



Fusion3TM

REACTORTM

3D Printing Software

USER MANUAL

Version 1.0.0

powered by
**Create it
REAL** 

Contents

1	Introduction	2
2	Technical Requirements	3
3	Activation	4
3.1	Offline Activation	4
4	Updating profiles	6
4.1	Online activated	6
4.2	Offline activated	6
5	Basic Workflow	7
5.1	Presets and Profiles	8
5.2	Model manipulations	9
5.3	Camera	13
5.4	Preparing and showing Layers	14
6	Terminology	16
7	Troubleshooting	18
7.1	Bed adhesion	18
7.2	Underextrusion	18
7.3	Zippers	19
8	Advanced Topics	20
8.1	Modifier objects	20
8.2	Planned Pauses	23
9	Index of settings	25
9.1	Quality Settings: Basic	25
9.2	Quality Settings: Advanced	29
9.3	Quality Settings: Expert - Printer	38
9.4	Quality Settings: Expert - Layers	43
9.5	Quality Settings: Expert - Support	55
9.6	Material settings: Basic	63
9.7	Material settings: Advanced	65
9.8	Material settings: Expert	73
9.9	Printer settings: Others	76

- 9.10 Printer settings: Workspace 77
- 9.11 Printer settings: Motion 81
- 9.12 Printer settings: Materials 83
- 9.13 Printer settings: Bed leveling 84
- 9.14 Printer settings - Custom G-Code 85
- 9.15 Application 85

Chapter 1

Introduction

REACTOR is 3D printing software designed for preparing your models for 3D printing. It is designed to be easy-to-use while including many advanced features required by expert users. The software, introduced in late 2020 and powered by the CreateItReal REALVISION slicing engine, is designed to work only with Fusion3 3D printers.

One of the first unique features is the integration with Fusion3's Certified Material List and provides real-time, online material profile updates for all the available materials certified to run on Fusion3 3D printers.

An exciting feature within REACTOR is the Variable Layer Height Modifier. While you can specify the overall quality setting of the entire part (ex: Standard Settings or 200 micron layer height), this feature allows you to add different layer height settings for different sections of your part. For example, you might have a curved section of a print encompassing 100 layers which you wish to print at 100 micron layer height. After selecting these in the Modifier tool, the algorithms within REACTOR will automatically tune the print quality in that section for the best quality and detail.

Over time please look for other unique features that harness the capabilities of your Fusion3 3D printer. Of course, should you have suggestions of new features that you'd like to see within the software, we welcome your ideas.

Chapter 2

Technical Requirements

Operating System:

Windows: Windows 7 or higher, 64 bit

Mac OS / Linux: Not Available – please contact Fusion3 for Alternatives

System Requirements:

Processors: Intel Core i5 4000 or similar

Hard Disk: 1Gb available disc-space

Memory: Minimum 4Gb of RAM

Chapter 3

Activation

Upon request you should have received an installer for the software, as well as an activation key that is required to use REACTOR. During the installation you will be asked for activation, which can be done at this moment or later.

If REACTOR is installed without activation, it can be activated later by following these steps:

1. Start REACTOR
2. Go to "Help" menu
3. Click "Activate"
4. Click "Retype product key"
5. Enter product key and validate

3.1 Offline Activation

REACTOR can be installed using an offline activation process for highly secure environments. To activate Offline you must have chosen this option from Fusion3. The Offline activation feature is only available for customers within the defense sector, or in branches of US government that require REACTOR to run in environments that cannot use the standard online activation. The offline activation requires a specific installer that can be provided by Fusion3.

Step 1: Run the REACTOR Offline Installer. When the installation is complete you can start the activation by navigating to "help", and selecting "Activate"

Step 2: Note your License ID. Your license ID needs to be provided to Fusion3 on the phone.

Step 3: Call Fusion3. On the phone you need to provide your License ID, and receive the activation key. Note the Serial Key, so you are ready to input it into REACTOR

Step 4: Input the Serial Key. Input the Serial Key in REACTOR and press activate.

Chapter 4

Updating profiles

REACTOR allows for easy updating of certified Fusion3 material profiles, both for Online activated and Offline activated versions.

4.1 Online activated

4.2 Offline activated

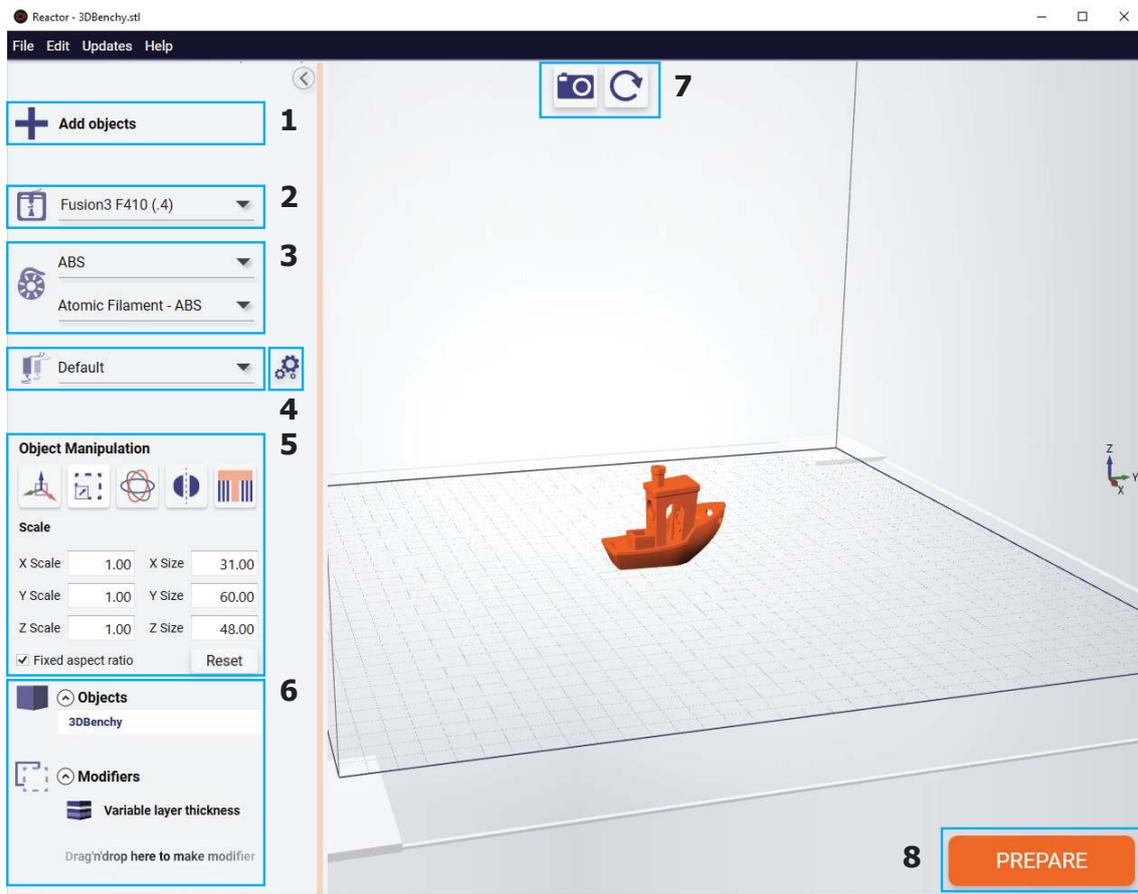
Whenever an official update of material profiles is ready, you will receive a .profilupdate file from Fusion3. This file needs to be transferred to the workstation where REACTOR is running.

After the profile update is complete the file will be deleted from the hard drive, so it can be a good idea to make a backup if necessary.

Chapter 5

Basic Workflow

Once REACTOR is installed you are ready to use it with your Fusion3 printer. The basic workflow of REACTOR, from opening a model to exporting a G-Code can be seen below.



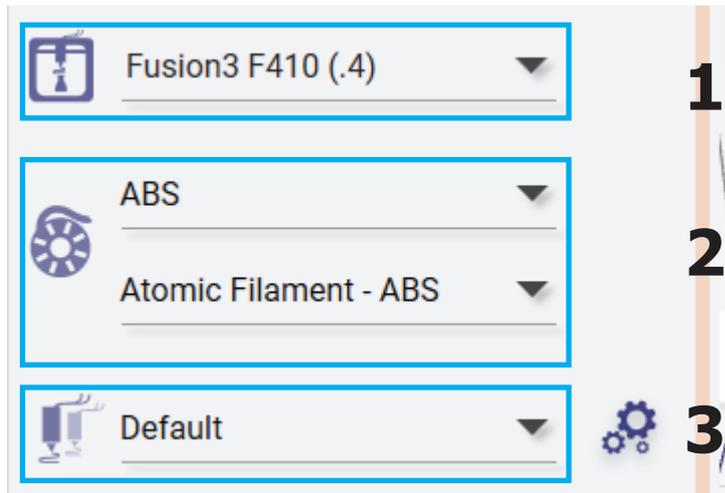
1. **Open File:** Start by opening an .STL file.

2. **Choose printer:** Make sure that you select the correct Fusion3 printer from the drop down.
3. **Choose material:** Use the dropdown to choose between different materials. For the full list of certified materials, choose "add new" from any one of the drop down menus.
4. **Choose preset** Either choose an existing preset or open settings to adjust the printing options. See section 5.1 for more info
5. **Model manipulations:** Set up the workspace using: Move, Scale, Rotate, Mirror, and Support. See section 5.2 for more info.
6. **Objects and Modifiers:** Drag'n'Drop a model to the modifier area, to make a modifier. This modifier can be used to apply certain settings to certain areas. See section 8.1 for further information.
7. **Workspace controls:** reset the camera or the workspace. Resetting the workspace will reload all .STL files in their original scale and default placement.
8. **Prepare:** Press "Prepare" when you are ready to create a G-Code file. This button will turn into an "Export" button once preparing is completed.

5.1 Presets and Profiles

REACTOR supports a wide range of materials, through the profiles.

1. Select which Fusion3 Printer you use from the list.
2. Choose material and brand. If the material is not found in the drop down press "Add new" and select from the complete list of certified materials.
3. Select a preset printing profile.



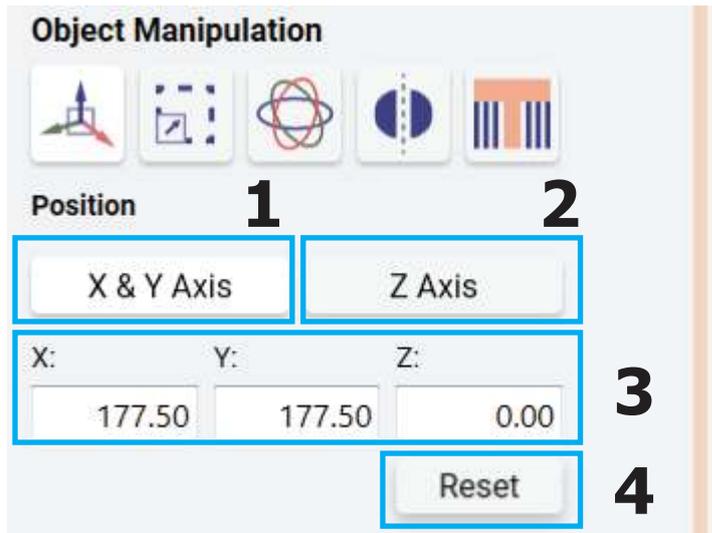
5.2 Model manipulations

In the following we will go through the model manipulation menu in more detail.

5.2.1 Movement

Movement is the default option in REACTOR

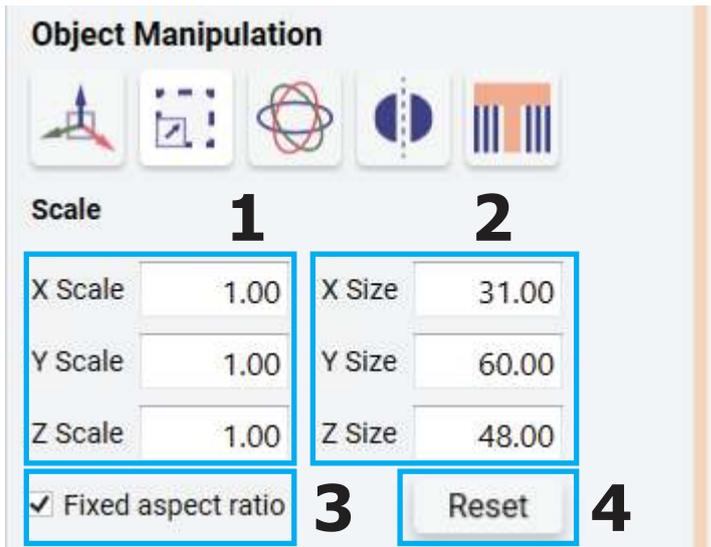
1. When this option is selected (which it is pr. default), you can move around objects in the workspace in the XY plane, by Left clicking and dragging.
2. By selecting this option, you can move an object on the Z-Axis. Note that moving an object below the workspace will result in only part of the object being printed.
3. These fields contain the coordinates of the currently selected model. Values can be added to the field to accurately change an object's location.
4. Pressing "Reset" will put all objects back in their default position.



