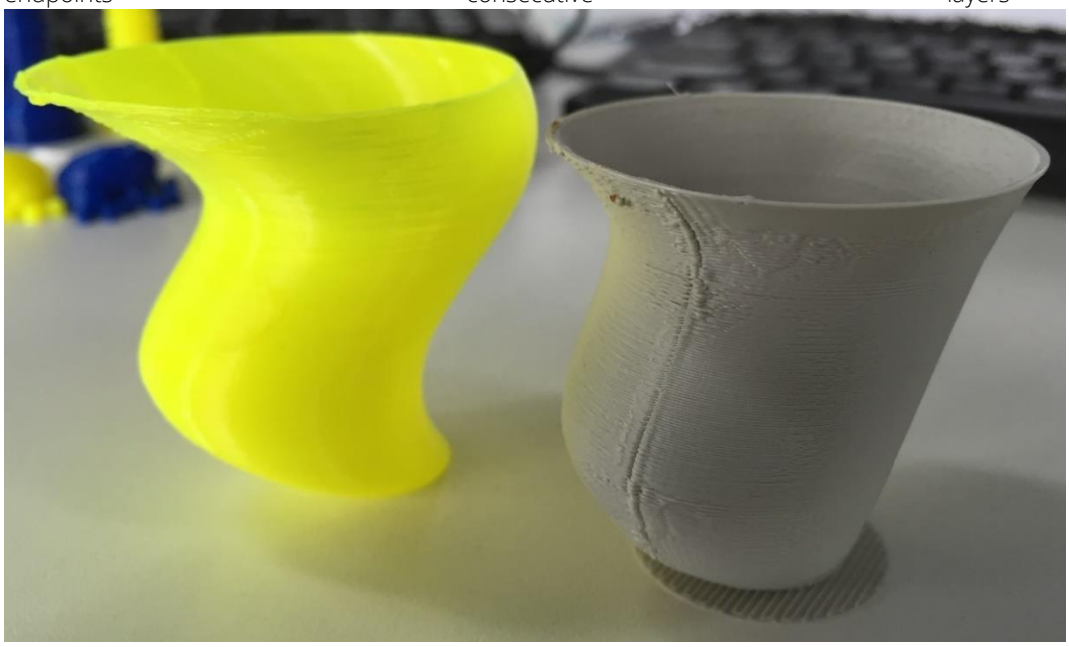


Figure: A visible seam on the grey object. The yellow was printed with the 'random' option.

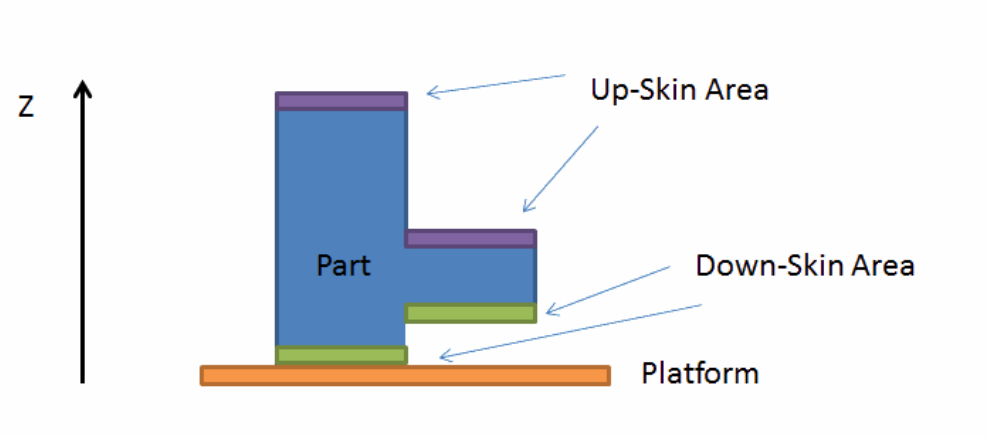
Repeat: all the contours will be printed from the same location at each layer. This can leave a seam on the printed part.

