User Manual
ELEGOO 3D Slicing Software

V1.8

ELEGOO 3D Printer

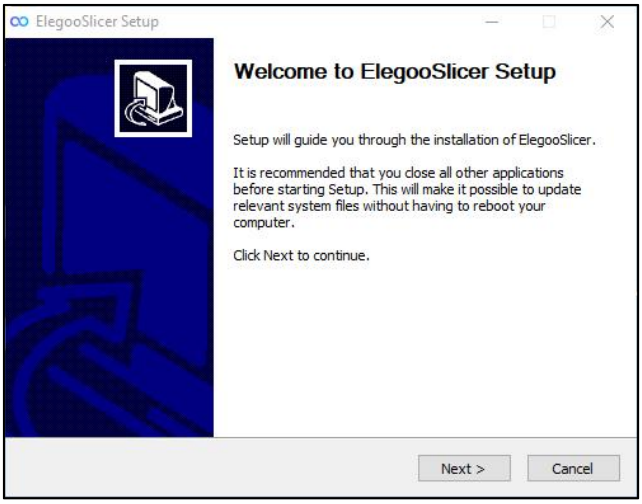
Table of Contents

I. Software Installation

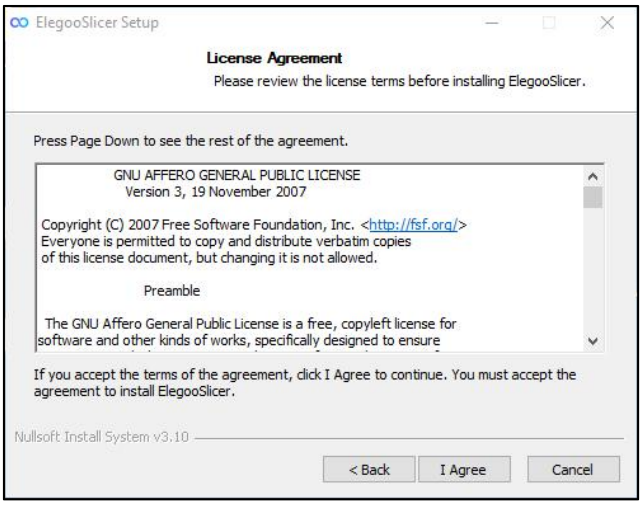
1. Find the installation package suitable for your computer system on the included USB drive and double-click to install with the left mouse button.

For example: If your computer runs on Windows, please select this option.

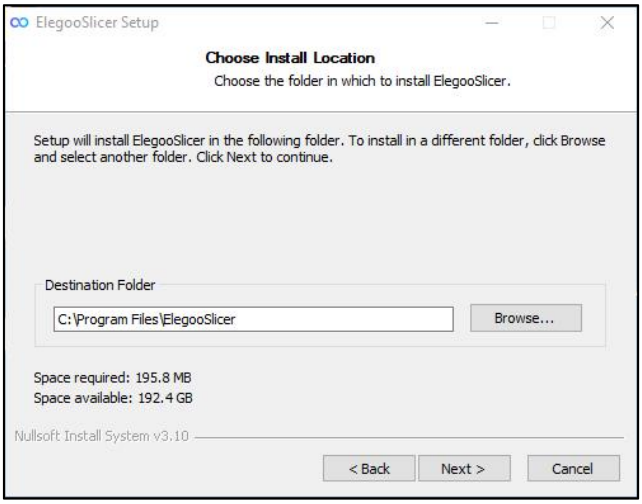
2. Click “Next.”



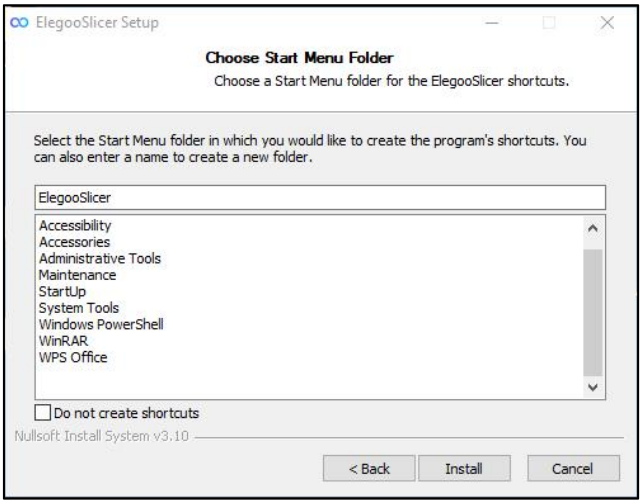
3. Click “I Agree.”



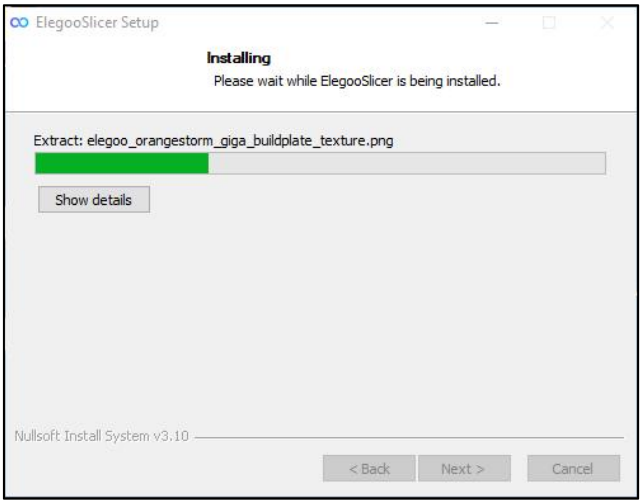
4. Choose your installation location, then click “Next.”



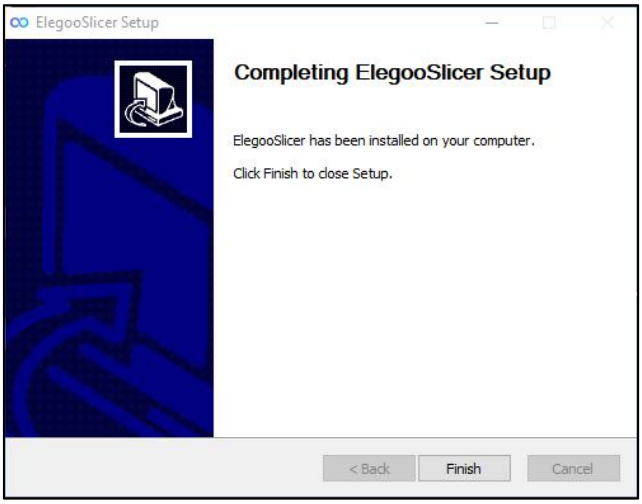
5. Click “Install.”



Installing... Please wait.