5.2.2 Scaling

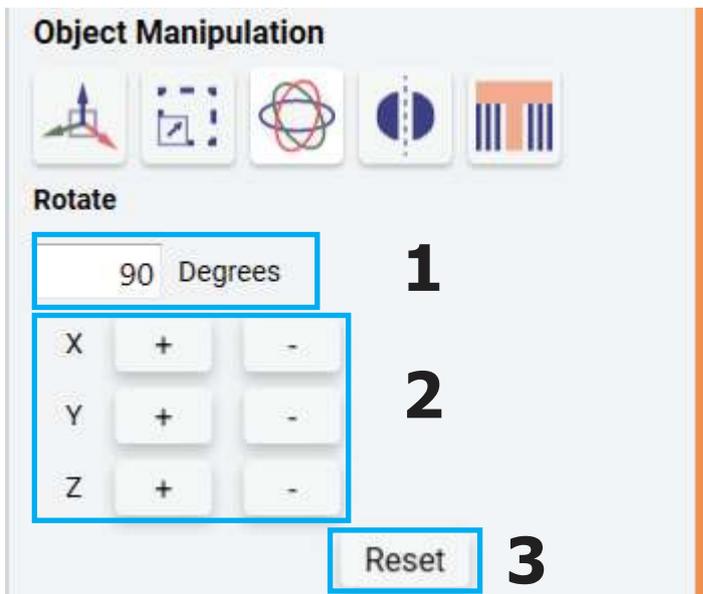
With these operations you can scale a model up to become larger, or down to become smaller.

1. These fields allow you to scale each axis with a certain factor. Scaling an object with a factor of 2 will make it twice as large.
2. These fields allows you to scale an object to an absolute size.
3. While this box is checked, scaling will happen uniformly on all axes. Unchecking it will allow scaling an object on only one axis.
4. Press this button to reset all scaling actions.



5.2.3 Rotating

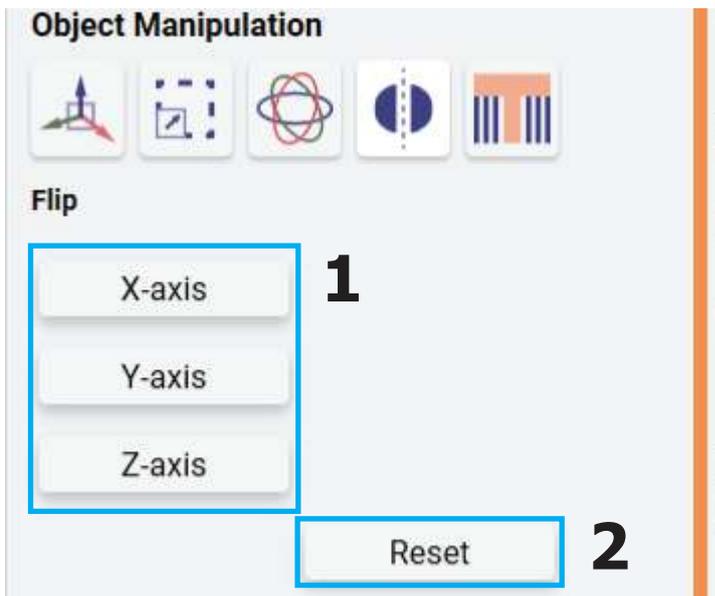
1. Choose the number of degrees you wish to rotate your model.
2. Rotate the model around the X, Y, or Z Axis.
3. Press the reset button to undo all rotations.



5.2.4 Flipping

This allows you to flip or mirror an object along an axis.

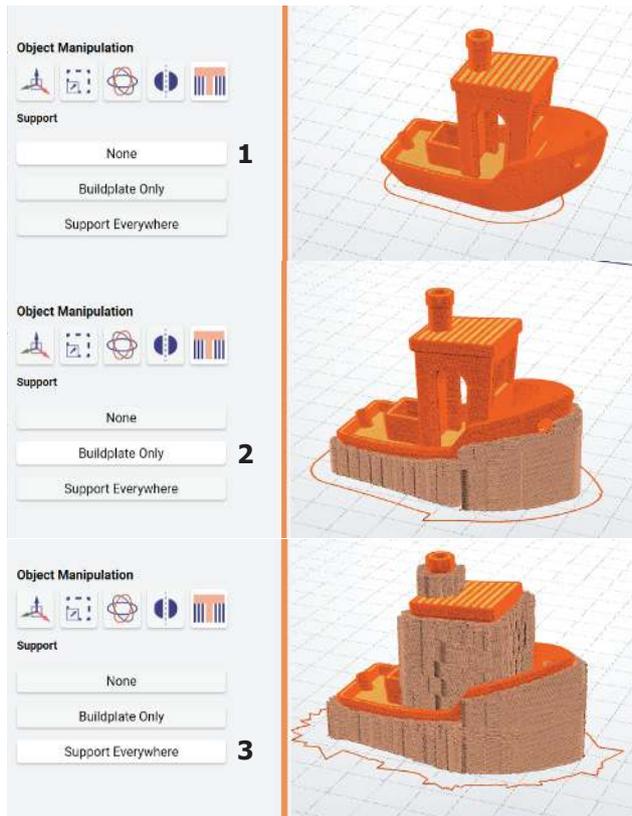
1. Choose which axis you want to flip or mirror your object along.
2. Press the reset button to undo all Flipping.



5.2.5 Support

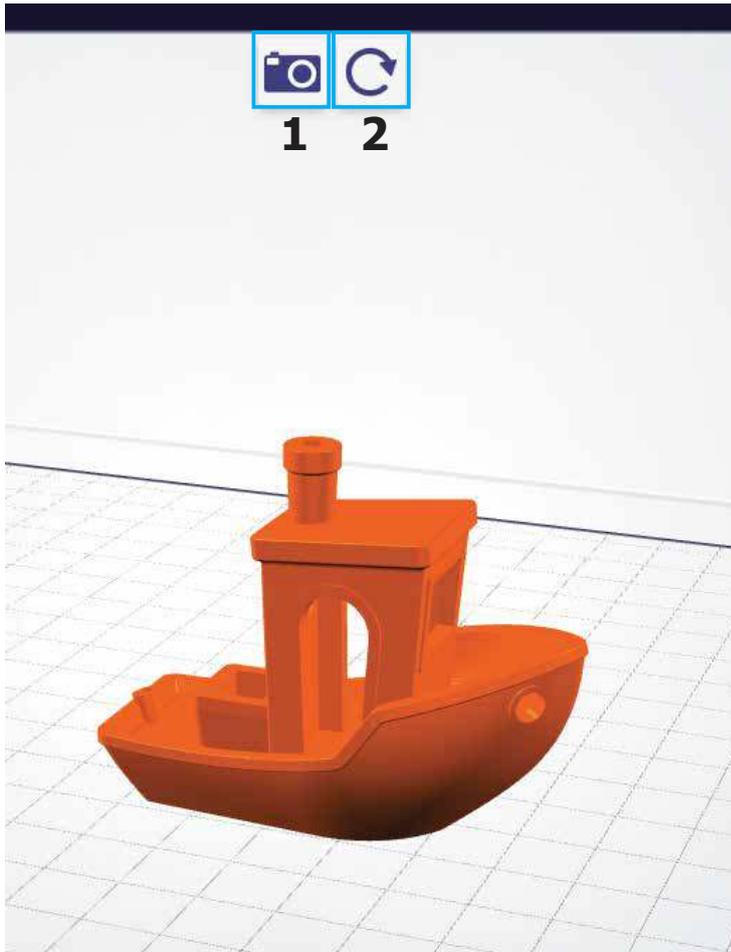
REACTOR provides three support structure options.

1. **None:** When this option is selected (which it is pr. default) no support is added.
2. **Buildplate only:** When this option is selected, support will only be added to overhangs that can be connected directly to the buildplate.
3. **Support everywhere** When this option is selected, support will be added everywhere on the model, including places where the support will be placed on the model itself.



5.3 Camera

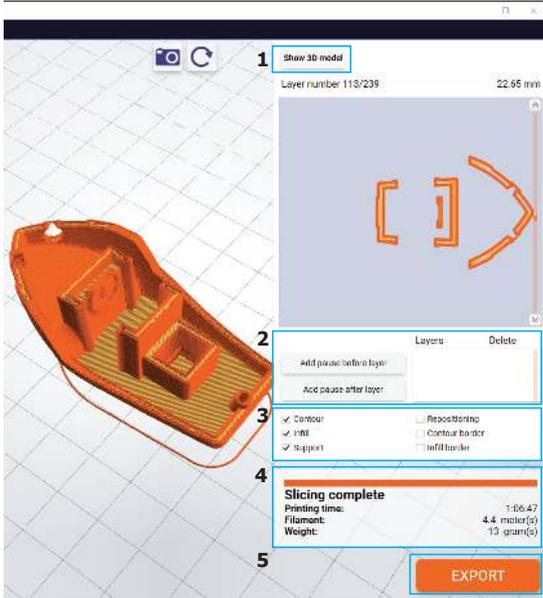
1. Resetting Camera will return the camera to both its original orientation and zoom settings.
2. Resetting the workspace will reload all .STL files in their original scale and default placement.



5.4 Preparing and showing Layers

After the Preparing is completed, the layer view will appear.

1. **Show slices:** Toggle between the 3D view of the layers and the default 3D view.
2. **Pauses:** Add a planned pause to the G-Code.
3. **Visualization:** Choose what movements you want to see visualized.
4. **Slicing info:** See information on printing time, and filament consumption.
5. **Export:** When ready, press EXPORT to save the G-Code.



Chapter 6

Terminology

The table below provides definitions for the terminology used in REACTOR.

Terminology	Explanation
Concepts	
Printer model	A specific printer, with a set of settings
Filament	Only used when referring specifically to material in "string-form" for FDM printers (such as in "filament diameter"). All other places "material".
Material	Printing materials in general, which could be referring to ABS, PLA or other names and abbreviations of materials.
Profile	A set of settings, containing all the necessary values to create a G-Code file.
Preset (Profile)	A profile defined by Fusion3, that cannot be overridden by the end user. These will be updated when the "Get newest presets" button in the "Updates" tab is pressed.
Custom (Profile)	A profile created by the end user. These will stay the same after a profile update or a software update, but the user is prompted to keep them or not.
Settings	One or more modifiable values that are used to modify slicing and thus the G-Code created.
Material Profile	The selection of a profile for a certain material.
Material Settings	Settings related to the material.
Printing Profile	Profiles related to the properties of the printed object.
Printing Settings	Settings related to the properties of the printed object.
Object	The 3D printed product made from a 3D model.
(3D) Model	A 3D model. supported 3D model formats include..STL and OBJ. Supported 3D model formats DO NOT include STP, DXG etc.
(Model) manipulations	Various changes that can be applied to a 3D model, mainly moving, scaling, rotating etc.
XY Axis	The two horizontal Axis of a 3D printer, defined by the home and the "Right hand" rule.
Z-Axis	The Vertical axis, or the "height".
Workspace (size)	The complete area or envelope that can be printed within
Buildplate	The "bottom" of the workspace. Also known as print bed.
Modifier	A model that modifies an object in the intersection between the object and the modifier.
Layer	One 3D printed path completed in the XY plane, before the Z-Height is increased.
Actions	
Importing (Profile)	Importing is taking a file containing information about the profile and loading it into the software, creating a new profile that has been shared with you by another user, or a support team member.
Exporting (Profile)	Exporting a file containing a set of settings, to send to someone who wants to replicate these.
Export (G-code or F-Code)	Saving the G-Code or F-Code file generated to a hard drive, or similar.
Adding (Profile)	Choosing which of the Profiles are accessible in the Material selection.
Preparing	When a model is transformed into a G-Code or F-Code with the chosen settings. This includes, simplification, Disc allocation, Slicing etc.

Chapter 7

Troubleshooting

In this chapter we will cover how to solve some common issues using REACTOR.

7.1 Bed adhesion

If the object does not stick to the print bed, you can try any of these fixes:

- Add brim: go to the "add brim" setting and choose either "always" or "Automatic". See Chapter 9.2.7 for more information
- Increase the temperature of the print bed. Note that this should only be done for certain materials. See Chapter 9.6.1 for more information

7.2 Underextrusion

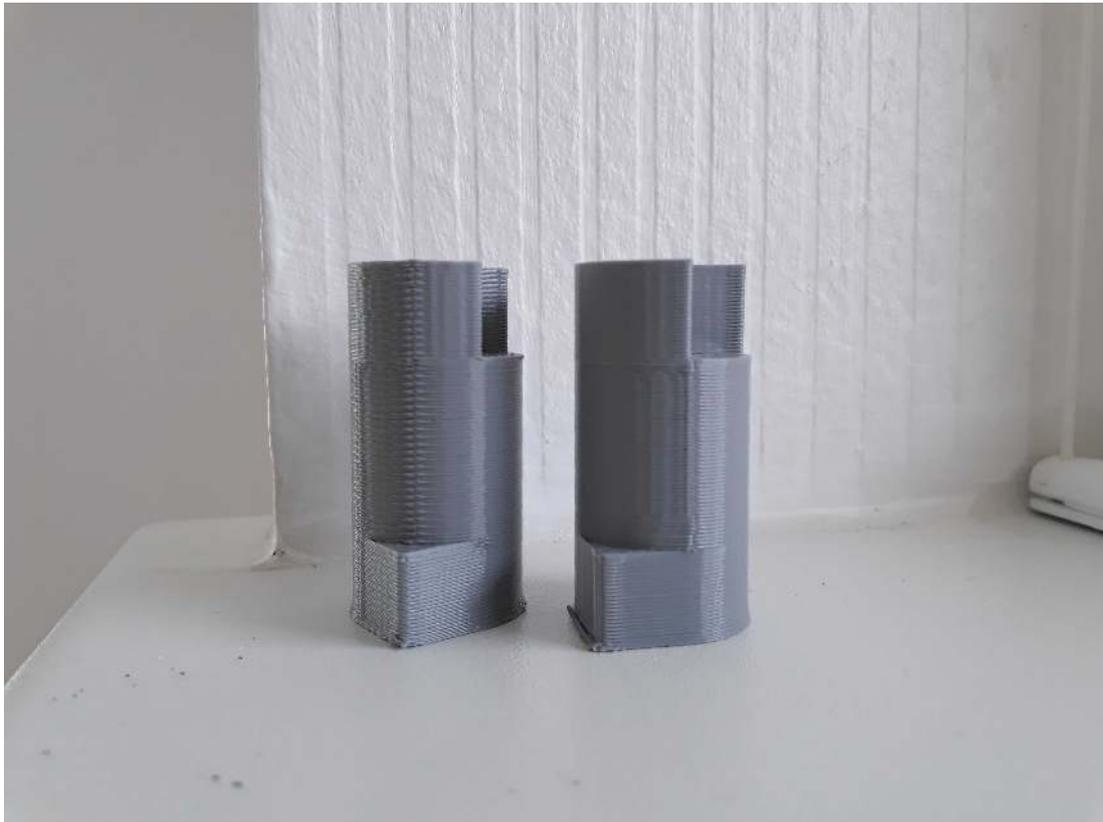
If the material does not come out of the nozzle in a smooth clean motion, it can have serious consequences for the print quality and object strength.