Sharpest angle: if a part has sharp angels in its geometry, the contours will start to be printed from the location where the sharp angles are. The seam will not be as notable.

Random: on the contrary to "Repeat", this option will randomize the start point of each contour in a layer. A seam will not form, but little defects will be spread over the surface.

9.5.2 Up skin / Down skin

Hatching parameters of the up-facing surfaces



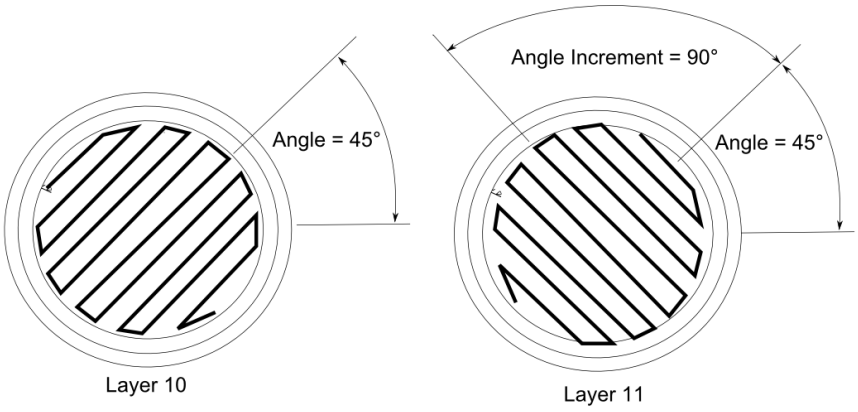
Number of layers: Number of layers considered upskin. These will be filled in solid, in order to improve surface quality.

Line distance: distance between the hatching vectors of the upskin. It is a relative value (%) of the default width of the print path. When it is set as 100%, the hatching vectors are printed right next to each other.

Infill overlap: This parameter defines the overlap % of the inner contour and the infill of the up/downskin. This ensures that there are no holes in the surface and the part is as strong as it can be.

Angle: initial angle of hatching vectors in the first upskin layer regarding to X-axis.

Angle increment: rotating angle of the hatching vectors in the current layer comparing to previous layer. 90° gives usually the best result.



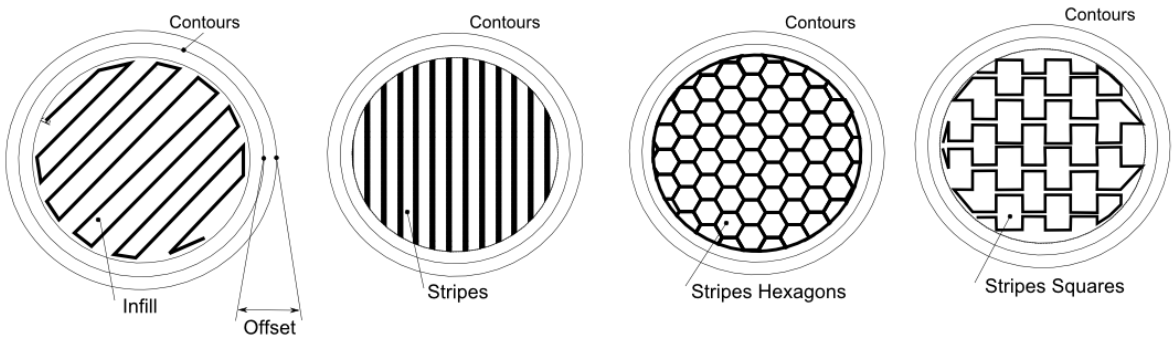
Print speed factor: nozzle moving speed when upskin vectors are printed. It is a relative value (%) of the print speed in Material settings.

Extrusion factor: amount of filament being extruded to print upskin vectors. It is a relative value (%) of the default. Influences the width of the printpath.