6. Click “Finish.”

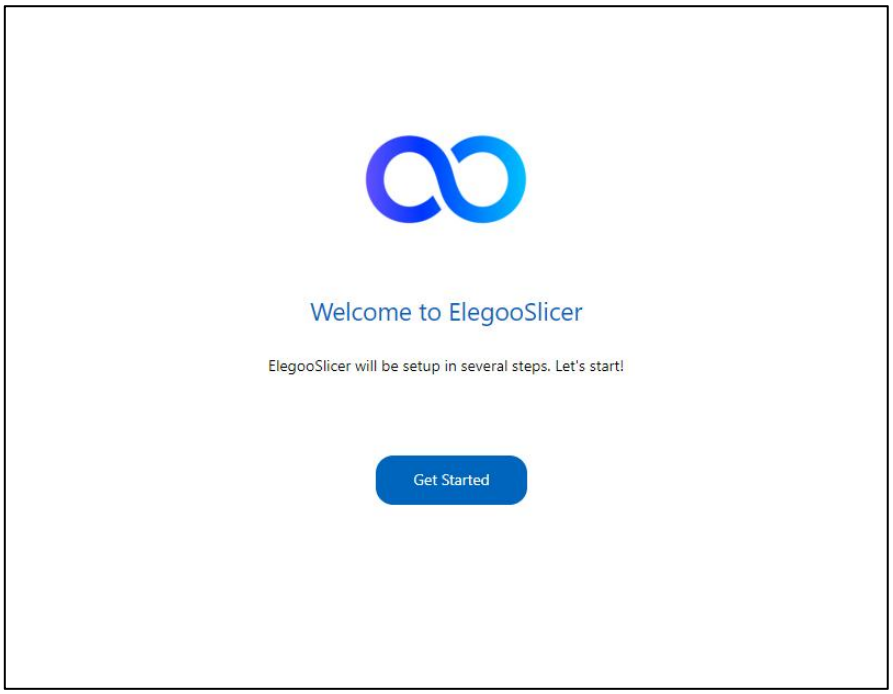


II. Software Initialization

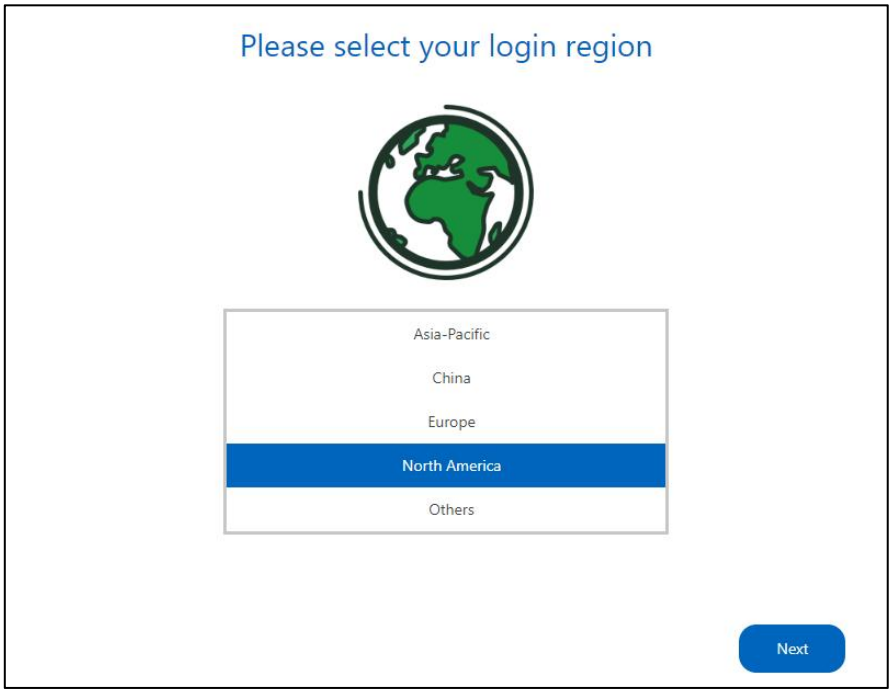
1. Double-click the ElegooSlicer icon on your computer desktop with the left mouse button to open the software.

Startup Screen

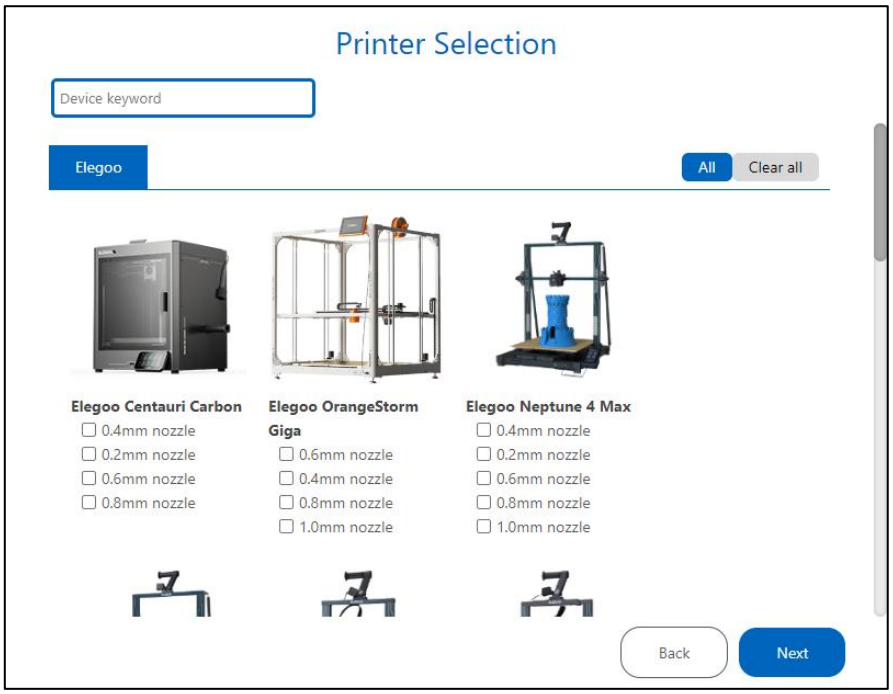
2. Click “Get Started” to proceed.



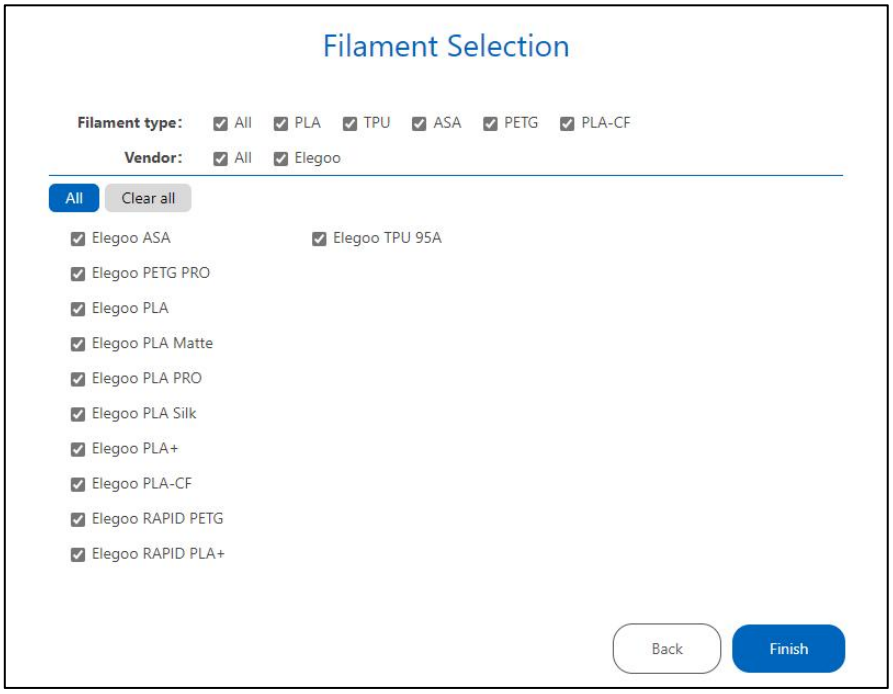
3. Select your login region, then click “Next.”