- Increase the temperature of the print head. See Chapter 9.6.1 for more information
- Make sure that you are using a certified material, and that the material is loaded correctly.

7.3 Zippers

Since a 3D printed object is created one layer at a time, the printer will have to move upwards after completing each layer. This can create a small defect known as a "Seam" or a "Zipper". REACTOR contains settings to manage how this transition happens, to minimize the appearance of this defect.

If the option "hide zippers" is enabled, REACTOR will attempt to put all the small defects in the same place, and hide them in convex or concave angles. If no angles are found that are concave or convex enough, for instance on a cylinder, the zippers will be placed on the back of the model. For certain objects it may be desirable to have the defects be spread around the model, in which case you can disable the "Hide zippers" option. In this case the defects will be placed arbitrarily. See an example below where the left model has hide zippers disabled, and the model on the right has hide zippers enabled. See Chapter 9.2.4 for more information



For more advanced control of the Zippers see: Chapter 9.4.15 for more information.

Chapter 8

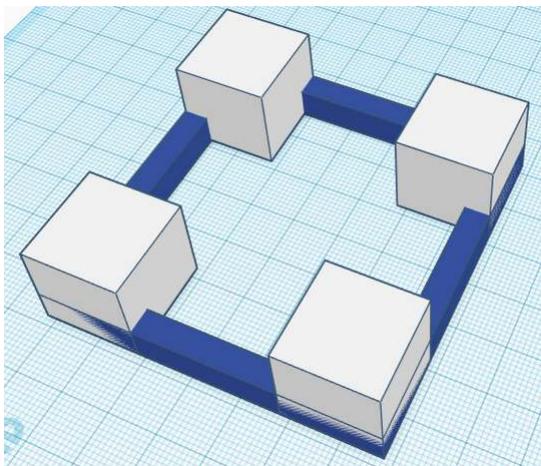
Advanced Topics

8.1 Modifier objects

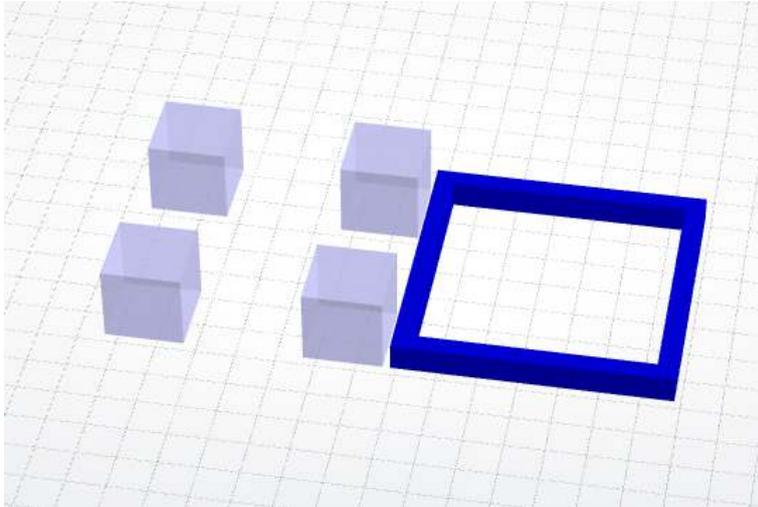
Modifier objects allow for changing settings of an object in certain areas, using one or more .STL files to modify another. This has many interesting fields of use, including weight optimization, print time reduction, strength optimization and more.

The recommended workflow for using this feature, is to use a CAD tool to draw zones where you would like the properties to be different. Below will follow an example of how to use this feature for creating a frame that has a higher infill density in the corners:

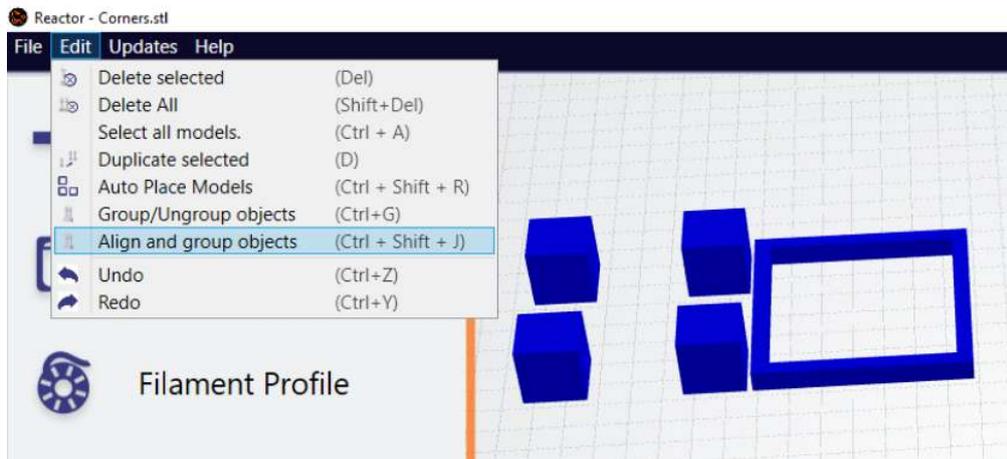
Step 1: In the picture below, a frame is designed (shown in blue) with four boxes in the corners (shown in white). The four corners are grouped together, and the frame and the corners are exported as individual .STL files.



Step 2: In the picture below the two .STL files are seen loaded into REACTOR.



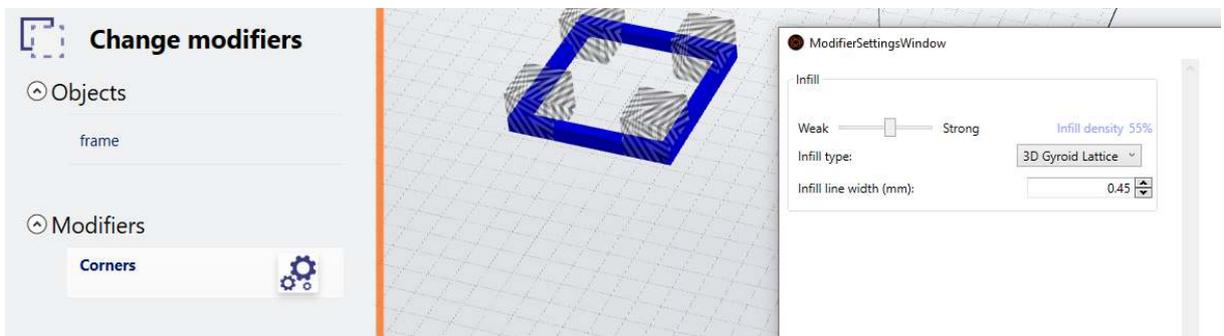
Step 3: Select both models by holding down SHIFT and left clicking them. Afterwards align and group them using the function in the "edit" tab, or the shortcut CTRL + SHIFT + j.



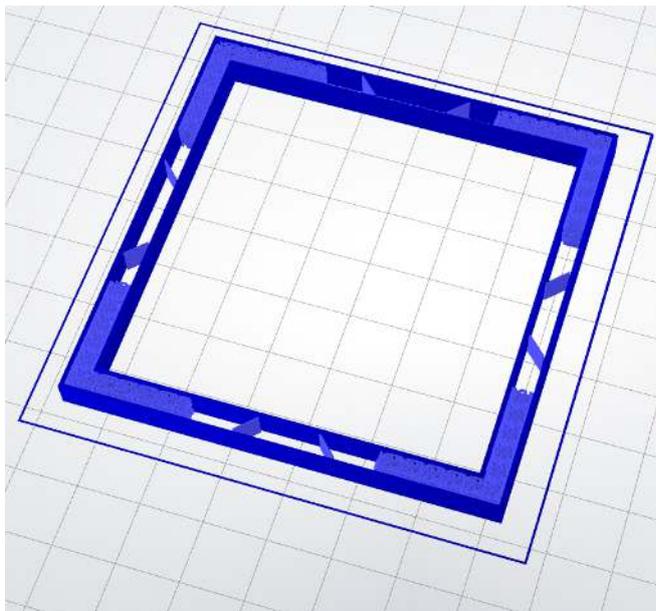
Step 4: Drag'n'drop the 3D model from the object area, into the modifier area. Now the modifier menu will become available for the modifier object.



Step 5: You now have access to the modifier menu, where you can change the settings for the modifier. This will change the settings in all the places where the modifier overlaps the model.



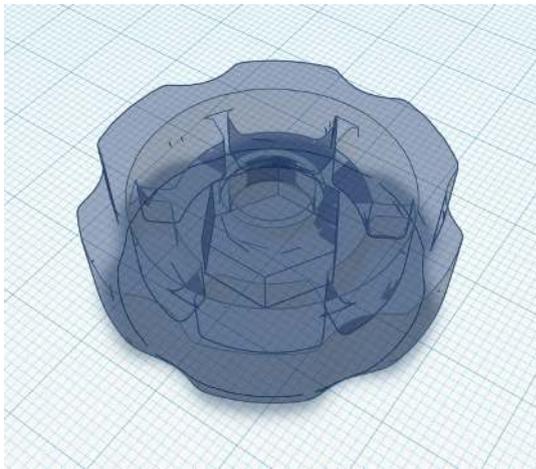
Step 6: Below the result can be seen, where the corners have a 55% infill density, whereas the rest of the model has 5%.



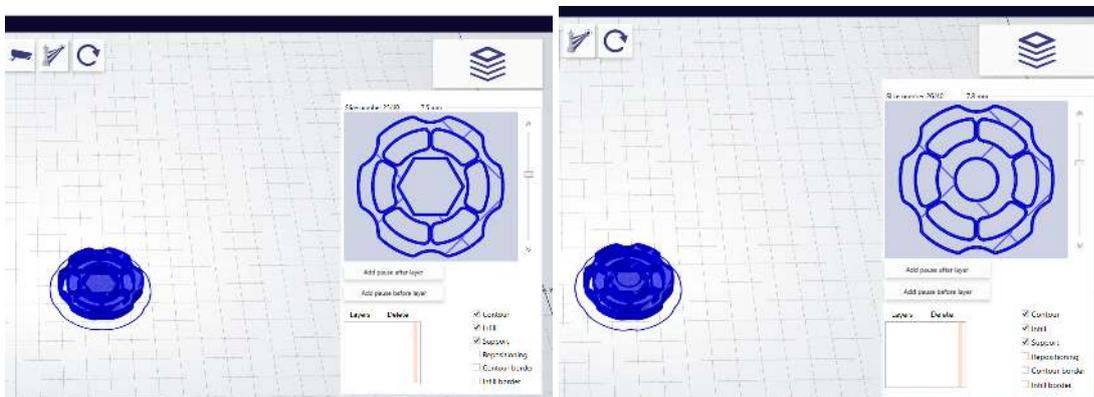
8.2 Planned Pauses

REACTOR includes a feature to insert planned pauses into your print process. This can be used to create a part that changes color after a certain layer, or to embed various objects into a 3D printed object. Here follows an example of how to embed a nut into a 3D printed object.

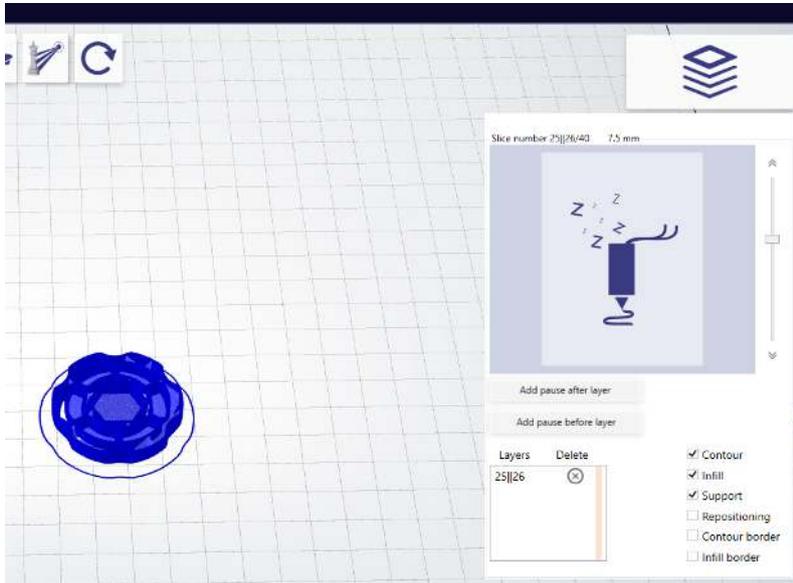
Step 1: Designing the 3D object When designing the CAD model you need to make space for the nut inside.



Step 2: Preparing model in REACTOR. Load the .STL file into REACTOR and prepare it. Now open the slices menu, and navigate to the layer where the nut will be completely embedded into the object. On the pictures it can be seen that the pause needs to be between layer 25 and 26. Press either the "Add pause after layer" or "Add pause before layer" - button. Now a pause has been added, and your printer will pause at this place.



Step 3: Printing. When the pause appears in the menu as seen in the picture below, you are ready to start the print. The printer will automatically pause between these two layers and prompt the user to take action.



Step 4: Finished product. When the product is finished, a bolt can be inserted into the nut.