9.5.3 Infill

Pattern: here the user can set the pattern that will be used for processing the infill area. There are 4 different patterns that can be used for infill:

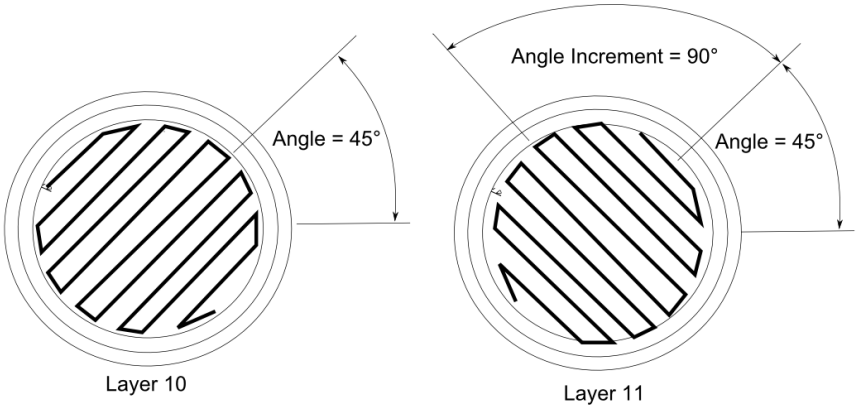
- Infill: this is a line filling pattern where we can define the density of the line fill.
- Stripes: this is a line pattern with a fixed space between the lines.
- Stripes Hexagons : this is a hexagon pattern
- Striper Squares : this is a square wave pattern



Size: here the user can set the size of the infill pattern as a percentage of the default size of the pattern

Infill overlap: This parameter defines the overlap % of the inner contour and the infill. This ensures that there are no holes in the surface and the part is as strong as it can be.

Angle: here the user can set the angle at which the infill pattern will be printed.



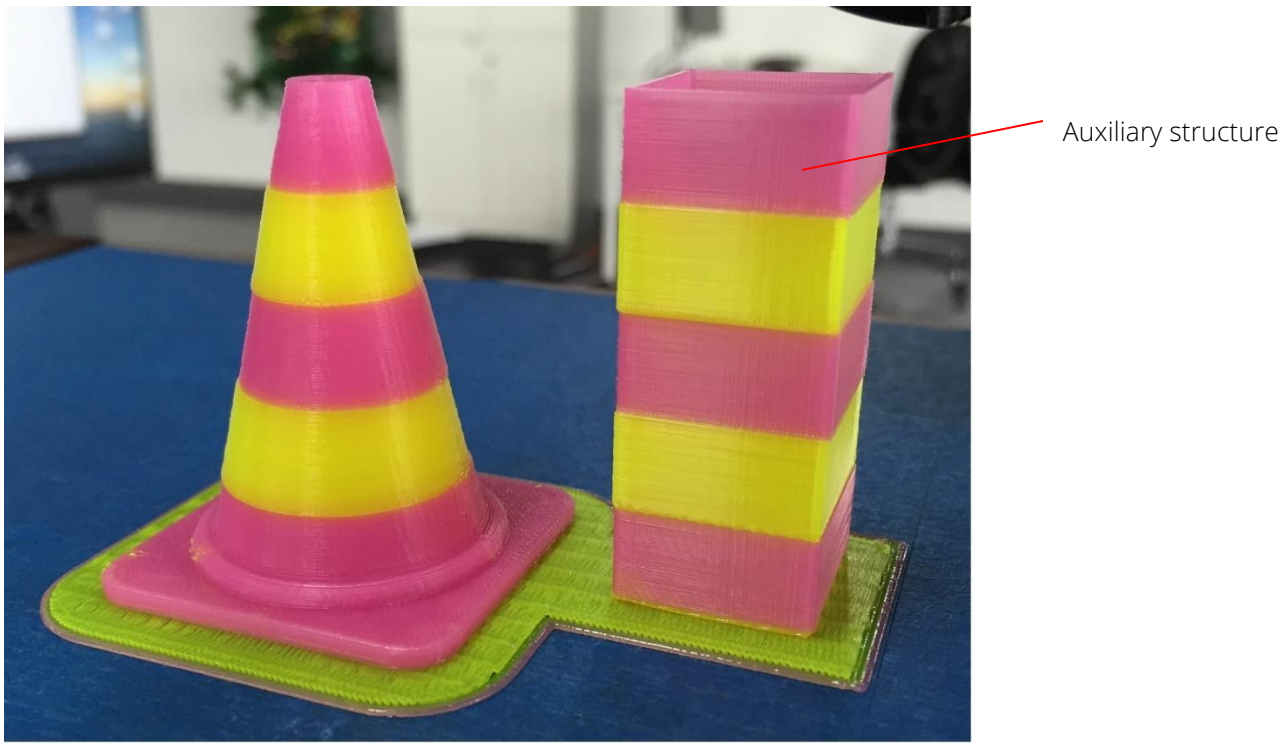
Angle increment: here the user can set the angle increment that will be used when printing consecutive layers.