4. Choose your printer, then click “Next.”

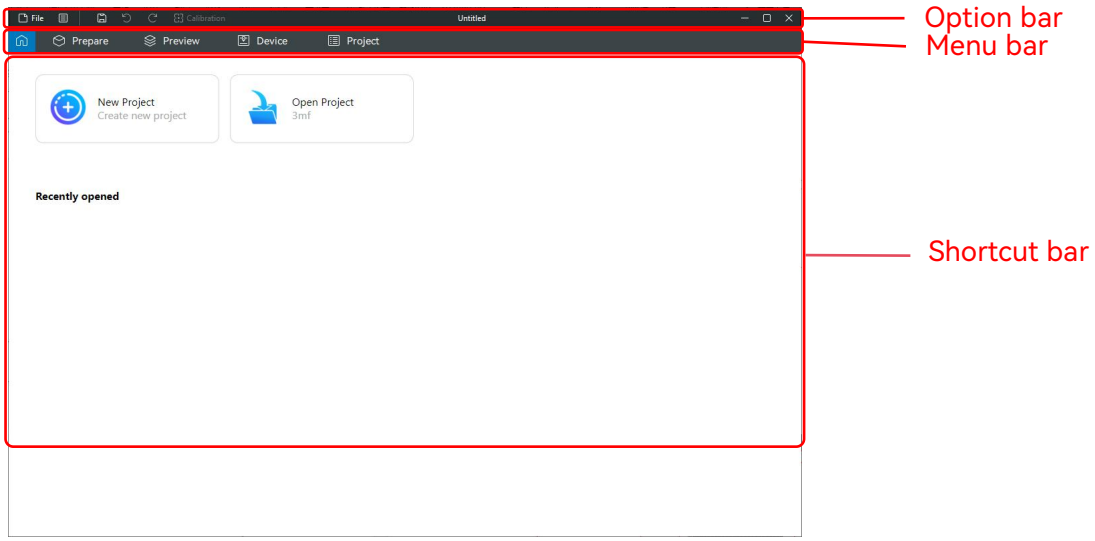


5. Select the filaments you will be using, then click “Finish.”

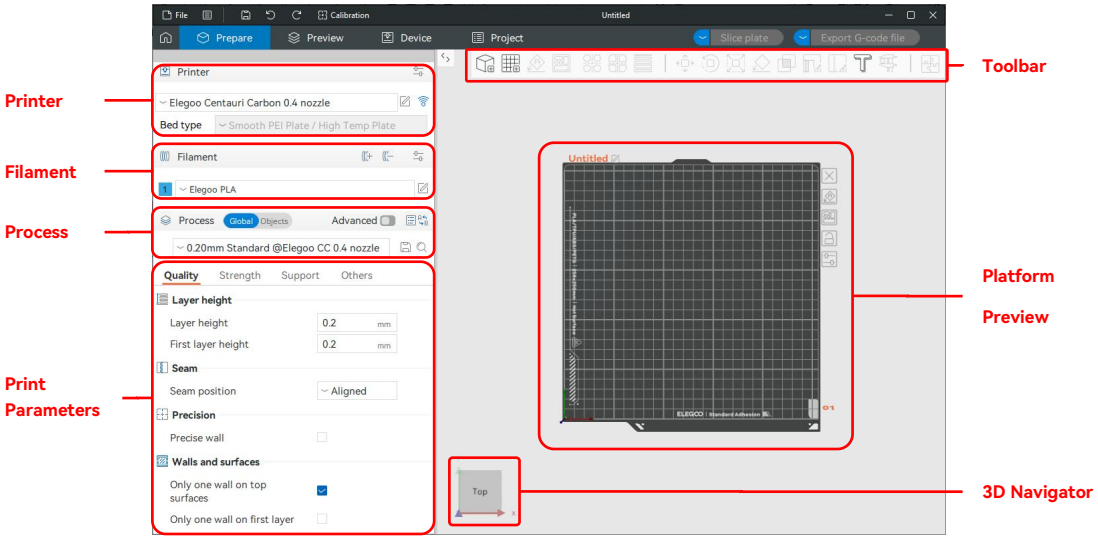


III. Software Function Introduction

【Software Main Interface】



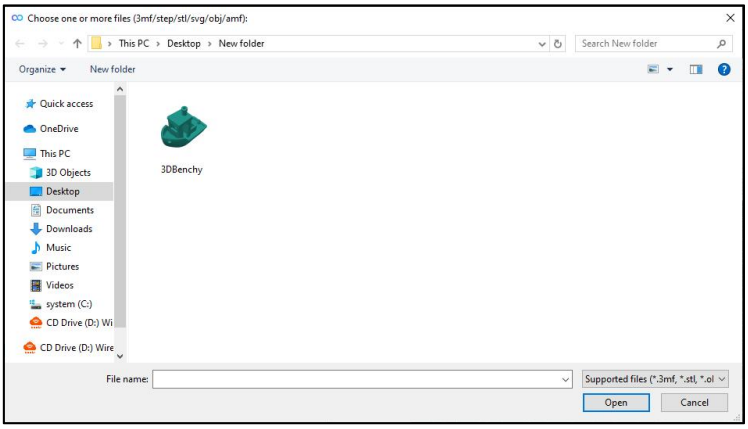
【Preparation Interface】

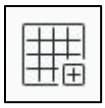


【Toolbar】

【Add】

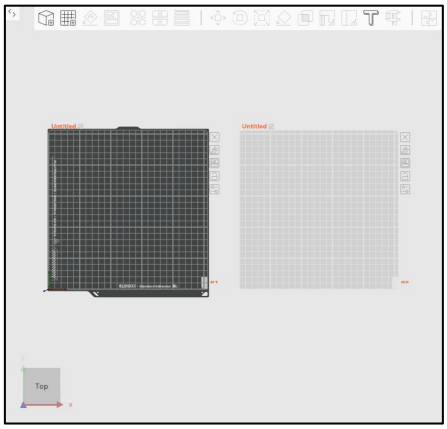
Add a new model.





【Add plate】

Add a new print platform, as shown in the image below.

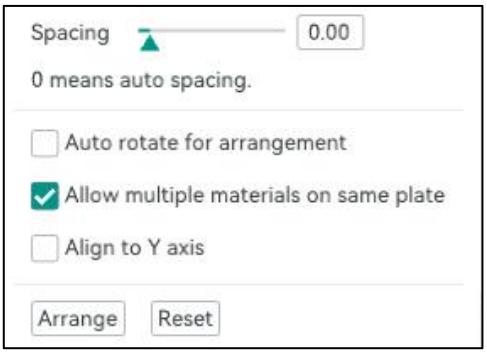


【Auto orient】

Find the optimal orientation based on several features of the selected object.

【Arrange all objects】

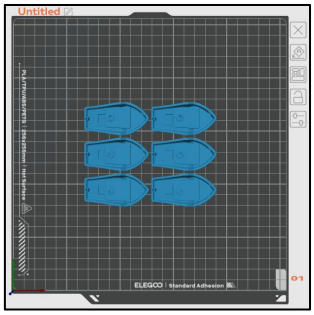
Arrange all objects on the platform.



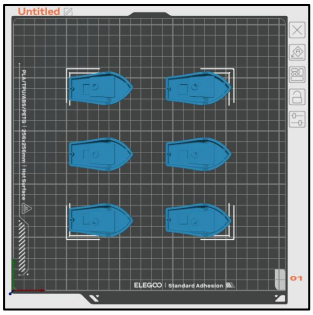
Settings menu

【Spacing】

Model-to-model spacing during auto-arrangement.



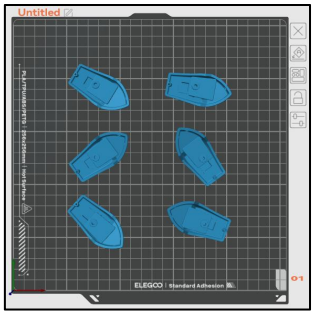
Spacing at 0



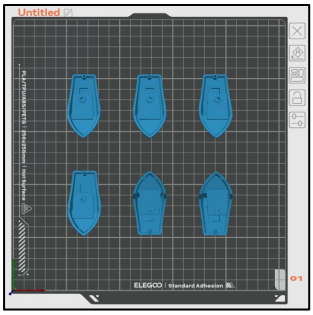
Spacing at 30

【Auto rotate for arrangement】

When enabled, models will be automatically rotated for the optimal orientation during auto-arrangement.



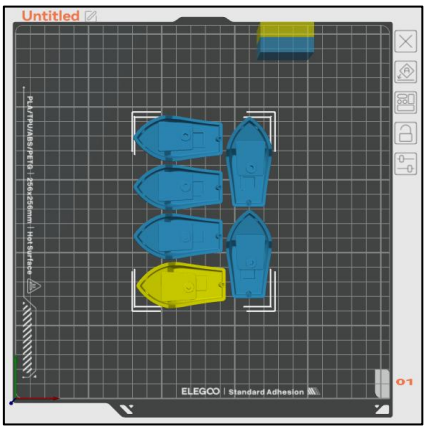
Not enabled



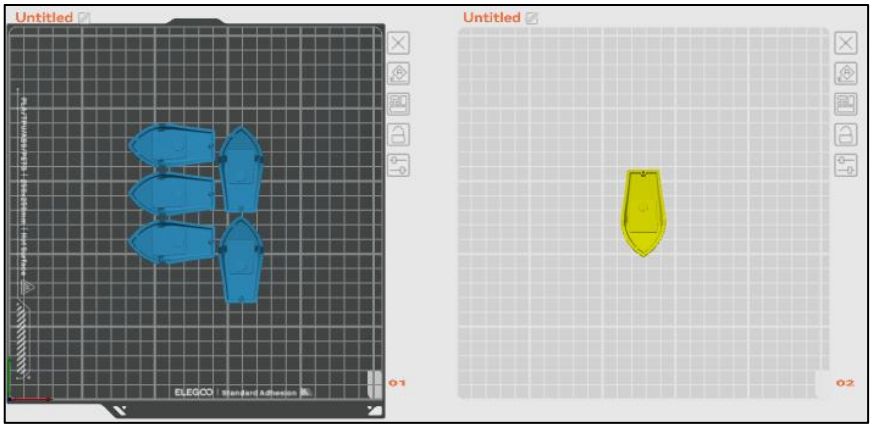
Enabled

【Allow multiple materials on same plate】

When allowed, models using different filaments will be placed on the same plate (models with significantly different print parameters will still be separated). When not allowed, models using different filaments will be placed on separate plates.



Allowed (yellow represents other filaments)

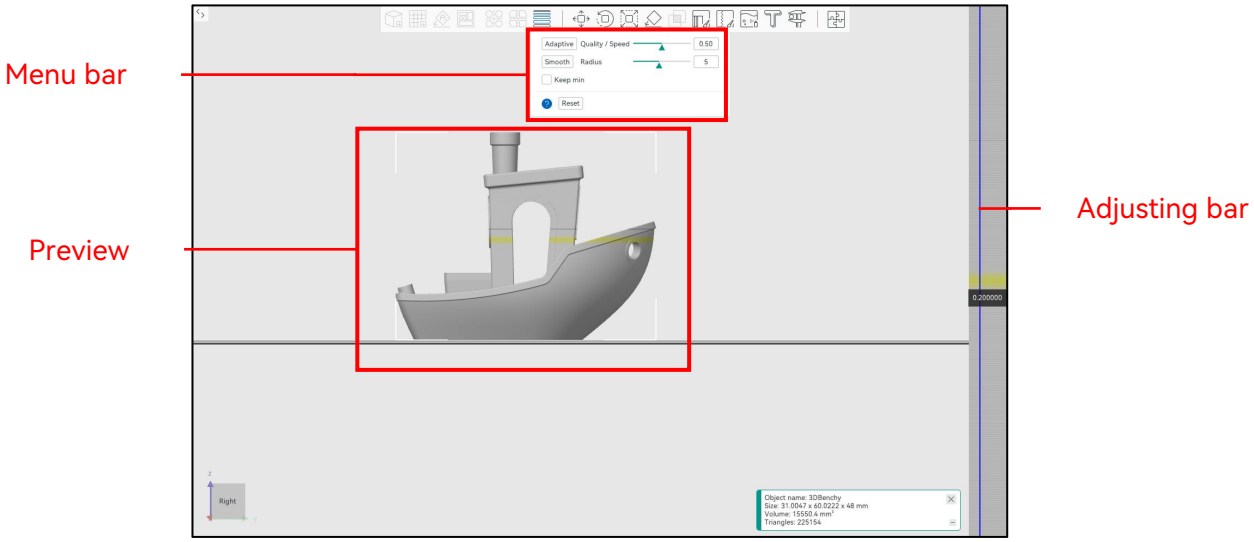


Not Allowed (yellow represents other filaments)

【Align to Y axis】

Align models along the Y-axis. When unchecked, it defaults to the X-axis.