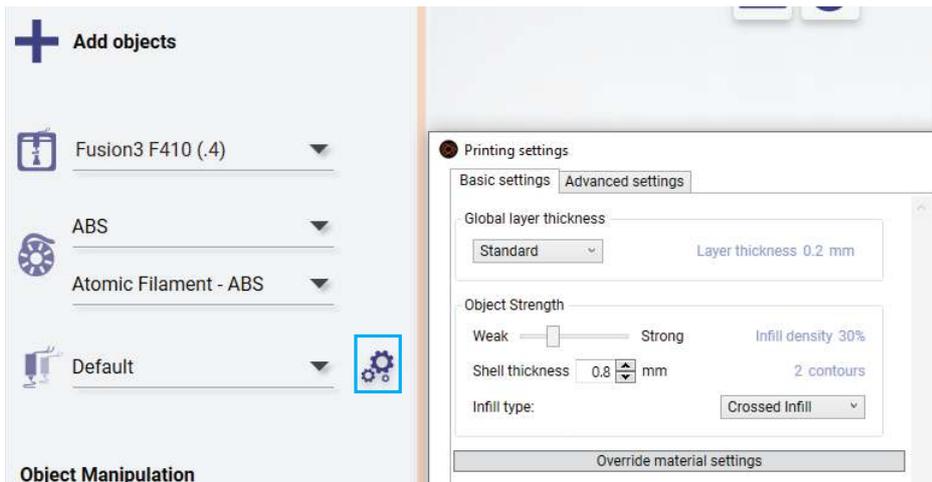


Chapter 9

Index of settings

9.1 Quality Settings: Basic

To open the Quality settings, press the settings button as showed below:



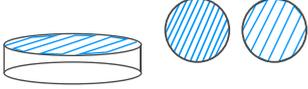
9.1.1 Layer Resolution

Placement	Setting Name	Illustration
Basic	Layer resolution	

Layer thickness (mm):

- Determines the height of each layer.
- Thinner layers can improve the surface quality of detailed objects, but will increase the print time.

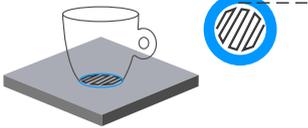
9.1.2 Infill Density

Placement	Setting Name	Illustration
Basic	Infill density	

Infill density:

- Higher infill will result in a stronger object, at the cost of material usage and print time.
- 100% will result in a solid object, and 0% will result in a hollow object.

9.1.3 Shell thickness

Placement	Setting Name	Illustration
Basic	Shell thickness	

Shell thickness:

- Defines the thickness of the solid outer shell of the object.
- It is defined as a multiple of the line width.
- A higher value will result in a stronger object at the cost of material usage and print time.

9.1.4 Infill type

Placement	Setting Name	Illustration
Basic	infill type	

REACTOR includes several different **Infill types** listed below. Note that some of them are 3D infills. These are geometries that are generated to be different for each layer. These have different interesting properties, for instance the Gyroids and diamond infills provide a high tensile strength while the Isomax provides a stiff geometry.

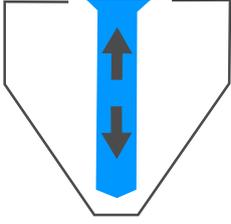
- Linear infill
- Crossed infill
- 3D Diamond
- 3D Double Gyroid
- 3D Gyroid lattice
- 3D Gyroid Skeleton
- 3D Isomax
- 3D Truncated Octahedron
- Fish
- Honeycomb
- Moroccan Star
- Octagonal
- Triangular

Find on the next page an overview of the properties of the 3D infills:

3D infill	Properties
3D diamond	<ul style="list-style-type: none"> - Evens out mechanical strength on all directions - Reduces print time by up to 10% for infill densities smaller than 15% * - Reduces object weight and material use - Surface area and channel connectivity properties can be used for a variety of applications: from tissue scaffolds, biofilters, metamaterials for thermal and sound insulation or photonic manipulation <p>https://www.sciencedirect.com/science/article/pii/S0032386117311175</p>
Double Gyroid	<ul style="list-style-type: none"> - Evens out mechanical strength on all directions - Large surface area - Surface area and channel connectivity properties can be used for a variety of applications: from tissue scaffolds, biofilters, metamaterials for thermal and sound insulation or photonic manipulation <p>https://www.sciencedirect.com/science/article/pii/S0032386117311175</p>
Gyroid lattice	<p>This structure optimizes object strength for stress in all directions. This keeps the object weight minimal and printing speed high.</p> <ul style="list-style-type: none"> - Reduces material use and object weight - Evens out mechanical strength on all directions - Increases mechanical strength - High degree of channel interconnectivity - Surface section has no straight lines - No planes of symmetry - Surface area and channel connectivity properties can be used for a variety of applications: from tissue scaffolds, biofilters, metamaterials for thermal and sound insulation or photonic manipulation <p>https://www.sciencedirect.com/science/article/pii/S0032386117311175</p>
Gyroid skeleton	<ul style="list-style-type: none"> - Evens out mechanical strength on all directions - Reduces object weight and material use - Surface area and channel connectivity properties can be used for a variety of applications: from tissue scaffolds, biofilters, metamaterials for thermal and sound insulation or photonic manipulation
3D isomax	<p>This structure optimizes object toughness and crush-resistance by using a structure that reaches the theoretical limits of elastic stiffness while also having a very low weight. The isomax also evens out strength in all directions.</p> <ul style="list-style-type: none"> - Big increase of mechanical strength - Evens out mechanical strength on all directions <p>https://www.nature.com/articles/s41467-020-15434-2</p>
3D truncated Octahedron	<p>A type of 3D honeycomb structure. Similar to the gyroid and the isomax, the truncated octahedron evens out mechanical strength in objects.</p>

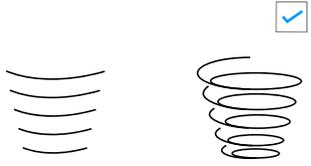
9.2 Quality Settings: Advanced

9.2.1 Retraction enable/disable

Placement	Setting Name	Illustration
Advanced	Retraction enable/disable	

Retraction is a feature to reduce dripping of excess plastic during repositioning. Before repositioning, the extruder sucks back the material to avoid drops. By the end of the repositioning, the material is pushed back to its original position and the printing continues as normal.

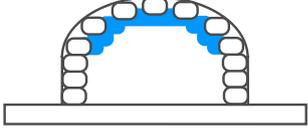
9.2.2 Continuous Z

Placement	Setting Name	Illustration
Advanced	Continuous Z	

Continuous Z:

- Continuous movement on Z axis during the entire print can increase wall quality by smoothing out the layer change if there are no repositioning within a layer.
- Use with care. This setting can create uneven layers and bad quality for complex objects and objects with repositioning within a layer.
- This setting will have very little effect on objects printed with infill and more than one contour.

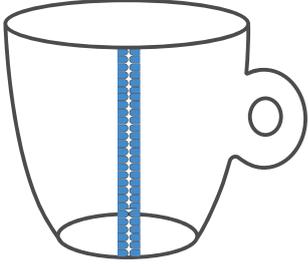
9.2.3 3D shell

Placement	Setting Name	Illustration
Advanced	3D shell	 A diagram showing a semi-circular object with a 3D shell. The shell is represented by a blue, semi-circular area with a grid pattern, indicating internal infill. The object is shown sitting on a flat rectangular base.

3D Shell:

- Creates a 3D shell within the object improving object's strength, helping closing open areas with visible infill and allowing better free forming ("building in the air").

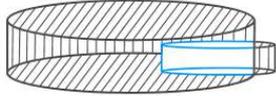
9.2.4 Hide zippers

Placement	Setting Name	Illustration
Advanced	Hide zippers	 A diagram of a cup with a handle. A vertical blue line runs down the center of the cup, representing a zipper. The line is composed of small '+' symbols, indicating the location of zippers.

Hide zippers:

The very start of a layer will often have very small defects called zippers. When this checkbox is checked, these defects will be hidden if possible.

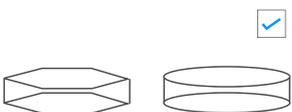
9.2.5 Preserve internal geometry

Placement	Setting Name	Illustration
Advanced	Preserve internal geometry	

Preserve internal geometry:

- Many 3D models are created as a combination of smaller objects intersecting each other. This results in unwanted internal geometries that are automatically removed during slicing.
- Use this option if you want to preserve the inside of geometry the models. For instance, if you have an 3D model that is designed to be a hollow object, but the inside is automatically filled during slicing.

9.2.6 Disable simplification

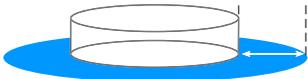
Placement	Setting Name	Illustration
Advanced	Disable simplification	

Disable simplification:

To decrease slicing time 3D models are simplified when opened.

Disable simplification if you experience that you have lost details on your object after slicing. Note that this will increase the slicing time significantly.

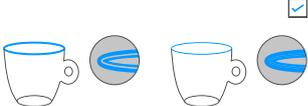
9.2.7 Print brim

Placement	Setting Name	Illustration
Advanced	Print brim	

Print brim:

- This builds an extra wide bottom layer by adding extra, easily removable contours.
- The maximum number of contours that fit inside this distance, will be generated.
- This is useful if the object has trouble sticking to the bed.
- If "Automatically" is selected, the shapes of the objects determine the brim settings.

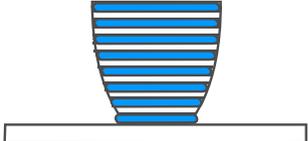
9.2.8 Fill thin walls

Placement	Setting Name	Illustration
Advanced	Fill thin walls	

Fill thin walls: Will attempt to fill up thin parts of the object that is normally too small to print. In some cases this can make parts of the print slightly thicker than expected. There are three options:

- None: Areas will only have infill if there is sufficient room for the filament. Use this if you have strict requirements for the dimension tolerances of your object
- Only visible: Only visible areas will be allowed to be filled extra up.
- All: The whole print is allowed to be filled extra up.

9.2.9 Infill height

Placement	Setting Name	Illustration
Advanced	Infill frequency	

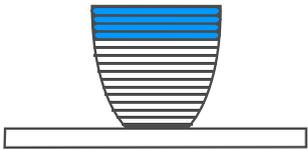
Infill height (mm):

Defines the thickness for infill and the distance between layers with infill. Full infill areas are not affected

and will be printed on each layer.

- This allows faster prints, by only printing infill on some layer, without decreasing the quality of visible areas.
- If the distance is the same as the layer thickness, there is infill on every layer.
- The "Infill frequency" value has an upper limit of $1.5 * \text{"Nozzle diameter"}$.

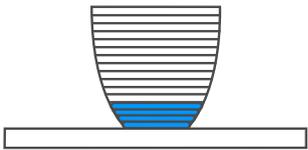
9.2.10 Solid layers: Top thickness

Placement	Setting Name	Illustration
Advanced	Solid layers: Top thickness	

Top thickness:

- Determines the amount of solid infill layers at the top shell of the object.
- Increasing the top thickness can improve the quality and strength of objects at the cost of material usage and print time.

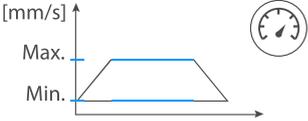
9.2.11 Solid layers Bottom thickness

Placement	Setting Name	Illustration
Advanced	Solid layers Bottom thickness	

Bottom thickness:

- Determines the amount of solid infill layers at the bottom shell of the object.
- Increasing the bottom thickness can improve the quality and strength of objects at the cost of material usage and print time.

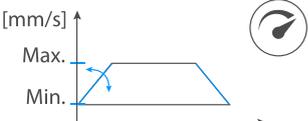
9.2.12 Visible areas: Speed

Placement	Setting Name	Illustration
Advanced	Visible Areas Speed	 <p>The illustration shows a graph with the vertical axis labeled '[mm/s]'. The graph depicts a trapezoidal speed profile, starting at a 'Min.' value, rising to a 'Max.' value, staying constant for a period, and then falling back to the 'Min.' value. To the right of the graph is a circular speedometer icon with a needle.</p>

Visible area speed (mm/s):

- The target maximum printing speed for visible areas.

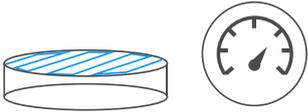
9.2.13 Visible areas: Acceleration rate

Placement	Setting Name	Illustration
Advanced	Visible Areas Acceleration rate	 <p>The illustration shows a graph with the vertical axis labeled '[mm/s]'. The graph depicts a trapezoidal speed profile, starting at a 'Min.' value, rising to a 'Max.' value, staying constant for a period, and then falling back to the 'Min.' value. Blue arrows indicate the slope of the rising and falling edges. To the right of the graph is a circular speedometer icon with a needle.</p>

Acceleration rate for printing visible areas (mm/s²):

- Defines the rate with which the speed changes during printing of the visible areas.

9.2.14 Inner areas: Speed

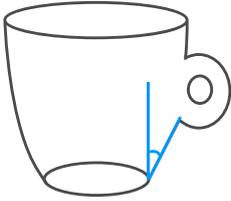
Placement	Setting Name	Illustration
Advanced	Inner areas: Speed	 <p>The illustration shows a 3D perspective view of a cylindrical part with blue diagonal hatching on its top surface. To the right of the cylinder is a circular speedometer icon with a needle.</p>

Inner area speed factor (mm/s):

- Determines the target speed while printing inner areas.

- Lowering this value may improve the quality of the object, at the expense of increasing the printing time.
- Increasing this value will decrease the printing time, but may result in bad quality.
- If a red "!" appears next to the value, the target speed get limited by the maximum extrusion. This means that the speed will not be reached, to avoid under-extrusion.

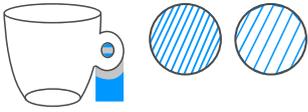
9.2.15 Support: Overhang angle

Placement	Setting Name	Illustration
Advanced	Support: Overhang angle	

Overhang angle (degrees):

- Maximum allowed angle for overhangs without building support.
- 0° = vertical, 90° = horizontal line with respect to the build platform.
- Check the "Show layers" window, to see if the support is applied at the desired places.