Print speed factor: here the user can set the speed at which the infill will be processed as a % of the nominal speed defined in the material profile.

Extrusion factor: here the user can set the amount of extrusion as a % of the master extrusion rate defined in material profile.

9.5.4 Auxiliary structure

This feature is for dual nozzle modes. It primes each nozzle by printing an auxiliary structure on the side of the platform when the nozzles are switched. In addition, the nozzle switching sequence will be automatically optimized when auxiliary structure is enabled so as to reduce nozzle switching times.



9.6 Support strategy

Name: here the user can set the name for the support strategy profile

9.6.1 Support

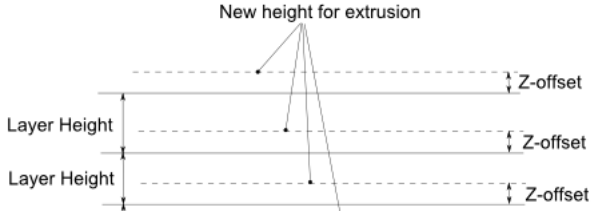
Enabled: here the user can activate the support generation function

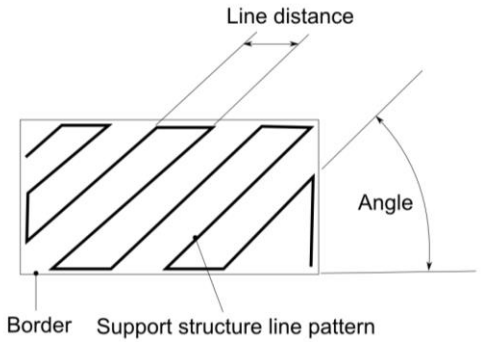
Add border: here the user can activate the generation of a border around the support structure.

Z-offset: here the user can set an additional offset on the layer height used to print the support structure.

Line angle: here the user can set the angle that will be used to draw the support structure lines.

Line distance: here the user can set the distance between 2 lines of the support structure pattern

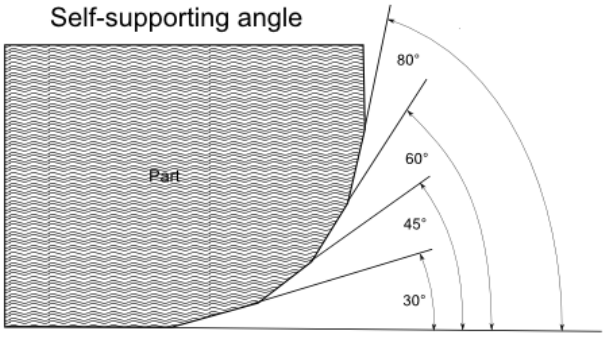




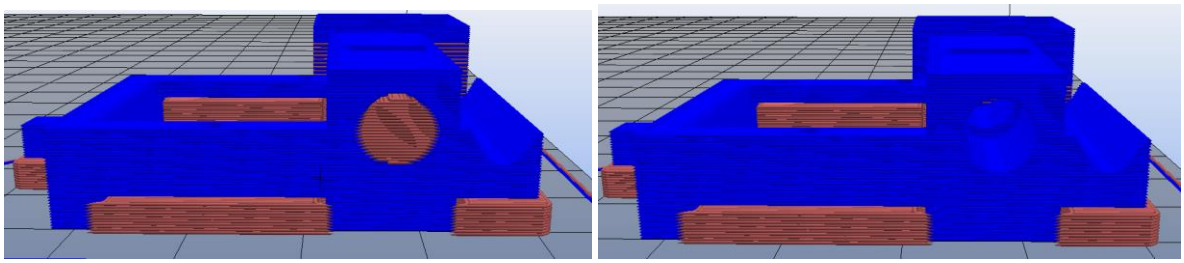
XY extension: here the user can set the offset in X & Y that will be used to generate the support structure. The support structure will be extended in X&Y