【Variable layer height】



【Menu bar】

Adaptive: Calculate layer configurations based on detail/speed settings.
Smooth: Make the contours of variable layer height smooth. The larger the radius, the smoother the curve.
Keep min: When checked, the minimum layer height will not be smoothed out and will be maintained at the minimum value.
: Click the blue question mark for related operation steps.
Reset: Reset all settings.

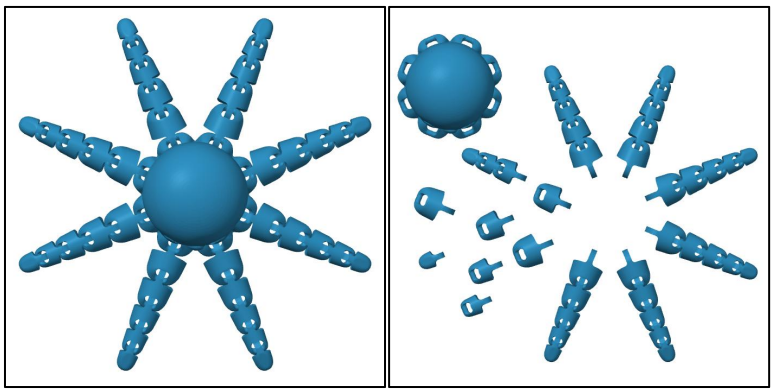
【Adjusting bar】

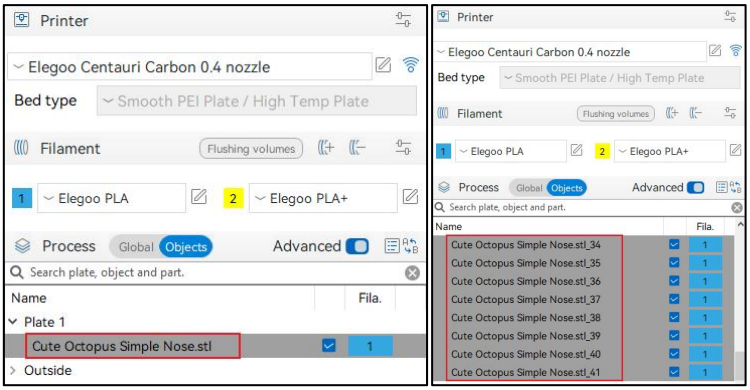
Use the mouse to adjust settings here. Click the blue question mark for related operation instructions.

【Split to objects】

Split a single model into distinct objects for individual manipulation; this option is disabled for non-composite models.

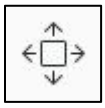
Example: After applying this function to the model below, the initial single unified model is split into multiple objects. Each object can be moved independently, and the printing parameters for each individual object can be separately adjusted in the Process section.





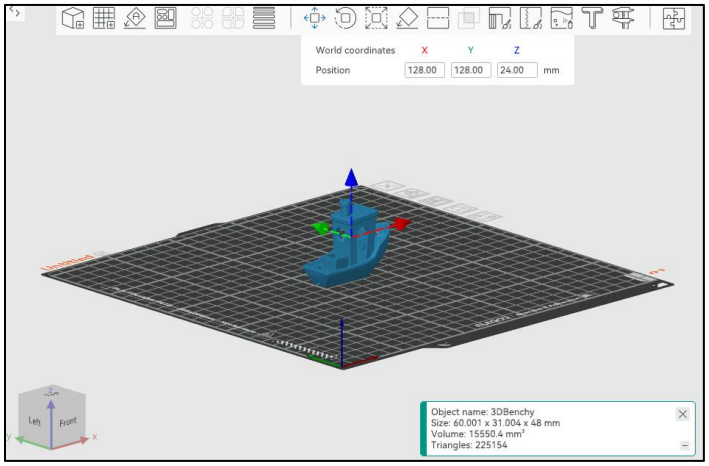
【Split to parts】

Divide a single model into distinct parts; this option is unavailable when the model is a single entity. Contrary to “Split into objects,” this function separates the model into distinct parts instead of individual objects.



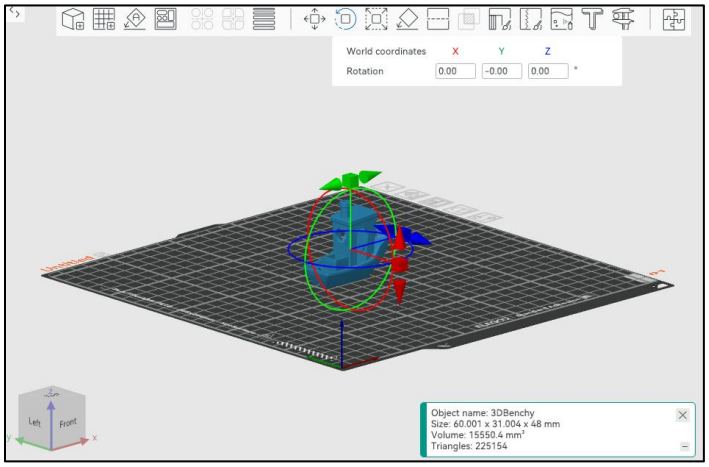
【Move】

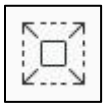
Use the mouse to drag the model and move it along the three axes on the platform or input precise coordinates for specific positioning.



【Rotate】

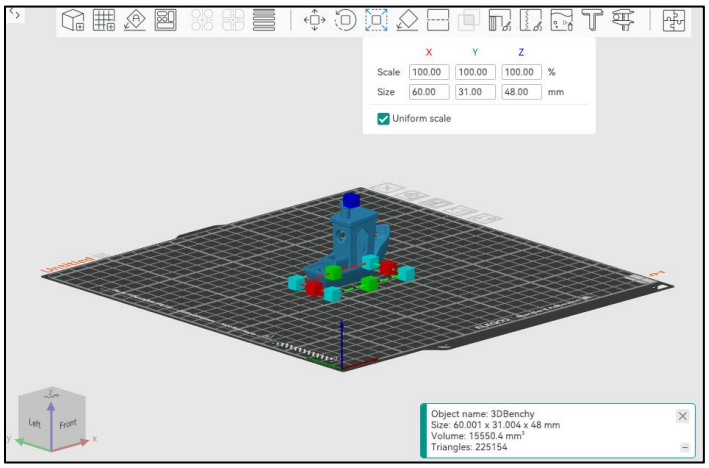
Rotate the model along the three axes using the mouse or input exact angles within the coordinate system for accurate and controlled rotation.

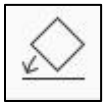




【Scale】

Adjust the model's size along the three axes by dragging with the mouse or input multipliers/exact sizes in the coordinate system to achieve the desired size. Uniform scale: When enabled, adjustments on one axis synchronize changes across all axes.



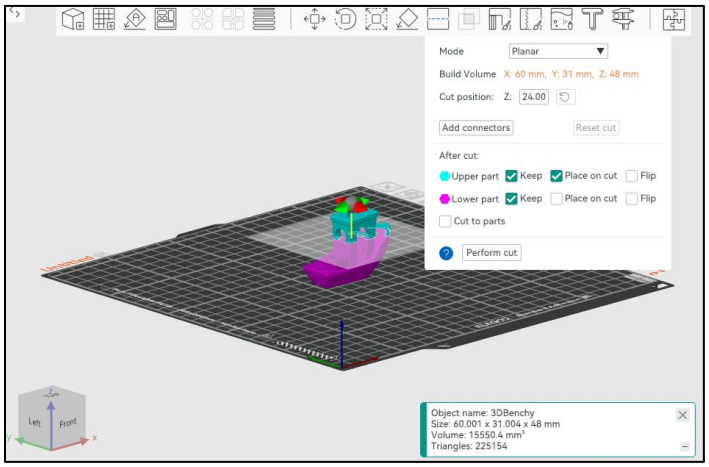


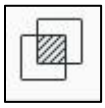
【Lay on face】

Choose a face of the model as the base for placement on the platform.

【Cut】

Cut the model at a specific height.



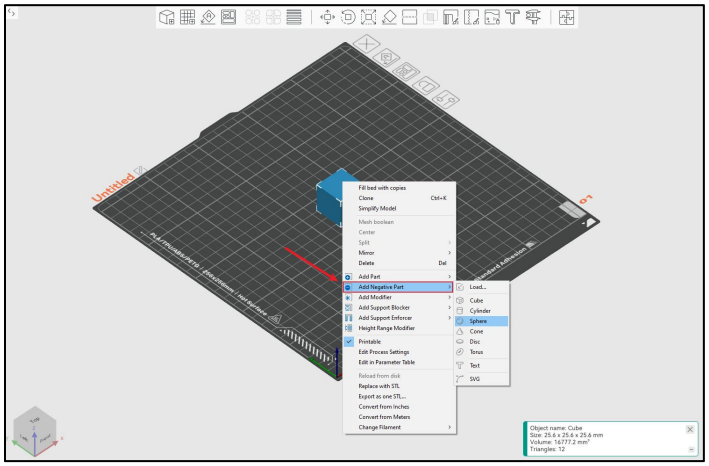


【Boolean operation】

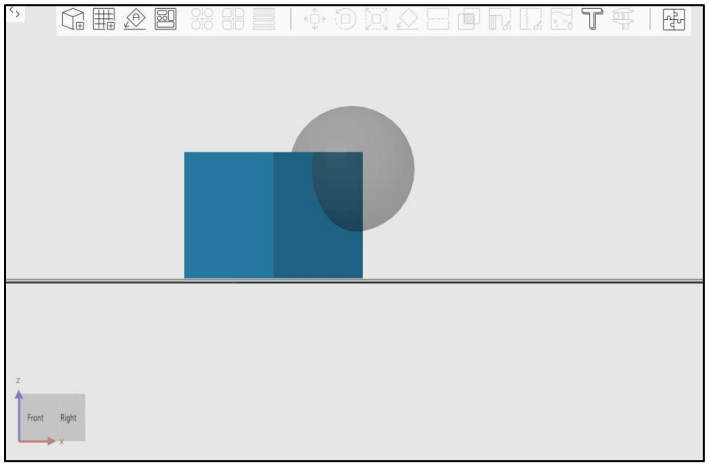
Execute union/intersection/difference operations on the model.