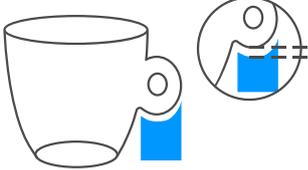
9.2.16 Support: infill density

Placement	Setting Name	Illustration
Advanced	Support: infill density	

Support Density:

- Density of of the support.
- 100% will print solid support layers, 0% will not make any infill.
- High support densities will create a more stable support, which will give better support under overhangs, but will be more difficult to remove and increase the print time.

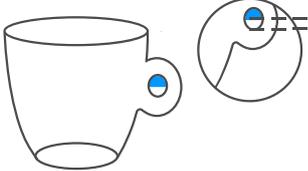
9.2.17 Support: Z-Offset top

Placement	Setting Name	Illustration
Advanced	Support: Z-Offset top	

Z-offset (mm):

- Distance between the support and the printed object on the Z axis.

9.2.18 Support: Z offset bottom

Placement	Setting Name	Illustration
Advanced	Support: Z offset bottom	

Z-offset (mm):

- Distance between the support and the printed object on the Z axis.

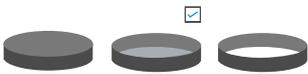
9.2.19 Object type Standard

Placement	Setting Name	Illustration
Advanced	Object type Standard	

Object Types :

- Closed (Left): The object will print as shown in the workspace.
- Vase (Center): The object will print without infill and without solid top layers. Should be used if the model has a solid top and infill.
- Bracelet (Right): The object will print without infill, solid top and bottom layers. Should be used if the model has a solid top and bottom and infill.

9.2.20 Object type Vase

Placement	Setting Name	Illustration
Advanced	Object type Vase	

Object Types :

- Closed (Left): The object will print as shown in the workspace.
- Vase (Center): The object will print without infill and without solid top layers. Should be used if the model has a solid top and infill.
- Bracelet (Right): The object will print without infill, solid top and bottom layers. Should be used if the model has a solid top and bottom and infill.

9.2.21 Object type Bracelet

Placement	Setting Name	Illustration
Advanced	Object type Bracelet	

Object Types :

- Closed (Left): The object will print as shown in the workspace.
- Vase (Center): The object will print without infill and without solid top layers. Should be used if the model has a solid top and infill.
- Bracelet (Right): The object will print without infill, solid top and bottom layers. Should be used if the model has a solid top and bottom and infill.

9.3 Quality Settings: Expert - Printer

9.3.1 Printing speed

Placement	Setting Name	Illustration
Expert Printer	Printing speed	<p>The illustration shows a graph with a vertical axis labeled '[mm/s]' and a horizontal axis. Two horizontal lines represent 'Max.' and 'Min.' speed levels. A blue trapezoidal shape is drawn between these two lines, representing a speed profile that ramps up to a maximum, stays constant, and then ramps down. To the right of the graph is a circular speedometer icon with a needle.</p>

Maximum printing speed (mm/s):

- Defines the target printing speed.
- The maximum value of the "Speed Factor" sliders, the Advanced tab are defined by this setting.
- When the "Speed Mode" slider in the Advanced tab are disabled, this value is the target maximum printing speed for inner areas. Disable these sliders with the checkbox below.
- Note that this value might not be reached if Maximum extrusion is too low.

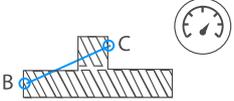
9.3.2 Visible area speed

Placement	Setting Name	Illustration
Expert Printer	Visible area speed	<p>The illustration shows a graph with a vertical axis labeled '[mm/s]' and a horizontal axis. Two horizontal lines represent 'Max.' and 'Min.' speed levels. A blue trapezoidal shape is drawn between these two lines, representing a speed profile that ramps up to a maximum, stays constant, and then ramps down. To the right of the graph is a circular speedometer icon with a needle.</p>

Visible area speed (mm/s):

- The target maximum printing speed for visible areas.

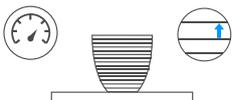
9.3.3 Repositioning speed

Placement	Setting Name	Illustration
Expert Printer	repositioning speed	

Repositioning speed (mm/s):

- The target maximum speed for repositioning between printing paths.

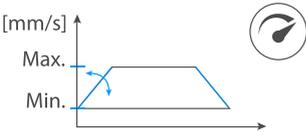
9.3.4 Z-Only repositioning speed

Placement	Setting Name	Illustration
Expert Printer	Z-Only repositioning speed	

Repositioning speed (mm/s):

- The target maximum speed for repositioning between printing paths.

9.3.5 Enable impulse and acceleration

Placement	Setting Name	Illustration
Expert	Enable impulse and acceleration	

Enable Impulse and Acceleration control in the G-Code file:

If checked, REACTOR will generate acceleration and impulse G-Codes.

If left unchecked, the printer firmware will use its own acceleration and jerk values. In this case print time estimation may be inaccurate.

9.3.6 Acceleration rate: Printing

Placement	Setting Name	Illustration
Expert Printer	Acceleration rate: Printing speed	

Acceleration rate for printing (mm/s^2):

- Defines the rate with which the speed changes during printing of the inner areas.

9.3.7 Acceleration rate: Visible areas

Placement	Setting Name	Illustration
Expert Printer	Acceleration rate: Visible areas	

Acceleration rate for printing visible areas (mm/s^2):

- Defines the rate with which the speed changes during printing of the visible areas.

9.3.8 Acceleration rate: Repositioning

Placement	Setting Name	Illustration
Expert Printer	Acceleration rate: Repositioning	

Acceleration Rate & Deceleration Rate for repositioning (mm/s²):

- Specifies the speed change during repositioning.

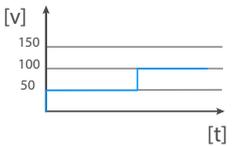
9.3.9 Acceleration rate: Z

Placement	Setting Name	Illustration
Expert Printer	Acceleration rate: Z	

Acceleration Rate & Deceleration Rate for repositioning (mm/s²):

- Specifies the speed change during repositioning.

9.3.10 Impulse: visible area

Placement	Setting Name	Illustration
Expert Printer	Impulse: visible area	

Impulse speed (mm/s):

- The allowed jump in speed on a motor axis.
- This value is used to assign speed limits on path corners.
- All movements have initial speeds based on the impulse speed of the moving axes and the direction of the first movement.
- Too high impulse speed values can cause ringing defects on the print surface.

9.3.11 Impulse Other

Placement	Setting Name	Illustration
Expert Printer	Impulse Other	

Impulse speed (mm/s):

- The allowed jump in speed on a motor axis.
- This value is used to assign speed limits on path corners.
- All movements have initial speeds based on the impulse speed of the moving axes and the direction of the first movement.
- Too high impulse speed values can cause ringing defects on the print surface.

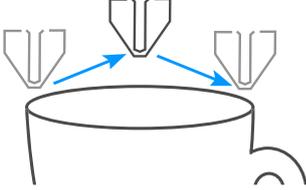
9.3.12 Impulse: Impulse Z-Axis

Placement	Setting Name	Illustration
Expert Printer	Impulse: Impulse Z-Axis	

Impulse speed (mm/s):

- The allowed jump in speed on a motor axis.
- This value is used to assign speed limits on path corners.
- All movements have initial speeds based on the impulse speed of the moving axes and the direction of the first movement.
- Too high impulse speed values can cause ringing defects on the print surface.

9.3.13 Use repositioning Hop

Placement	Setting Name	Illustration
Expert Printer	Use repositioning Hop	 A diagram showing a cylindrical printing bed with a nozzle assembly above it. Three blue arrows point from the nozzle assembly to the bed's surface, indicating the movement of the nozzle during a repositioning hop.

Repositioning hop:

- During repositioning, the head will move away from the bed.
- This feature is used to avoid leaving marks when crossing contours.

9.4 Quality Settings: Expert - Layers

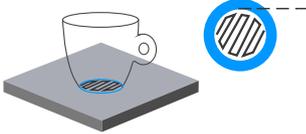
9.4.1 Layer Thickness

Placement	Setting Name	Illustration
Expert Layers	slice thickness	 Two 3D models of a cylindrical object on a grey base. The left model shows a stack of fewer, thicker layers, while the right model shows a stack of more, thinner layers, illustrating the effect of slice thickness.

Layer thickness (mm):

- Determines the height of each layer.
- Thinner layers can improve the surface quality of detailed objects, but will increase the print time.
- To enable this value, select "Custom" as Layer resolution in the Basic tab.

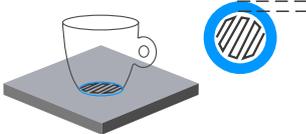
9.4.2 Number of contours

Placement	Setting Name	Illustration
Expert Layers	Shell thickness	

Number of contours:

- Number of lines in a wall.
- Increasing the number of contours can improve strength and print quality.

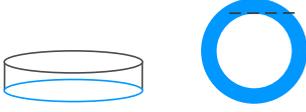
9.4.3 Contour width

Placement	Setting Name	Illustration
Expert Layers	contour width	

Contour width:

- The width of the printed lines for contours.
- This value changes automatically when the value of the setting "Nozzle diameter" is changed.

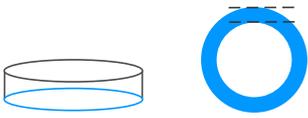
9.4.4 Contour width smallest detectable

Placement	Setting Name	Illustration
Expert Layers	contour width smallest detectable	

Contour width smallest printable/detectable:

- When printing very small details that cannot be printed with regular contour line width, the smallest detectable line width is applied.
- These details are printed with the smallest printable contour line width.
- The smallest printable contour width should be bigger than the smallest detectable contour width to avoid underextrusion for very slow extrusion.

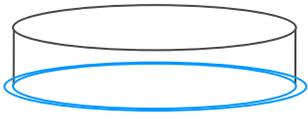
9.4.5 Contour width smallest printable

Placement	Setting Name	Illustration
Expert Layers	contour width smallest printable	

Contour width smallest printable/detectable:

- When printing very small details that cannot be printed with regular contour line width, the smallest detectable line width is applied.
- These details are printed with the smallest printable contour line width.
- The smallest printable contour width should be bigger than the smallest detectable contour width to avoid underextrusion for very slow extrusion.

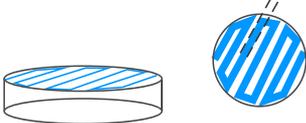
9.4.6 Contour overlap

Placement	Setting Name	Illustration
Expert Layers	Contour overlap	

Contour overlap:

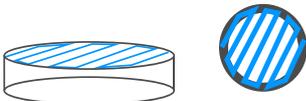
- How much of the contour overlaps the neighboring contours.
- 100% means that half of the contour overlaps the contour to the right and the other half overlaps the contour to the left.
- 0% means that the contours don't overlap each other.

9.4.7 Infill line width

Placement	Setting Name	Illustration
Expert Layers	Infill line width	

The width of the printed filament line for infill. This value changes automatically when Nozzle diameter (9.12.1) is changed.

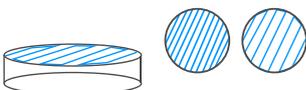
9.4.8 Infill overlap

Placement	Setting Name	Illustration
Expert Layers	Infill overlap	

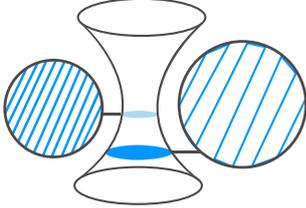
Determines how much the infill lines may overlap the inner contour. 50% means that the the infill may overlap half of the width of the inner contour. If the contours are not completely attached to the infill, increase this value. This can sometimes be necessary when using flexible filaments.

9.4.9 Infill density (same as basic)

Same as Section 9.1.2

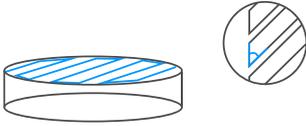
Placement	Setting Name	Illustration
Expert Layers	Infill density (same as basic)	

9.4.10 Solid infill threshold area

Placement	Setting Name	Illustration
Expert Layers	solid infill threshold area	

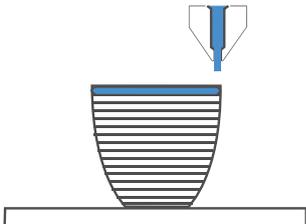
Areas smaller than this value will be using 100% infill density to ensure rigidity of the object.

9.4.11 infill angle

Placement	Setting Name	Illustration
Expert Layers	infill angle	

Determines the angle the infill lines. The angle only affects Crossed and Linear infill. Typical infill values are 45 or 0 degrees.

9.4.12 Visible infill extrusion ratio

Placement	Setting Name	Illustration
Expert Layers	Visible infill extrusion ratio	

Determines the extrusion rate for visible infill lines such as local top layers. Use this to compensate for cases when there is visible space between the infill lines off visible areas.

9.4.13 Generate automatic infill connections

Placement	Setting Name	Illustration
Expert Layers	Generate automatic infill connections	None

If enabled, REACTOR will try to make sure infill lines are connected at the ends. If disabled, you can specify the length of additional infill before and after infill lines. These will follow the shape of the contour. The distance of these connecting lines can be defined in the field "Leading/trailing infill line length" 9.4.14.

9.4.14 Leading/trailing infill line width

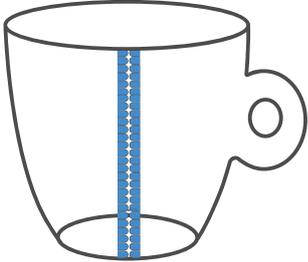
This field is only available if "Generate automatic infill connections" 9.4.13 is disabled, and in that case gives the ability to control a trail before the infill line.

Placement	Setting Name	Illustration
Expert Layers	Leading/trailing infill line width	None

Specifies the distance that infill should not go above when trying to connect infill lines. If the specified length is not possible to achieve, the closest possible length will be used.

Set this to 0 to remove all infill connections.