Distance to part: here the user can set the distance the support structure needs to keep from the part(s) during generation of the support structure.

Self-supporting angle: here the user can set the self-supporting angle that will be used to determine which parts of a model need support and which not during the support generation. All areas that have a support angle lower than the self-supporting angle will be supported.



Small area threshold: Small areas under this threshold will not get support. Useful when support is difficult to remove. Set this threshold can remove the support generation from the areas smaller than the threshold. An example is given below. The support generated inside of the tube is different to remove after the build. With this feature, setting the threshold can remove the support to be built inside of the tube.



The left image is a g-code view with the small area threshold set to 0; the right image is a g-code view with the small area threshold set to 50.

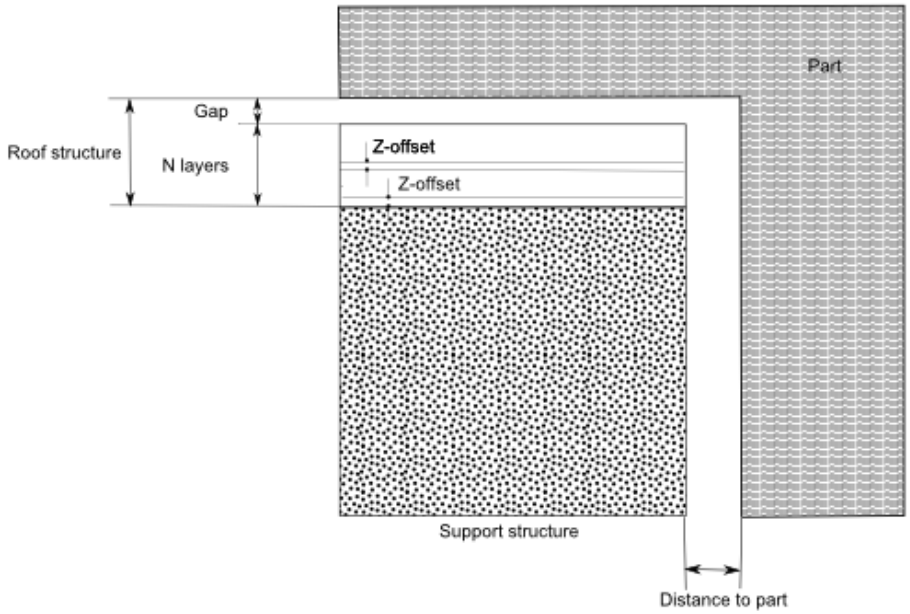
Print speed factor: here the user can set the speed at which the support structure will be processed as a % of the nominal speed defined in the material profile.

Extrusion factor: here the user can set the amount of extrusion that will be used for the support structure as a % of the master extrusion rate defined in material profile.

Roof

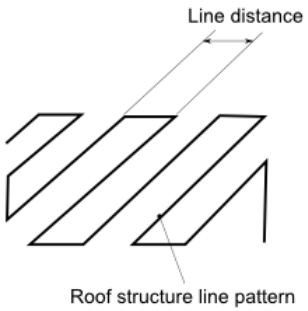
Enabled: here the user can activate the generation of a roof on top of the support structure to make an easy removable interface between part and support.