Instructions:

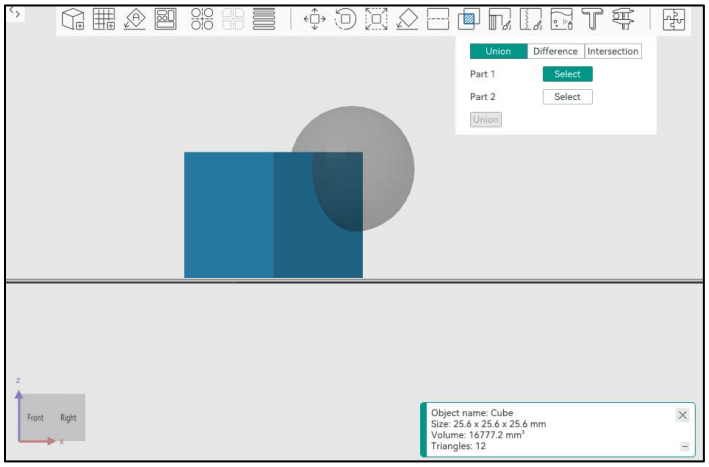
1. Right-click on the model and choose “Add Negative Part.”



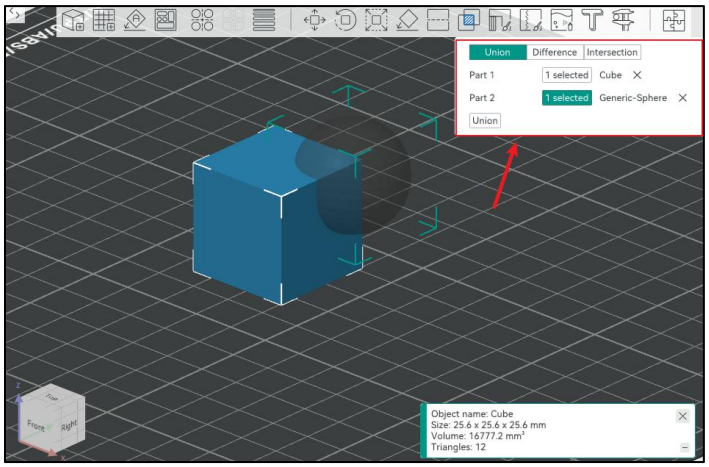
For instance, select a sphere as shown below, and move it to the appropriate position.



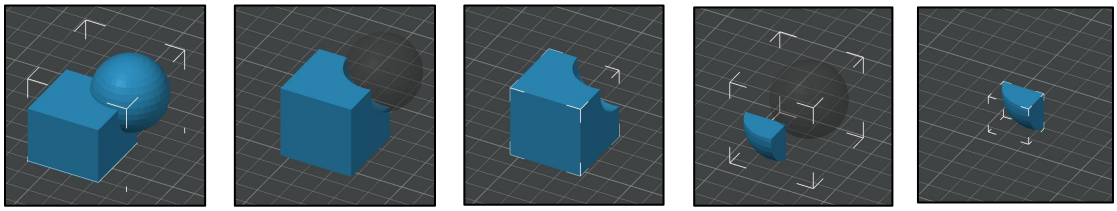
2. Re-select the original model (cube) and click on Boolean operation.



3. Set Part 1 as the original model (cube) and Part 2 as the negative part (sphere), then select the desired operation.



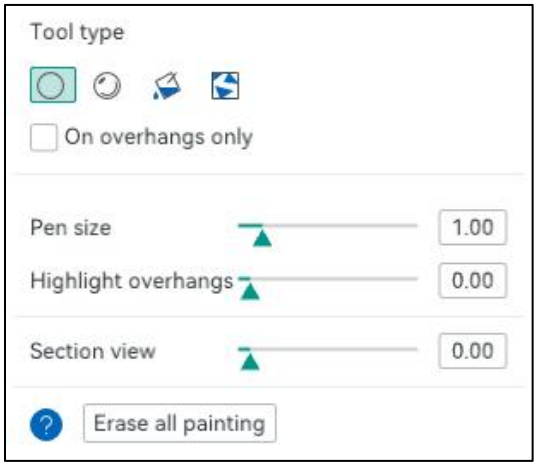
Preview of various operations



Union | Difference | Difference (Delete input) | Intersection | Intersection (Delete input)

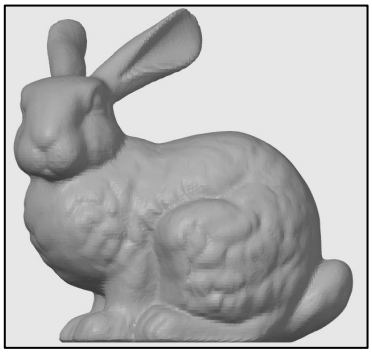
【Support painting】

Mark areas on the model where support structures are needed, which will be printed in these areas subsequently.

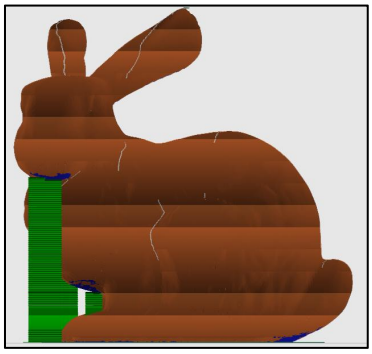


Tool type: Various drawing tools.
On overhangs only: When enabled, drawing is effective only for overhang areas.
Pen size: Adjust different pen sizes.
Highlight overhangs: Adjust the threshold for highlighting overhang areas.
Section view: Display cross-sectional views of the model from different angles.
: Click the blue question mark for related operation methods.
Erase all painting: Reset all previous paintings.

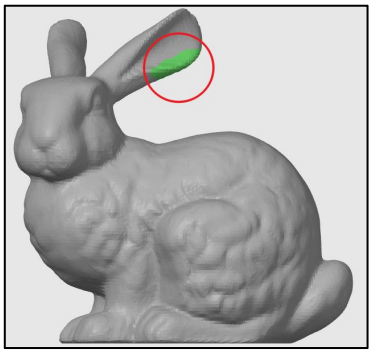
Example:



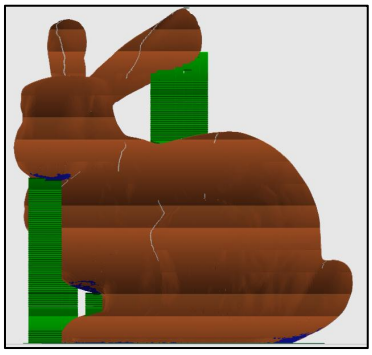
Original model



Original support position (green)



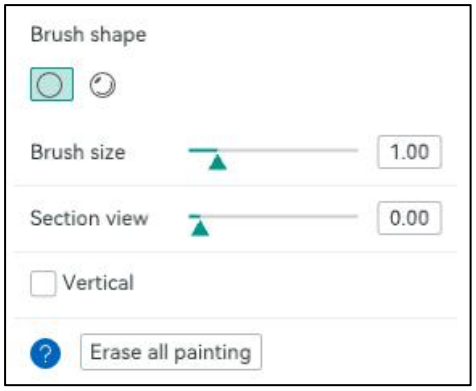
Draw a support at the ear position (green)



New support

【Seam painting】

Mark areas on the model to control where Z-seams are visible or concealed. Z-seams will be printed according to these markings.



Brush shape: Choose between round or spherical.

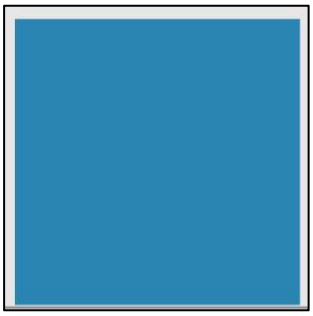
Brush size: Adjust the brush size.

Section view: Adjust the model's cross-sectional view.

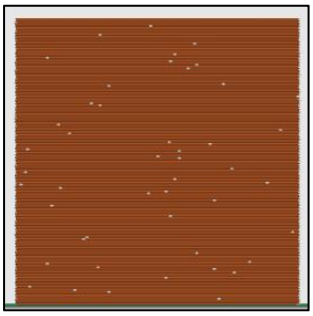
Vertical: Enable this option to restrict the brush to vertical movement only.

: Click the blue question mark for related operation methods.

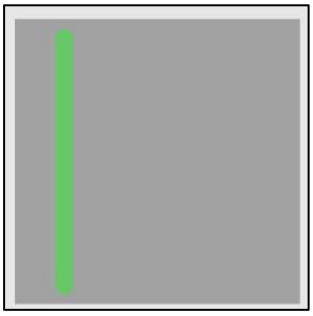
Erase all painting: Reset all previous paintings.