9.4.15 Hide zippers

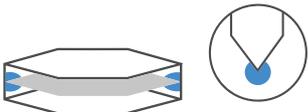
Placement	Setting Name	Illustration
Expert Layers	Hide zippers	

Hide Zippers:

Hiding the zippers will be prioritized in the order they appear:

- 1) Hide them inside concave sharp angles if possible. A concave angle points into the object. Specify the biggest acceptable angle in degrees.
- 2) Hide them inside convex sharp angles if possible. A convex angle points out of the object. Specify the acceptable interval of angles in degrees.
- 3) Put them on the back of the object.

9.4.16 Hide in concave angles

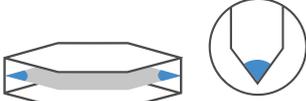
Placement	Setting Name	Illustration
Expert Layers	Hide in concave angles	

Hide Zippers:

Hiding the zippers will be prioritized in the order they appear:

- 1) Hide them inside concave sharp angles if possible. A concave angle points into the object. Specify the biggest acceptable angle in degrees.
- 2) Hide them inside convex sharp angles if possible. A convex angle points out of the object. Specify the acceptable interval of angles in degrees.
- 3) Put them on the back of the object.

9.4.17 Hide in convex angles

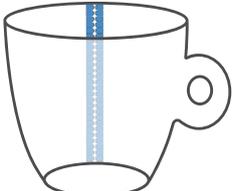
Placement	Setting Name	Illustration
Expert Layers	Hide in convex angles	

Hide Zippers:

Hiding the zippers will be prioritized in the order they appear:

- 1) Hide them inside concave sharp angles if possible. A concave angle points into the object. Specify the biggest acceptable angle in degrees.
- 2) Hide them inside convex sharp angles if possible. A convex angle points out of the object. Specify the acceptable interval of angles in degrees.
- 3) Put them on the back of the object.

9.4.18 Hide on the back

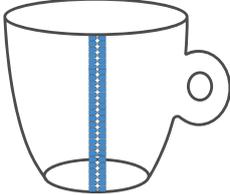
Placement	Setting Name	Illustration
Expert Layers	Hide on the back	

Hide Zippers:

Hiding the zippers will be prioritized in the order they appear:

- 1) Hide them inside concave sharp angles if possible. A concave angle points into the object. Specify the biggest acceptable angle in degrees.
- 2) Hide them inside convex sharp angles if possible. A convex angle points out of the object. Specify the acceptable interval of angles in degrees.
- 3) Put them on the back of the object.

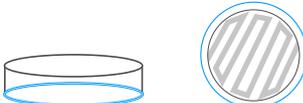
9.4.19 Stitch Seam

Placement	Setting Name	Illustration
Expert Layers	Stich Seam	

Stitch seams:

If enabled, the print head will move slightly into the object after finishing the contour in order to fuse the start and end of the path.

9.4.20 Contour printing strategy

Placement	Setting Name	Illustration
Expert Layers	Contour printing strategy	

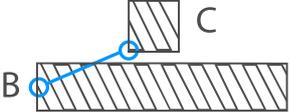
Contour printing strategy:

Control the printing order of contours. The options are:

- Inner first - this will print the inner contours before the outer contours
- Outer first - this will print the outer contours, and then the inner

Note: Printing outside contours first can decrease print quality on the outer surfaces, especially on areas with overhangs.

9.4.21 Repositioning strategy

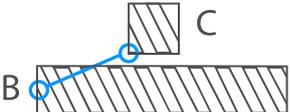
Placement	Setting Name	Illustration
Expert Layers	Repositioning strategy	

Repositioning strategy:

Define which path finding strategy is used during slicing of the model(s)

- Support first - A strategy which ensures that support structures are printed first on every layer
- Closest distance - A strategy which looks for the closest distance to the printhead

9.4.22 Avoid printed parts

Placement	Setting Name	Illustration
Expert Layers	Avoid printed parts, Avoid distance	

"Avoidance distance" is only available if "Avoid printed parts" 9.4.22 is enabled.

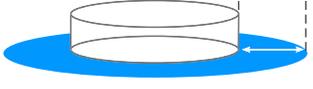
Avoid printed parts:

Avoid printed parts when repositioning between areas by moving around them.

Avoid distance (mm):

- The distance from the printed parts to the repositioning path.
- This value changes automatically when the value of the setting "Nozzle diameter" is changed.

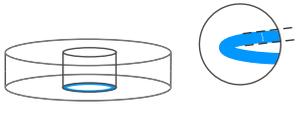
9.4.23 Brim offset

Placement	Setting Name	Illustration
Expert Layers	Brim offset	

Print brim:

- This builds an extra wide bottom layer by adding extra, easily removable contours.
- The maximum number of contours that fit inside this distance, will be generated.
- This is useful if the object has trouble sticking to the bed.
- If "Automatically" is selected, the shapes of the objects determine the brim settings.

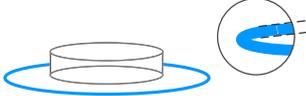
9.4.24 Print internal brim with offset

Placement	Setting Name	Illustration
Expert Layers	Print internal brim with offset.	

Print internal brim:

- Adds a brim inside holes of objects and objects inside holes.
- The maximum number of contours that fit inside this distance, will be generated.
- This is useful if the printed object have trouble sticking to the bed.
- Brim width is defined as the distance from the edge of the object to the edge of the brim.
- Internal brim isn't illustrated in the workspace. Use show layers to check if the brim is as intended.
- Using internal brim with objects that require internal support may cause overlap.

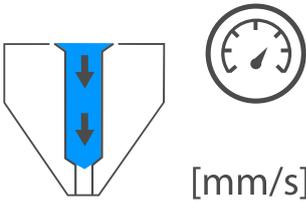
9.4.25 Print skirt with Offset

Placement		Setting Name	Illustration
Expert	Layers	Print skirt with Offset	

Print skirt:

- Build an offset line of material around the object on the first layer to ensure that the printhead is ready to print.
- Printing a skirt can improve bed adhesion.

9.4.26 Initial push-out speed

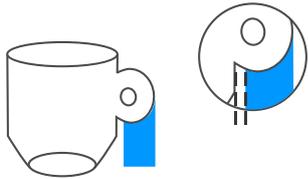
Placement		Setting Name	Illustration
Material	Advanced	Initial push-out speed	

Initial push-out speed:

- Speed of the initial push-out
- This is also the speed used for manual extrusion in the Printing window.

9.5 Quality Settings: Expert - Support

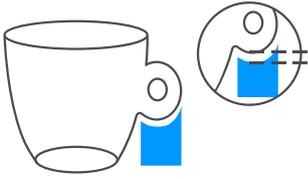
9.5.1 XY offset

Placement	Setting Name	Illustration
Expert Support	XY offset	

XY offset (mm):

- Distance between the support and the printed object on X and Y axes

9.5.2 Z-Offset top layer

Placement	Setting Name	Illustration
Expert Support	Z-Offset top layer	

Z-offset (mm):

- Distance between the support and the printed object on the Z axis.

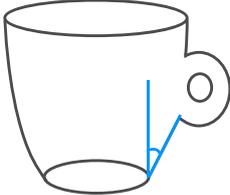
9.5.3 Z-Offset bottom layers

Placement	Setting Name	Illustration
Expert Support	Z-Offset bottom layers	

Z-offset (mm):

- Distance between the support and the printed object on the Z axis.

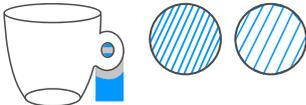
9.5.4 Overhang angle

Placement	Setting Name	Illustration
Expert Support	Overhang angle	

Overhang angle (degrees):

- Maximum allowed angle for overhangs without building support.
- 0° = vertical, 90° = horizontal line with respect to the build platform.
- Check the "Show layers" window, to see if the support is applied at the desired places.

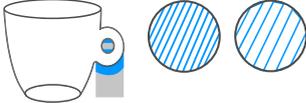
9.5.5 Support: Infill density

Placement	Setting Name	Illustration
Expert Support	Infill density	

Support Density:

- Density of of the support.
- 100% will print solid support layers, 0% will not make any infill.
- High support densities will create a more stable support, which will give better support under overhangs, but will be more difficult to remove and increase the print time.

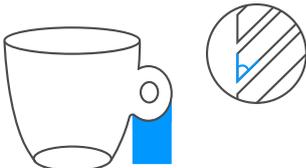
9.5.6 Support: Full infill density

Placement	Setting Name	Illustration
Expert Support	Full Infill density	

Full support density (%):

- Defines the density of the solid support layers used at the local tops of the support.
- 100% will print solid support layers, 0% will not make any infill.

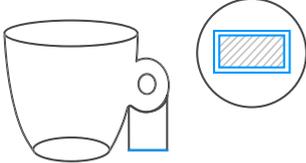
9.5.7 Support: Infill angle

Placement	Setting Name	Illustration
Expert Support	Infill angle	

Infill angle:

- Determines the angle the support infill lines.
- Typical support infill are 45 or 0 degrees. 45 degree is recommended.

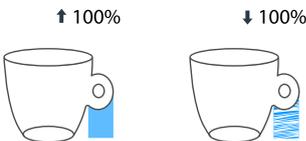
9.5.8 Number of contours

Placement	Setting Name	Illustration
Expert Support	Number of contours	

Number of support contours:

- Number of lines in a support contour / wall.
- Increasing the numbers of contours can improve the stability of the support.

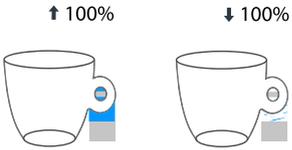
9.5.9 Over/under extrusion ratio

Placement	Setting Name	Illustration
Expert Support	Over/under extrusion ratio	

Over/under-extrusion ratio (%):

- Set over/under-extrusion ratio to a value greater than 100% to over-extrude and less to under-extrude.
- Under-extrusion is beneficial for support as it results in a brittle support that is easier to remove.

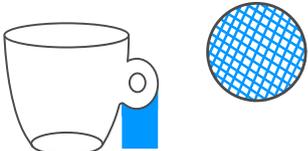
9.5.10 Top/bottom over/under extrusion

Placement	Setting Name	Illustration
Expert Support	Top/bottom over/under extrusion	

Over/under-extrusion ratio Top/Bottom (%):

- Set over/under-extrusion ratio to a value greater than 100% to over-extrude and less to under-extrude.
- 100% or slightly over-extruded is beneficial, since this is the part that sticks to the bottom.

9.5.11 Crossed infill

Placement	Setting Name	Illustration
Expert Support	Crossed infill	

Crossed support:

- Support infill will be printed in the crossed infill pattern.

9.5.12 Support Contour Width

Placement	Setting Name	Illustration
Expert Support	Support contour width	None

Support contour width:

- The width of printed support contours.
- This value changes automatically when the value of the setting "Nozzle diameter" is changed.

9.5.13 Support Contour overlap

Placement	Setting Name	Illustration
Expert Support	Support contour overlap	None

Support contour overlap:

- How much of the contour overlaps the neighboring contours.
- 100% means that half of the contour overlaps the contour to the right and the other half overlaps the contour to the left.
- 0% means that the contours don't overlap each other.

9.5.14 Support Contour overlap smallest detectable

Placement	Setting Name	Illustration
Expert Support	Contour overlap smallest detectable	None

Support contour width smallest detectable:

- When printing very small details, which require support that cannot be printed with regular support contour width, the smallest detectable width is applied.

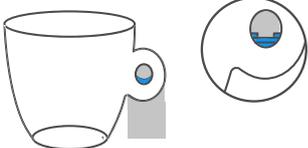
9.5.15 Support top thickness

Placement	Setting Name	Illustration
Expert Support	Number of support top layers	

Support top thickness (mm):

- Defines the distance from the top of a support structure with solid support infill.

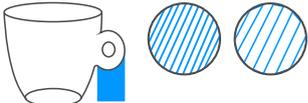
9.5.16 Support bottom thickness

Placement	Setting Name	Illustration
Expert Support	Number of support bottom layers	

Support bottom thickness (mm):

- Defines the distance from the bottom of a support structure with solid support infill.

9.5.17 Support infill distance

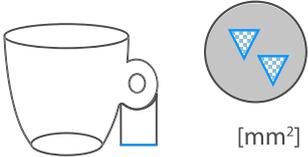
Placement	Setting Name	Illustration
Expert Support	Support infill frequency layers	

Support Infill distance (mm):

Defines the layer thickness for support infill. Full infill areas are not affected and will be printed on each layer.

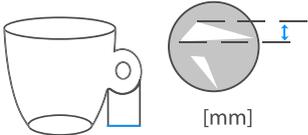
- This allows faster prints, by printing less support infill.
- If the distance is the same as the layer thickness, there is infill on every layer.
- Infill frequency has upper limit to 1.5 x Nozzle diameter

9.5.18 Remove holes smaller than

Placement	Setting Name	Illustration
Expert Support	Remove holes smaller than	

Remove holes in the support with an area smaller than the specified value

9.5.19 Remove holes smaller than in 3d

Placement	Setting Name	Illustration
Expert Support	Remove holes smaller than in 3d	

Remove holes in the support with a width less than the specified value

9.5.20 Generate automatic infill connections

Placement	Setting Name	Illustration
Expert Support	Generate automatic infill connections	None

If enabled, REACTOR will try to make sure infill lines are connected at the ends. If disabled, you can specify the length of additional infill before and after infill lines. These will follow the shape of the contour. The distance of these connecting lines can be defined in the field "Leading/trailing infill line length" 9.5.21.

9.5.21 Leading/trailing infill line width

This field is only available if "Generate automatic infill connections" 9.5.20 is disabled, and in that case gives the ability to control a trail before the infill line.