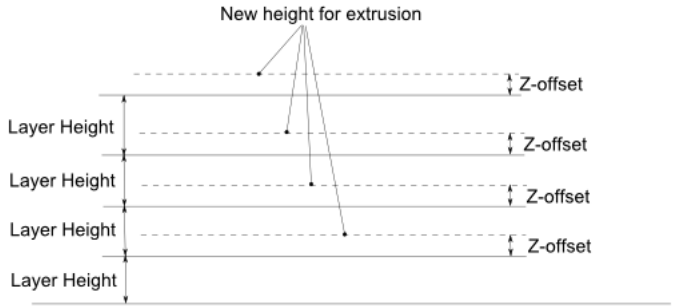
Gap: distance between the last layer of the roof and the first layer of the downskin area. This setting determines how easy the support generation (including roof) can be removed from the part after the print. Larger the gap is, it is easier to remove the support generation. However, quality of the down skin surface can be affected. In general, gap is multiple of the layer thickness.



Number of layers: here the user can set the number of layers that will generated to construct the roof.

Line distance: here the user can set the distance between 2 lines of the line infill used to generate the roof structure.





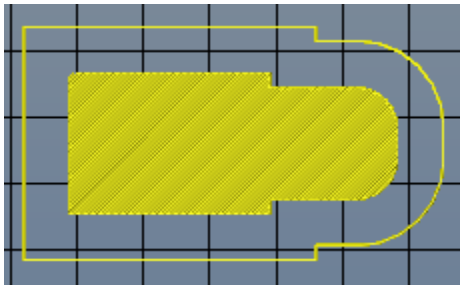
Z-offset: here the user can set an additional offset on the layer height used to print the roof structure.

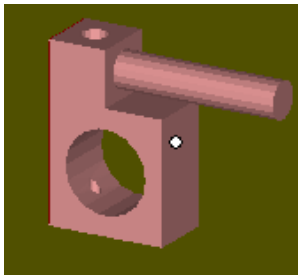
Print speed factor: here the user can set the speed at which the support structure will be processed as a % of the nominal speed defined in the material profile.

Extrusion factor: here the user can set the amount of extrusion that will be used for the support structure as a % of the master extrusion rate defined in material profile.

9.6.2 Skirt

Enabled: here the user can activate the generation of a skirt for priming the nozzles.

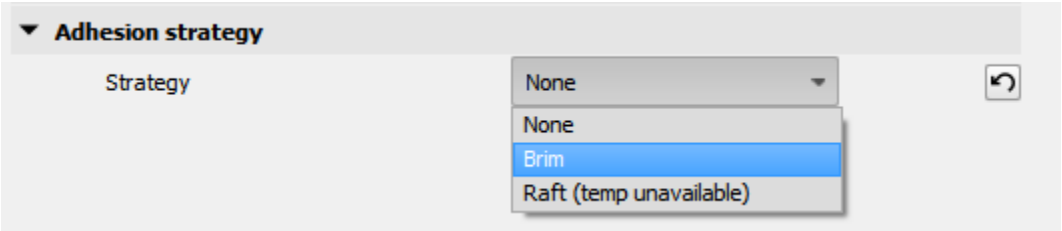




Print speed factor: here the user can set the speed at which the skirt structure will be processed as a % of the nominal speed defined in the material profile.

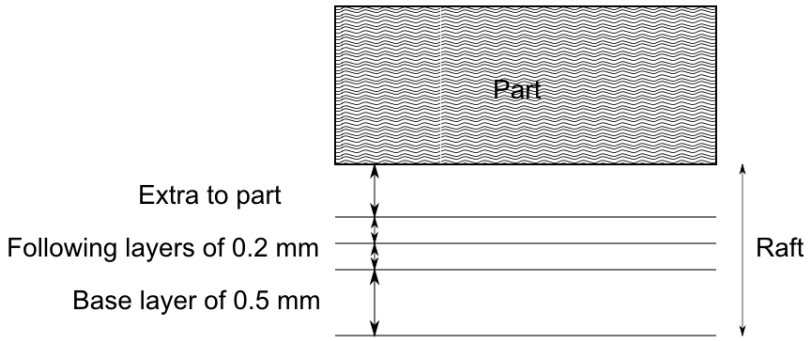
Extrusion factor: here the user can set the amount of extrusion that will be used for the skirt structure as a % of the master extrusion rate defined in material profile.