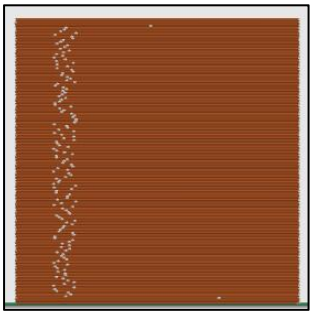
Original model (cube)



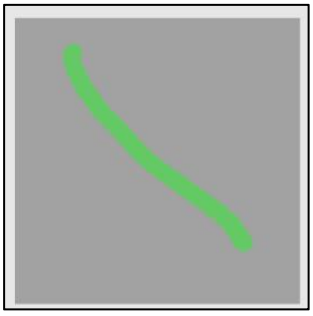
Original Z-seam (type: random)



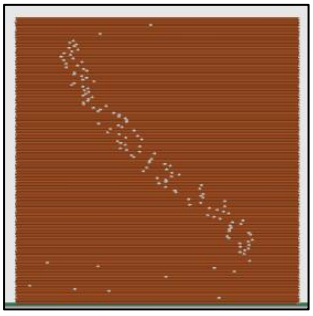
Z-seam painting (green) (vertical enabled)



Preview of Z-seam painting (green) (vertical enabled)



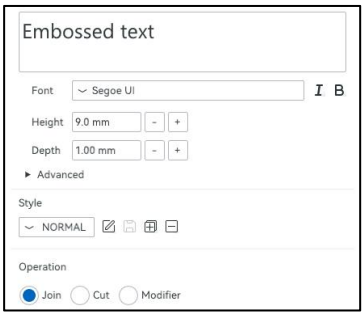
Z-seam painting (green) (vertical disabled)



Preview of Z-seam painting (green) (vertical disabled)

【Embossed】

Create embossed text on the model by entering the desired text.

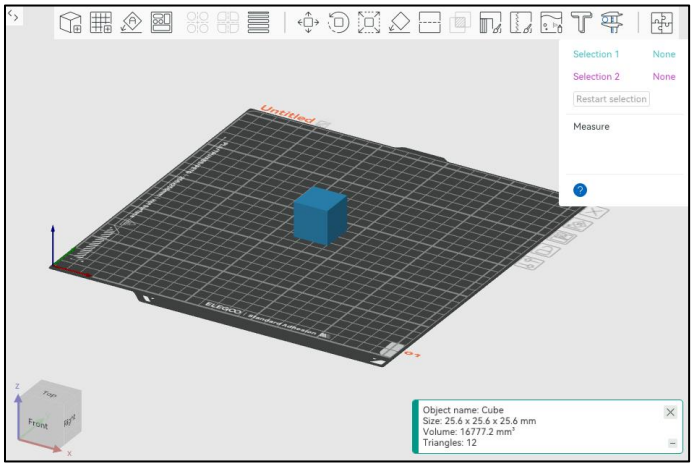


【Measure】

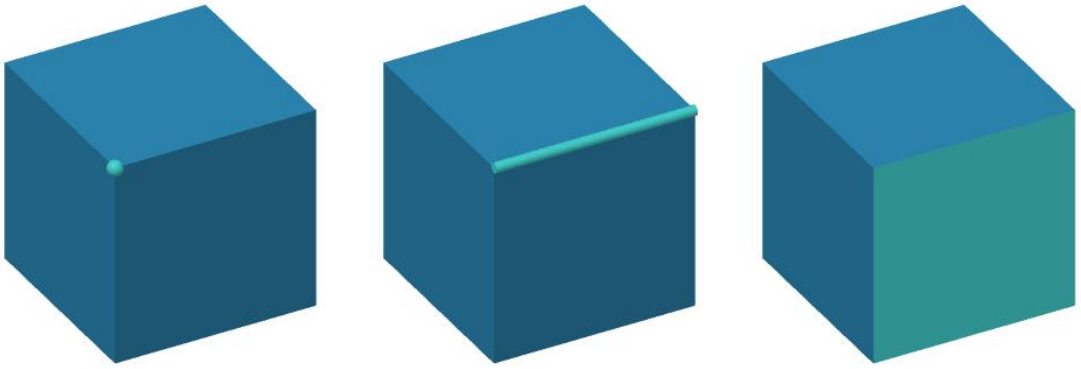
Calculate the angle and distance between two selected points on the model.

Instructions

1. Select the model first, then click on this function.

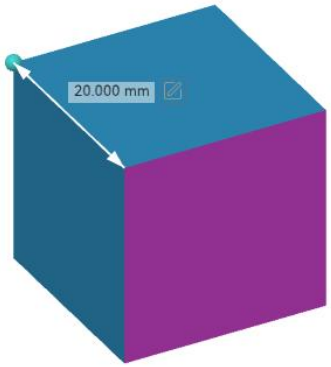


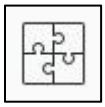
2. Choose the first point, which can be a vertex, edge, or plane.



Vertex Edge Plane

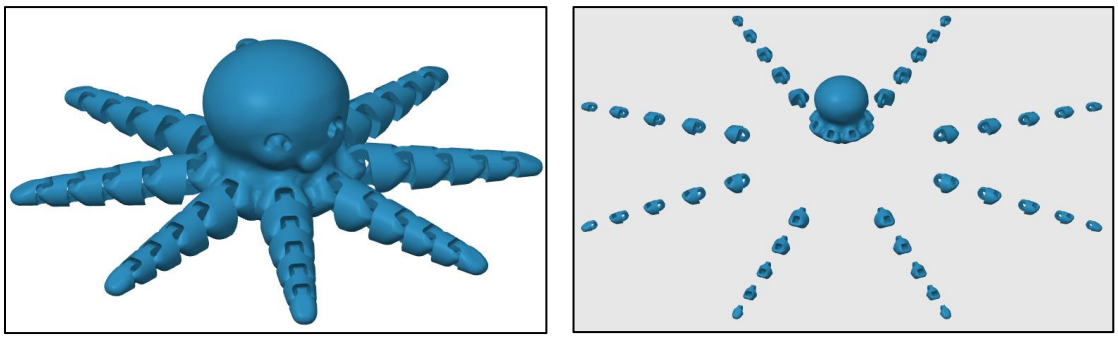
3. Then select the second point, which can also be a vertex, edge, or plane, to obtain the measurement result.





【Assembly view】

To display the model in an assembly view, split it with “Split to objects/Split to parts” functions, and then adjust the explode ratio to view the exploded assembly.



【Platform Preview】

Real-time preview of the simulated model's orientation and printing effects on the heated bed.

【3D Navigator】

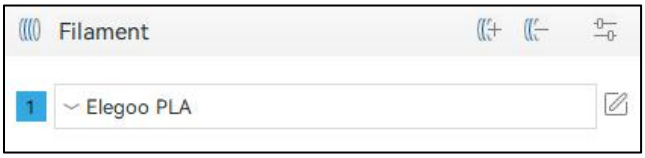
Quickly switch views in different orientations using this feature.

【Printer】

Select different printers here.

【Filament】

Select different filaments here.

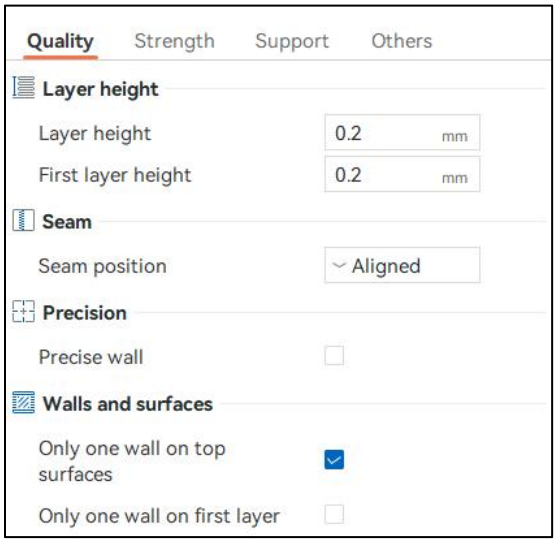


【Process】

Select different process types here, and the corresponding printing parameters will change accordingly.

【Print Parameters】

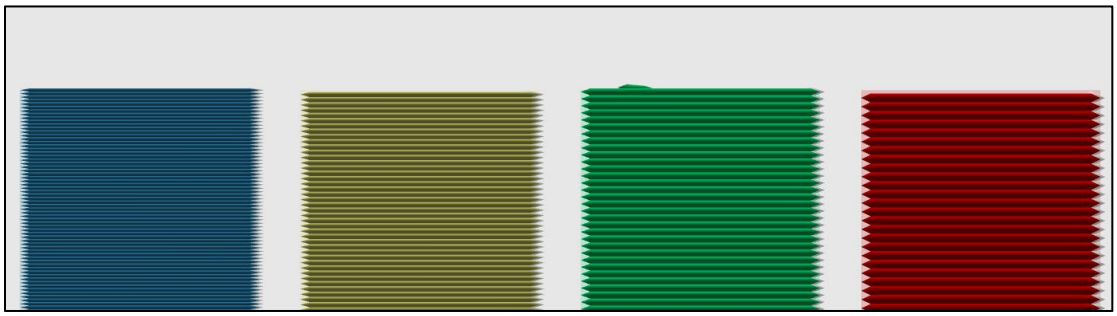
【Quality】



【Layer height】

The height of each layer of the model should be between 25% and 75% of the nozzle diameter. A smaller value results in finer surface details but longer print times, while a larger value leads to rougher surfaces and shorter print times.