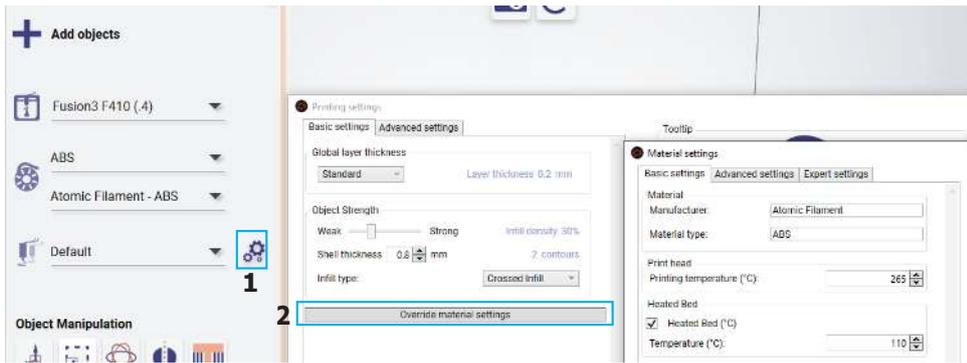
Placement	Setting Name	Illustration
Expert Support	Leading/trailing infill line width	None

Specifies the distance that infill should not go above when trying to connect infill lines. If the specified length is not possible to achieve, the closest possible length will be used. Set this to 0 to remove all infill connections.

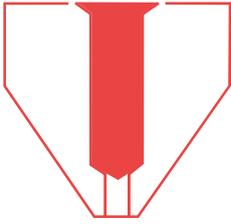
9.6 Material settings: Basic

To open the Material settings:

1. Press "settings"
2. Open "Material settings override"



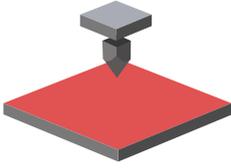
9.6.1 Print head temperature

Placement		Setting Name	Illustration
Material	Basic	Print head temperature	

Printing temperature:

- The temperature at which the material is melted in the printhead.

9.6.2 Print bed temperature

Placement		Setting Name	Illustration
Material	Basic	Print bed temperature	

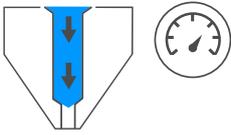
Heated bed temperature (°C):

- The temperature of the heated bed while printing.

- If you have problems with objects not sticking to the bed, consider raising the temperature.

9.7 Material settings: Advanced

9.7.1 Maximum extrusion

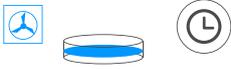
Placement	Setting Name	Illustration
Material Advanced	Maximum extrusion	

Maximum extrusion (mm³/s):

- Defines the maximum amount of material per second that is allowed to be pushed out of the print head.
- Value is dependent on the material type, since different materials melt at different temperatures.
- This setting will limit the maximum printing speed, if necessary.

Tip: If you experience under-extrusion try to decrease this value.

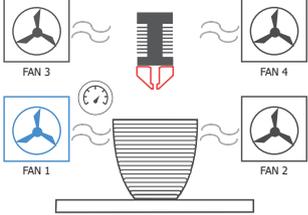
9.7.2 Min time pr layer

Placement	Setting Name	Illustration
Material Advanced	Min time pr layer	

Minimum time per layer in seconds:

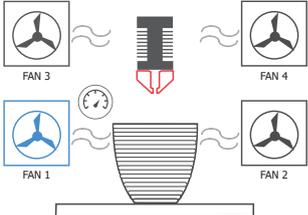
- Defines the minimum allowed time for a layer to be printed.
- Slows down the speed at small layers in order to have enough time to cool and harden the material.
- Dynamically increases the fan speed to cool down the material.
- Considerably improves the quality of details of small layers.
- To bypass, enter 0 as value.

9.7.3 Cooling

Placement		Setting Name	Illustration
Material	Advanced	Maximum fan speed	

MISSING

9.7.4 Maximum fan speed

Placement		Setting Name	Illustration
Material	Advanced	Maximum fan speed	

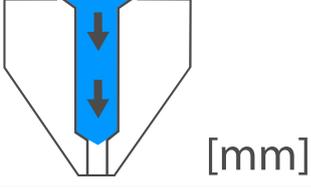
Maximum fan speed:

This value determines the maximum speed that the fan is allowed to reach.

If this value is higher than the target fan speed, REACTOR will dynamically increase the fan speed towards the maximum allowed speed, if a layer is slowed down, due to Minimum time per layer violations.

In order to deactivate this feature, set the default and maximum fan speed to the same value.

9.7.5 Initial pushout

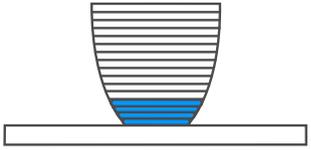
Placement		Setting Name	Illustration
Material	Advanced	initial push-out	

Initial push-out (mm):

The length of material pushed into the printhead before printing. The initial push-out will happen as soon as the target temperatures are reached.

- This is done in order to ensure that there is material in the printhead.

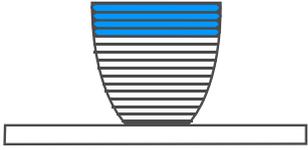
9.7.6 Custom first layers: Number of layers

Placement		Setting Name	Illustration
Material	Advanced	Custom first layers: Number of layers	

Number of bottom layers:

- Choose the number of bottom layers that will use 100% infill.

9.7.7 Custom first layers: printer speed

Placement		Setting Name	Illustration
Material	Advanced	Custom first layers: printer speed	

Print speed for bottom layers:

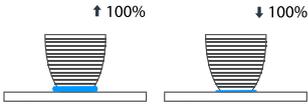
- The speed of print for the custom bottom layers.

9.7.8 Custom first layers: Repositioning speed

Repositioning speed for bottom layers:

-The speed of repositioning for the custom bottom layers.

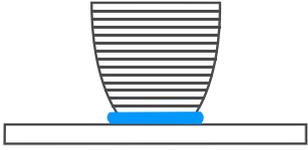
9.7.9 Custom first layers: Over/underextrusion rate

Placement		Setting Name	Illustration
Material	Advanced	Custom first layers: Over/under-extrusion rate	

Over/under-extrusion ratio (%):

- Set extrusion ratio to a value greater than 100% to over-extrude and less to under-extrude.

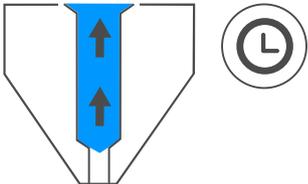
9.7.10 Custom first layers: First layer thickness

Placement	Setting Name	Illustration
Material Advanced	Custom first layers:First slice thickness	

First layer thickness (mm):

- It can be useful to let this be thicker than the other layers in order to stick better to the printing surface.
- Setting this value to 0 will cause REACTOR to use the selected layer thickness in basic settings for the first layer instead.

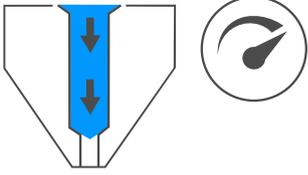
9.7.11 Push-out speed

Placement	Setting Name	Illustration
Material Advanced	push-out speed	

Push-out speed (mm/s):

- Push-out starts at half impulse speed and accelerates to maximum speed. By the end of push-out, the extruder speed drops to minimum speed and extrusion starts.
- Push-out happens by the end of repositioning that crosses the contour of the printed object.

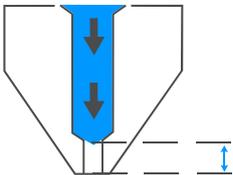
9.7.12 Push-out acceleration

Placement		Setting Name	Illustration
Material	Advanced	push-out acceleration	

Push-out acceleration rate (mm/s²)

- The acceleration rate and deceleration rate of push-out.
- This is speed change per second and the value is limited by the strength of the extruder.
- Reduce this value if either the plastic gets grinded during push-out or the motor cannot keep up.

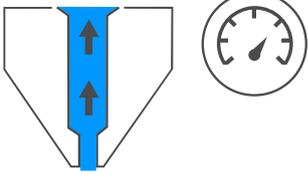
9.7.13 Push-out distance

Placement		Setting Name	Illustration
Material	Advanced	push-out distance	

Push-out distance (mm):

- Distance of material moved during retraction push-out
- A value of 0 turns off any push-out.

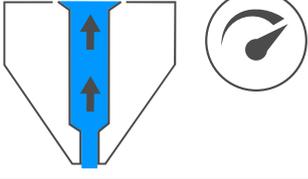
9.7.14 Suck-back speed

Placement		Setting Name	Illustration
Material	Advanced	Suck-back speed	

Suck-back speed (mm/s):

- Suck-back starts at half the impulse speed and accelerates to maximum speed. By the end of suck-back, the extruder speed drops to minimum and extrusion ends.
- Suck-back happens just before repositioning that crosses the contour of the printed object.

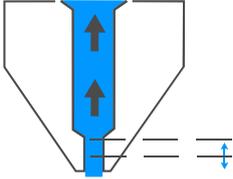
9.7.15 Suck-back acceleration

Placement		Setting Name	Illustration
Material	Advanced	Suck-back acceleration	

Suck-back acceleration rate (mm/s²):

- The acceleration rate and deceleration rate of suck-back.
- This is speed change per second and the value is limited by strength of the extruder.
- Reduce this value if either the plastic gets grinded during suck-back or the motor cannot keep up.

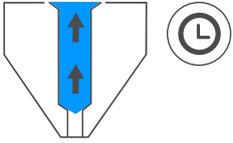
9.7.16 Suck-back distance

Placement		Setting Name	Illustration
Material	Advanced	suck-back distance	

Suck-back distance (mm):

- Distance of material moved during retraction suck-back
- A value of 0 turns off any suck-back

9.7.17 Maximum Suck back during Movement

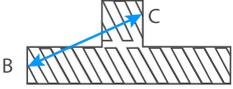
Placement		Setting Name	Illustration
Material	Advanced	Maximum Suck back during Movement	

Max suck-back during movement(s):

The suck-back can start during the last part of a path. This setting controls how long time during the end of a path, can contain suck-back.

- Put a value of 0 seconds to force the whole suck-back to be performed while waiting at the end of the path.

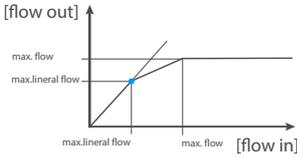
9.7.18 Longest repositioning line

Placement	Setting Name	Illustration
Material Advanced	Longest Repositioning line	

If a repositioning line is longer than this distance, it will do retraction even if the print head only moves inside the object.

9.8 Material settings: Expert

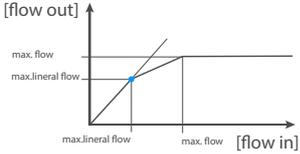
9.8.1 Flow factor

Placement	Setting Name	Illustration
Material Expert	Default flow factor	

Flow factor (%):

- Flow is used to compensate for material properties of different types.
- Higher flow results in increased width of the extrusion.
- The flow should be changed if you experience quality issues after switching to a different material.

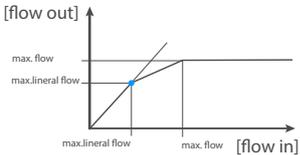
9.8.2 Use advanced filament profiles

Placement	Setting Name	Illustration
Material Expert	Use advanced filament profiles	 <p>The graph plots [flow out] on the y-axis against [flow in] on the x-axis. A diagonal line represents linear flow, and a horizontal line represents the maximum flow rate. A blue dot marks the intersection of the linear flow line and the horizontal maximum flow line. Vertical dashed lines from the x-axis indicate the 'max.linear flow' (at the blue dot) and the 'max. flow' (at the horizontal line's start).</p>

Use advanced material profile if you have more details of the material you are currently printing with. This option enables compensation for some under-extrusion by knowing when the material stops extruding linearly.

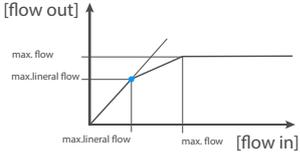
- Note that when this option is enabled, the value for "Maximum extrusion (mm³/s)" in the Advanced tab will be the value of "Max linear flow rate (mm³/s)".

9.8.3 Max flowrate

Placement	Setting Name	Illustration
Material Expert	Max flowrate	 <p>The graph plots [flow out] on the y-axis against [flow in] on the x-axis. A diagonal line represents linear flow, and a horizontal line represents the maximum flow rate. A blue dot marks the intersection of the linear flow line and the horizontal maximum flow line. Vertical dashed lines from the x-axis indicate the 'max.linear flow' (at the blue dot) and the 'max. flow' (at the horizontal line's start).</p>

Maximum achievable flow rate (out) and the applied flow rate to achieve this limit (in).

9.8.4 Max linear flowrate

Placement	Setting Name	Illustration
Material Expert	Max linear flowrate	 <p>The graph plots '[flow out]' on the vertical axis against '[flow in]' on the horizontal axis. A diagonal line starts from the origin. A horizontal line is drawn at a level labeled 'max. flow'. A vertical line is drawn at a level labeled 'max. linear flow'. The intersection of the diagonal line and the horizontal line is marked with a blue dot. The horizontal line continues to the right, and the diagonal line continues upwards.</p>

Maximum extrusion (mm³/s):

- Defines the maximum amount of material per second that is allowed to be pushed out of the print head.
- Value is dependent on the material type, since different materials melts at different temperatures.
- This setting will limit the maximum printing speed, if necessary.

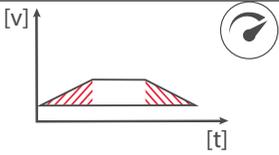
Tip: If you experience under-extrusion try to decrease this value.

9.8.5 Minimum layer speed

Placement	Setting Name	Illustration
Material Expert	Minimum layer speed	None

When reducing the printing speed to slow down a layer, the speed can never be reduced below this value.

9.8.6 Minimum Acceleration

Placement	Setting Name	Illustration
Material Expert	Minimum acceleration	 <p>The graph plots velocity '[v]' on the vertical axis against time '[t]' on the horizontal axis. A trapezoidal shape is shown with red diagonal hatching, representing a velocity profile that ramps up, stays constant, and ramps down. A circular icon with a checkmark is located in the top right corner of the graph area.</p>

Minimum Acceleration (mm/s²):

- Minimum acceleration is the smallest value for acceleration and deceleration rate. This is used when a

path is slowed down.