9.6.3 Adhesion factor



9.6.4 Raft

Number of following layers: here the user can set how many layers will be generated after the base layer.



Final raft layer temperature: here the user can set the extrusion temperature for the final layer of the raft.

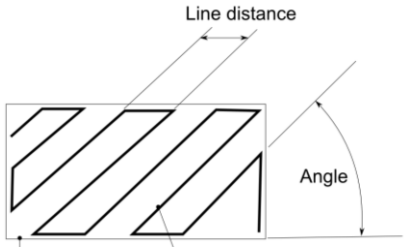
Extra to part: here the user can set the gap between the raft final layer and the part.

Base layer / Following Layers

Layer thickness: here the user can set the layer thickness of the base layer

Line angle: here the user can set the line angle of the line infill of the base layer or following layers

Line distance: here the user can set the distance of the line infill of the base layers or following layers



Print speed factor: here the user can set the speed at which the raft structure first layer or following layers will be processed as a % of the nominal speed defined in the material profile.

Extrusion factor: here the user can set the amount of extrusion that will be used for the raft structure first layer or following layers as a % of the master extrusion rate defined in material profile.

9.6.5 Brim

As a user you can define a brim to ensure that you parts will adhere with the platform.

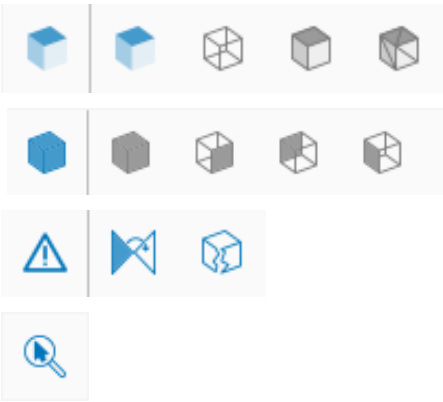
Number of contours: the number of times it will circle around the part at the first layer. More contours will result in a better adhesion.

Print speed factor: The print speed factor controls the speed for the brim relative to the print speed defined in the material profile.

Extrusion factor: here the user can set the amount of extrusion that will be used for the brim structure as a % of the master extrusion rate defined in material profile.

10 Visualize

10.1 Workplace visualization bar



10.1.1 View options

Shading: display the parts with shaded surfaces.

Wireframe: shows edges of the object. The edges are defined by the angle of incidence between triangles.

Wireframe and shading: this is a combined view state off shading and wireframe.

Triangles: shows the edges of all the triangles in the part. This view visualizes the raw STL data.

10.1.2 Visualize options

This option is there to change the view state to a ISO view.

Options:

- Default view
- Front view
- Back view
- Left view
- Right view
- Top view
- Bottom view

10.1.3 Visualize STL –errors

In this bar you are able to choose if you would like to visualize the STL-errors or not.

The button will enable the visualization of flipped triangles, if enabled the flipped triangles will be displayed in red.

The button will enable the visualization of bad edges, if enabled the bad edges will be visualized in yellow.

11 System requirements

11.1 MAC

11.1.1 Minimal hardware requirements

Hardware:

- A Mac computer with an Intel Core 2 Duo, Core i3, Core i5, Core i7, or Xeon processor
- Minimum 2 GB of memory
- About 150 MB of disk space on the boot volume (Macintosh HD) for installation
- Minimum resolution (scaled): 1280 x 800

Software:

- OS X Mavericks 10.9 or later

11.2 WINDOWS

11.2.1 Minimal hardware requirements

Hardware:

- CPU: Core i3, Core i5, Core i7, or Xeon processor
- Minimum 4 GB of memory
- About 150 MB of disk space on the C: drive for installation
- Minimum resolution: 1280 x 800
- 'NVidea GeForce400' or 'ATI/AMD Radeon HD 5000' or higher recommended
- OpenGL 1.5 compatible graphical card
- At least 1 GB of memory

Software:

- Windows 7 (64bit)
- Windows 8 / 8.1 (64bit)