Example: Below are simulated effects of a cube model with different layer heights.

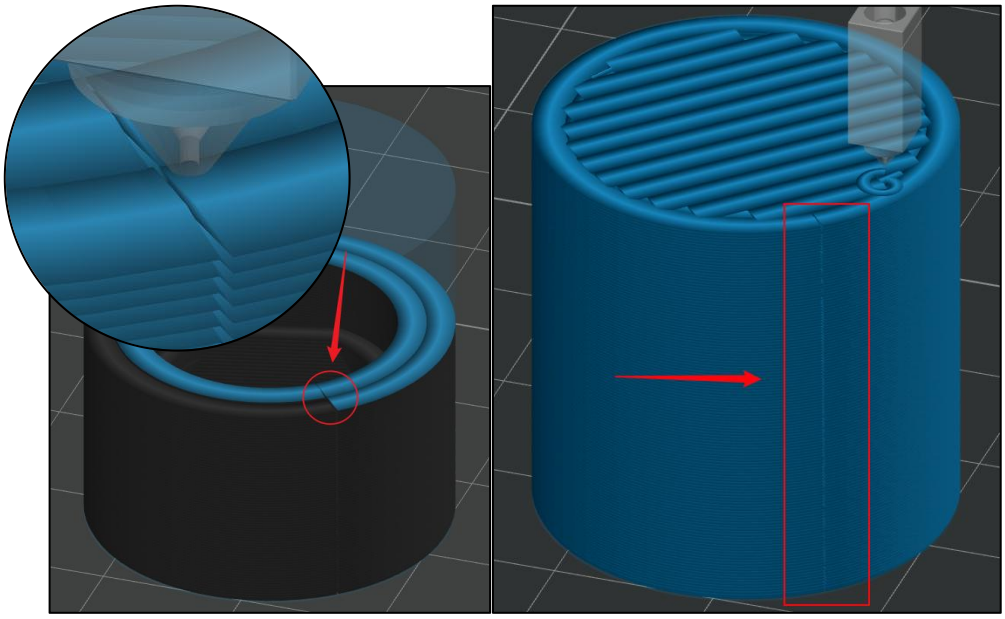


【First layer height】

This refers to the height of the first layer printed on the platform. A slightly higher first layer can improve adhesion to the heated bed.

【Seam】

Understanding what a seam is: When printing each layer of a model, there are specific starting and ending points. Due to various reasons, achieving a seamless closure at these points can be challenging. Consequently, as the layers build up, a visible gap can emerge where these points meet. This visible gap is commonly referred to as a seam or “zipper,” a normal occurrence that can be mitigated through adjustments but not completely eliminated.



【Seam position】

【Nearest】

The starting point of the next layer is the last stopping point of the nozzle on the model from the previous layer.
Pros: Less path deviation, saving print time.
Cons: Seam points are randomly generated on the model with changing print positions.
Note: Randomness isn't necessarily negative. If each seam point is well controlled, this dispersed pattern can prevent flaws from accumulating significantly, sometimes resulting in near-perfect models. However, poor control will lead to visible defects randomly scattered on the model surface.
Example:

"Nearest" type seam points (white dots in the image)

Ending point of the previous layer

Starting point of the next layer

【Aligned】

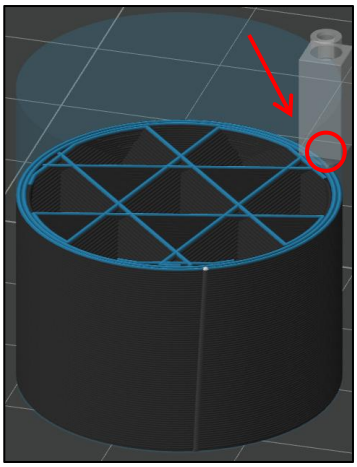
In this mode, the nozzle moves more paths to align the seam points at the same position for each layer.

Pros: Generally used as an alternative for poorly adjusted seam points, as despite not being perfect, they are concentrated together without damaging other surfaces of your model. If controlled effectively, it can result in a fairly perfect model.

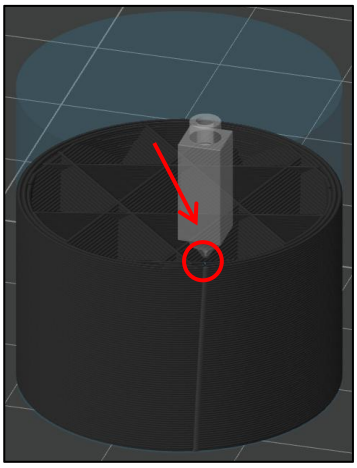
Cons: Increased print time due to additional nozzle movement for seam alignment.

Example:

"Aligned" type seam points (white in the image)



Ending point of the previous layer



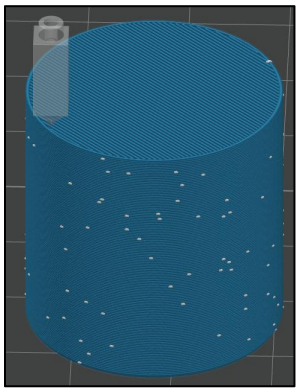
Starting point of the next layer

【Back】

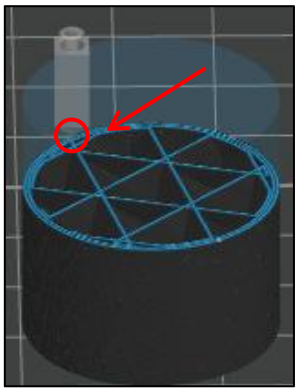
Pros: Seam points are strategically concentrated on the backside of the plate (not the back of the model, irrespective of model orientation). This option ensures that even if the seam handling is less than perfect, it does not impact the model's frontal appearance, particularly beneficial for character, animal, or display models requiring a pronounced frontal effect. With the model correctly positioned, the frontal formation remains undisturbed, maintaining the desired front-facing outcome.
Cons: Back works similarly to Aligned, resulting in similar drawbacks with centralized seam points.

【Random】

Pros & Cons: Refer to the explanation of the Nearest mode above. Generally efficient distribution reduces seam visibility for better results, while poor placement worsens seam appearance, negatively affecting overall print quality. The starting and ending point are also randomized.
Example:



"Random" type seam points (white dots in the image)



Ending point of the previous layer

Starting point of the next layer

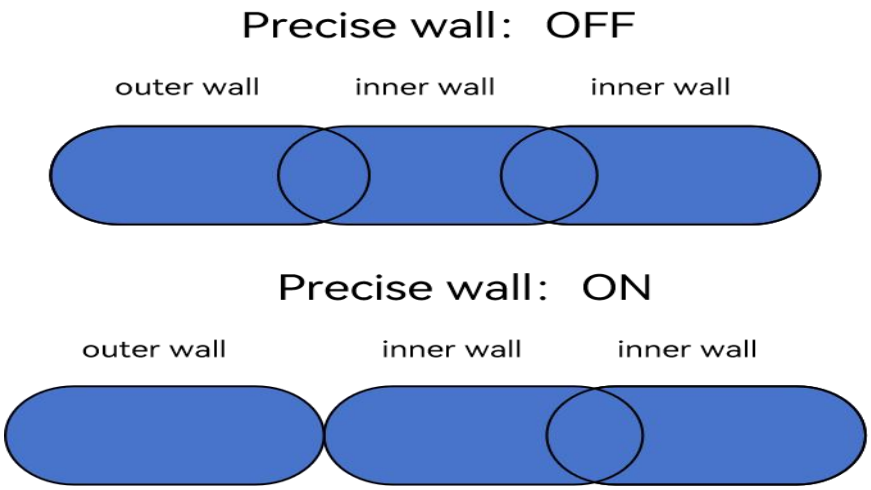
【Precision】

【Precise outer wall size】

This section refers to the official OrcaSlicer documentation. You can find the original link here: https://github.com/SoftFever/OrcaSlicer/wiki/Precise-wall.
"The 'Precise Wall' is a distinctive feature introduced by OrcaSlicer, aimed at improving the dimensional accuracy of prints and minimizing layer inconsistencies by slightly increasing the spacing between the outer wall and the inner wall.

Below is a technical explanation of how this feature works. First, it's important to understand some basic concepts like flow, extrusion width, and space. Slic3r has an excellent document that covers these topics in detail. You can refer to this article: https://manual.slic3r.org/advanced/flow-math.

Now, let's dive into the specifics. Slic3r and its forks, such as PrusaSlicer, SuperSlicer, and OrcaSlicer, assume that the extrusion path has an oval shape, which accounts for the overlaps. For example, if we set the wall width to 0.4 mm and the layer height to 0.2 mm, the combined thickness of two walls laid side by side is 0.714 mm instead of 0.8 mm due to the overlapping.



This approach enhances the strength of 3D-printed parts. However, it does have some side effects. For instance, when the inner-outer wall order is used, the outer wall can be pushed outside, leading to potential size inaccuracy and more layer inconsistency.