- This value is the minimum value of the Boost-slider in Printing settings -> Advanced tab. It is assigned when the Boost-slider is at 0%.

9.9 Printer settings: Others

9.9.1 Print file flavour

Placement		Setting Name	Illustration
Printer	Others	Print file flavour	None

Specify the print file flavour to fit the firmware of your printer

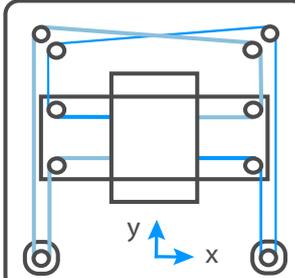
9.9.2 Enable pauses

Placement		Setting Name	Illustration
Printer	Others	Enable pauses	None

This setting enables pause capabilities for your printer. This includes planned pause and pause from the printer screen. Make sure that the printer has the mechanics to support this feature.

9.10 Printer settings: Workspace

9.10.1 Coordinate system

Placement	Setting Name	Illustration
<p>Printer</p> <p>Workspace</p>	<p>Coordinate system</p>	 <p>The illustration shows a top-down view of a printer's workspace. A central rectangular print head is positioned within a square frame. Four vertical support arms connect the corners of the frame to the print head. A coordinate system is shown at the bottom center, with a vertical arrow labeled 'y' and a horizontal arrow labeled 'x'.</p>

Cartesian:

This coordinate system is the regular type with a normal X, Y and Z axis.

CoreXY

Instead of cartesian coordinate system a 3D printer can use a CoreXY coordinate system. This affects the movement on the X and Y axes.

Delta

In this coordinate system, three arms move the print head around. The motors move three carriages, which are fixed to the other end of the three arms.

9.10.2 Printer front

Placement	Setting Name	Illustration
Printer Workspace	Printer front	

Choose printer front:

- To ensure the symmetry of the 3D printed object, the printers coordinate system is defined using the right hand rule. This means that X,Y and Z are always represented as shown on the picture above.
- X, Y and Z axis should therefore be plugged into the motherboard accordingly as well as the HOME define by the limit switches X1,Y1 and Z1.
- This option allows defining the front of the printer and at the same time defines where the HOME position of the printer is.

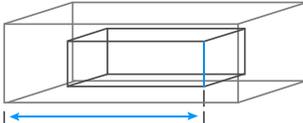
9.10.3 Print head moves on

Placement	Setting Name	Illustration
Printer Workspace	Print head moves on	

Printer type:

- Specify which axis the printhead moves on.
- By default the printhead moves on X and Y, and the printbed moves on Z.

9.10.4 Workspace size: XYZ min and max

Placement	Setting Name	Illustration
<p>Printer Workspace</p>	<p>Workspace size: XYZ min and max</p>	

Workspace maximum position on X:

- Represents the last point of the printer workspace on X axis according to defined home reposition.
- Workspace minimum and maximum positions define the length of the workspace on which is possible to print. Both minimum and maximum positions are calculated according to home reposition values which defining the (0,0,0) coordinates.
- Printer workspace is bounded by printer scale but should be bounded by printer bed.

Workspace minimum position on X:

- Represents the first point of the printer workspace on X axis according to defined home reposition.
- Workspace minimum and maximum positions define the length of the workspace on which is possible to print. Both minimum and maximum positions are calculated according to home reposition values which defining the (0,0,0) coordinates.
- printer workspace is bounded by printer scale but should be bounded by printer bed

Workspace maximum position on Y:

- Represents the last point of the printer workspace on Y axis according to defined home reposition.
- Workspace minimum and maximum positions define the length of the workspace on which is possible to print. Both minimum and maximum positions are calculated according to home reposition values which defining the (0,0,0) coordinates.
- Printer workspace is bounded by printer scale but should be bounded by printer bed.

Workspace minimum position on Y:

- Represents the first point of the printer workspace on Y axis according to defined home reposition.
- Workspace minimum and maximum positions define the length of the workspace on which is possible to print. Both minimum and maximum positions are calculated according to home reposition values which defining the (0,0,0) coordinates.
- Printer workspace is bounded by printer scale but should be bounded by printer bed.

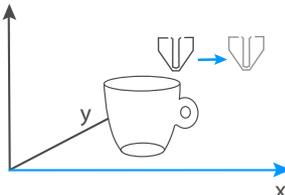
Workspace maximum position on Z:

- Represents the last point of the printer workspace on Z axis according to defined home reposition.
- Workspace minimum and maximum positions define the length of the workspace on which is possible to print. Both minimum and maximum positions are calculated according to home reposition values which defining the (0,0,0) coordinates.
- The printer workspace is bounded by printer scale

Workspace minimum position on Z:

- Represents the first point of the printer workspace on Z axis according to defined home reposition.
- Workspace minimum and maximum size define the length of the workspace on which is possible to print. Both minimum and maximum positions are calculated according to home reposition values which defining the (0,0,0) coordinates.
- The printer workspace is bounded by printer scale.

9.10.5 End of print position XYZ

Placement	Setting Name	Illustration
<p>Printer Workspace</p>	<p>End of print position XYZ</p>	

The "End of print" setting defines the target position in millimetres of the printhead after a successful 3D print is done.

When the print is finished, the Z-axis will move this distance away from the print head.

- Set this value to the workspace Z size to always make the bed move all the way away from the print head.

9.10.6 Heatup position

Placement		Setting Name	Illustration
Printer	Workspace	Heatup position	None

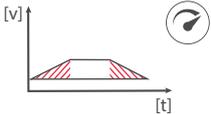
Heatup position:

Represents the coordinate of the print head during the initial heating.

- The X/Y coordinates must be a position where the Z home sensor can be triggered.
 - The coordinates are in mm.
 - This option is made to avoid damaging the printbed due to repeated heatup at the same point.
 - The heatup position can be outside the workspace so that the initial push-out does not hit the heat bed.
- Make sure to pick a safe position that is reachable.

9.11 Printer settings: Motion

9.11.1 Maximum acceleration

Placement		Setting Name	Illustration
Printer	Motion	Maximum acceleration	

Maximum Acceleration Rates (mm/s²):

- Maximum allowed acceleration and deceleration for printing and repositioning. These values are limited by the strength of the motors and the mechanical friction of the axes.

9.11.2 Maximum speed limit

Placement	Setting Name	Illustration
Printer Motion	Maximum speed limit	None

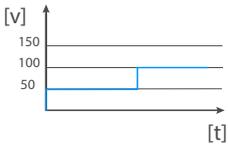
Determine the highest values allowed for the maximum speeds .

9.11.3 Minimum speed limit

Placement	Setting Name	Illustration
Machine Motion	Minimum slider speed	None

This is the target print speed when the "Speed Mode" sliders in Printing settings -> Advanced tab are moved to the left-most position (towards Quality).

9.11.4 Impulse lookahead distance

Placement	Setting Name	Illustration
Machine Motion	Impulse lookahead distance	

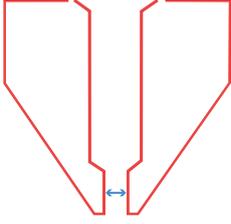
Impulse lookahead distance (mm):

When calculating the speed during the path, the angles between the line segments in the path decide the speed limits. If the object has a very high resolution, the angles will be very small.

This is the distance to lookahead in order to combine the angles of many small lines and give the same result for high and low resolution objects.

9.12 Printer settings: Materials

9.12.1 Nozzle diameter

Placement	Setting Name	Illustration
Printer Materials	Nozzle diameter	

Nozzle diameter (mm):

- The diameter of the printhead's nozzle.
- This value should only be changed when the nozzle size of the printhead change.

Tip: The following settings will change automatically when Nozzle diameter change:

- Layer resolution (Layer thickness)
- Contour width
- Contour width smallest detectable
- Contour width smallest printable
- Infill line width
- Infill height (affects the maximum allowed value)

9.12.2 Enable primary heater

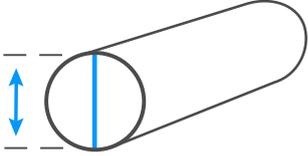
Placement	Setting Name	Illustration
Printer Materials	Enable primary heater	None

Defines whether or not a heater is present for the primary print head.

When enabled, the settings related to the primary heater are enabled and heat-up G-Code is generated.

When disabled, the settings related to all heaters are disabled and no heat-up GCode is generated.

9.12.3 Filament diameter

Placement	Setting Name	Illustration
Printer Materials	Filament diameter	

Diameter of the filament.

9.13 Printer settings: Bed leveling

9.13.1 Bed leveling method

Placement	Setting Name	Illustration
Printer Bed leveling	Bed leveling method	None

Bed levelling method:

Chose if the printer is set up for manual or automatic bed levelling.

9.13.2 Z-Offset after leveling

Placement	Setting Name	Illustration
Printer Bed leveling	Z-Offset after leveling	None

Levelling sensor offset:

The offset between the levelling sensor and the nozzle

9.14 Printer settings - Custom G-Code

Placement	Setting Name	Illustration
Printer Custom G-Code	Explain the different fields	None

9.15 Application

Settings related to the REACTOR application.

9.15.1 Enable REALview

Placement	Setting Name	Illustration
Manufacturer Others	Enable REALview	None

Enable the use of REALview. This feature will allow you to see 3D models next to you in augmented reality. Requires a webcam.

9.15.2 Show slider values

Placement	Setting Name	Illustration
Manufacturer Motors	Show slider values	None

If checked, the values next to the speed sliders in Printing settings -> Advanced tab will show the actual calculated speeds and accelerations. If unchecked, it will be relative values in %.

9.15.3 3D model color in workspace

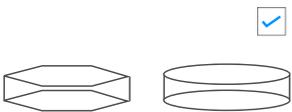
Placement	Setting Name	Illustration
Advanced	3D model color in workspace	None

Print bed model color:

- Defines the color of the printing bed model.

Here you can specify the folder you wish to use for downloading models from MyMiniFactory. If you do not specify the path here, you will be asked to do so the next time you download a model.

9.15.4 Model simplification

Placement	Setting Name	Illustration
Expert Application	Model simplification	

Disable simplification:

To decrease slicing time 3D models are simplified when opened.

Disable simplification if you experience that you have lost details on your object after slicing. Note that this will increase the slicing time significantly.

9.15.5 Desired number of polygons

Placement	Setting Name	Illustration
Expert Application	Desired number of polygons	None

If simplification is enabled, REACTOR will attempt to simplify the 3D model by reducing its number of polygons to the desired number of.

Higher values yield higher 3D model resolution at the cost of computation time, where as lower values

yields shorter computation time at the cost of detail.

If simplification aggressiveness is too low, the desired number of polygons might not be reached.

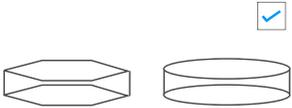
9.15.6 Simplification aggressiveness

Placement	Setting Name	Illustration
Expert Application	simplification aggressiveness	None

High aggressiveness causes the simplification algorithm to use all means necessary in order to reach the desired target polygon count even if it means removing large parts of the 3D model.

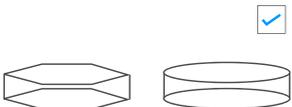
Low aggressiveness will result in the algorithm being more cautious with removing detail, even if it means that the desired number of polygons might not be reached.

9.15.7 Use path simplification

Placement	Setting Name	Illustration
Expert Application	Use path simplification	

Path simplification reduces the number of points in the printing path. This speeds up slicing time and helps avoiding computational bottlenecks for printer models with simple controllers, at the cost of potentially losing small details.

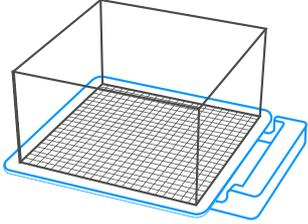
9.15.8 Line segment resolution

Placement	Setting Name	Illustration
Expert Application	Line segment resolution	

Line segment resolution (mm):

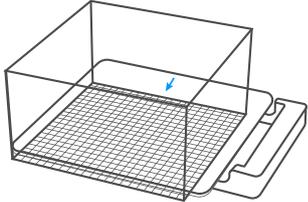
- Defines the allowed detail loss for path simplification
- Large values reduce path points. Too large values removes too much detail.
- Small values preserve the print details. Too small values results in high slicing time and increases the risk of computational bottlenecks for simple printer controllers.

9.15.9 Show print bed

Placement	Setting Name	Illustration
Expert Application	Show print bed	

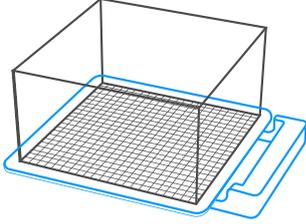
Enable/Disable the visualization of the print bed in the workspace.

9.15.10 Print bed model look XYZ

Placement	Setting Name	Illustration
Manufacturer Workspace	Print bed model look XYZ	

The grid home offset setting defines the position of the 0,0,0 point of the grid relative to the printing bed model's origin.

9.15.11 Bed rotation

Placement	Setting Name	Illustration
Manufacturer Workspace	Bed rotation	

Use this setting to rotate your bed clockwise in steps of 90 degrees.

9.15.12 Color

Placement	Setting Name	Illustration
Manufacturer Workspace	Color	None

Print bed model color:

- Defines the color of the printing bed model.

9.15.13 Opacity

Placement	Setting Name	Illustration
Manufacturer Workspace	Opacity	None

Print bed model opacity:

- Defines the opacity of the printing bed model. The lower the value the more transparent it will be.

9.15.14 Model download path

Placement	Setting Name	Illustration
Advanced	Model download path	None

9.15.15 Print logging

Placement	Setting Name	Illustration
Advanced	Print logging	None

Check this if you wish to log all prints from this computer in the appdata folder.