It's important to keep in mind that this approach to handling flow is specific to Slic3r and its forks. Other slicing software, such as Cura, assumes that the extrusion path is rectangular and, therefore, does not include overlapping. Two 0.4 mm walls will result in a 0.8 mm shell thickness in Cura. OrcaSlicer adheres to Slic3r's approach to handling flow. To address the downsides mentioned earlier, OrcaSlicer introduced the 'Precise Wall' feature. When this feature is enabled in OrcaSlicer, the overlap between the outer wall and its adjacent inner wall is set to zero. This ensures that the overall strength of the printed part is unaffected, while the size accuracy and layer consistency are improved."

【Walls and surfaces】

In this context, the term “walls” specifically refers to the model’s outer shell.

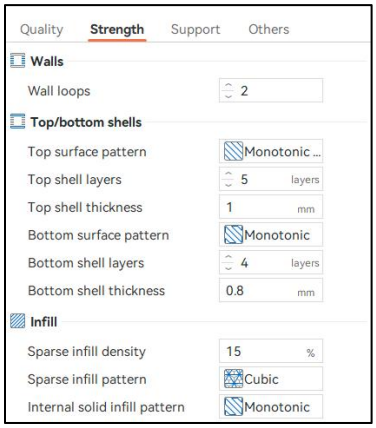
【Only one wall on top surfaces】

Generally, the top of the model consists of two or more wall layers. With this feature, only one wall layer is printed, with the remaining parts filled or made solid, contributing to a more consistent and visually pleasing top surface.

【Only one wall on first layer】

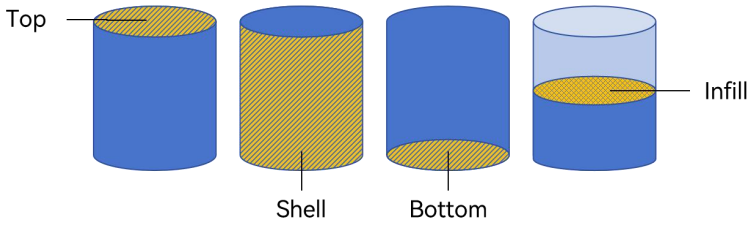
While similar to having a single wall on the top surface, using only a single wall for the first layer is not recommended. This is because subsequent model constructions rely heavily on the printing quality of the first layer. The poor stability of the outer wall of the first layer can easily lead to printing failures. Moreover, the first layer is typically dense enough to ensure a smooth bottom, reducing the need for a single-wall structure.

【Strength】



【Walls】

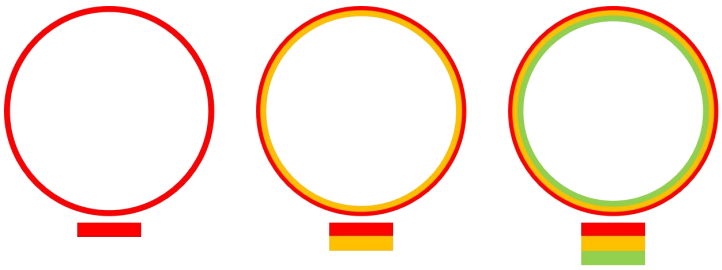
In general, a model consists of a bottom, a top, a shell, and internal infill. In this context, the term "walls" specifically refers to the model's outer shell.



【Wall loops】

It is recommended to set a minimum of 2 wall loops. Too few loops can lead to a fragile outer shell, while too many loops can unnecessarily increase print time.

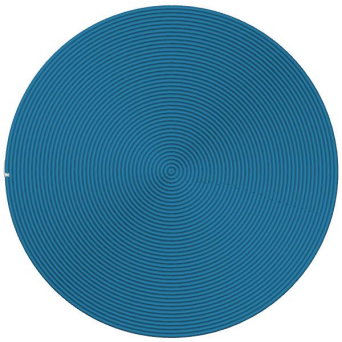
Example: Below is a preview showing a hollow model's cross-sectional with 1, 2, and 3 wall loops from left to right.



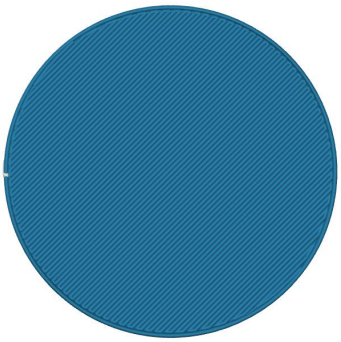
【Top/bottom shells】

【Top surface pattern】

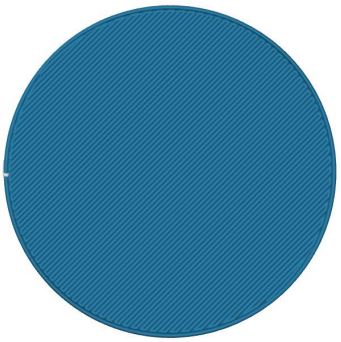
This refers to the pattern used to fill the top surface. Below are eight types illustrated, with rectilinear, monotonic, monotonic line, and aligned rectilinear appearing almost identical when densely arranged.



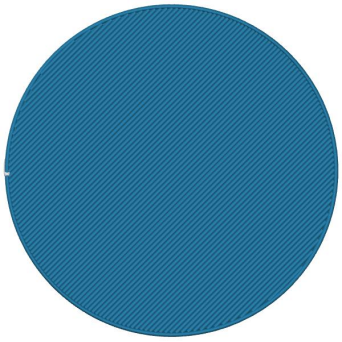
Concentric



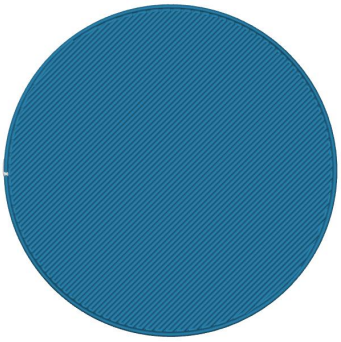
Rectilinear



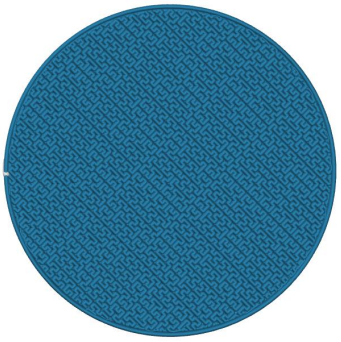
Monotonic



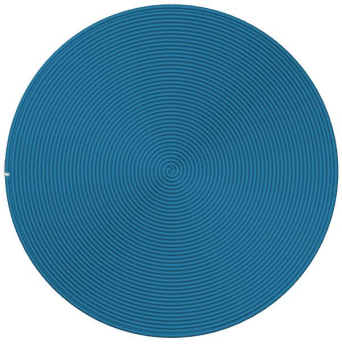
Monotonic line



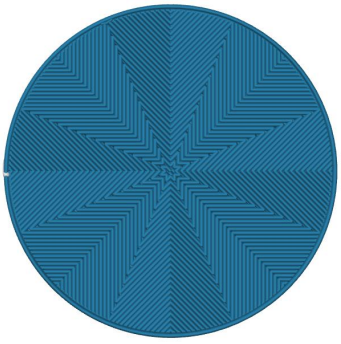
Aligned Rectilinear



Hilbert Curve



Archimedean Chords



Octagram Spiral

【Top shell layers】

The recommended range for top shell layers is between 3 and 5. Increasing the number of layers can result in a denser top surface and a smoother appearance. Avoid excessive layers to prevent material wastage.

【Top shell thickness】

This refers to the thickness of the solid top infill. If the calculated thickness (top shell layers * layer height) is less than this value, the software will automatically increase the top shell layers. A value of 0 disables this setting, making the top shell thickness solely determined by the top shell layers.

【Bottom surface pattern】

【Bottom shell layers】

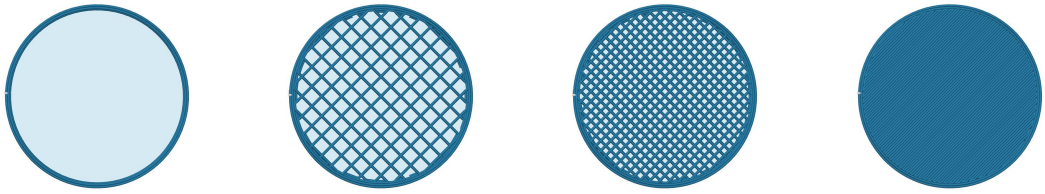
【Bottom shell thickness】

In line with the top settings detailed earlier, the above three options apply specifically to the bottom surface.

【Infill】

【Sparse infill density】

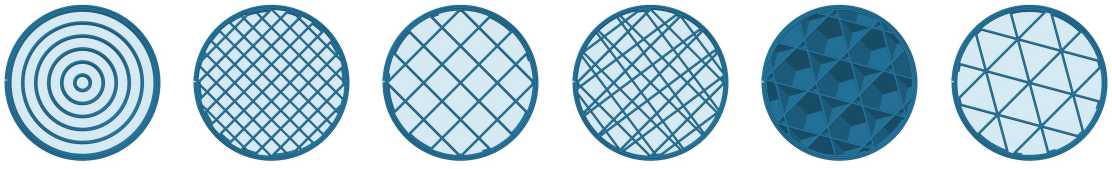
Models are typically not solid inside to save material and time. A density of 15% to 20% is usually sufficient to support most model structures. See the diagrams below for different infill densities:



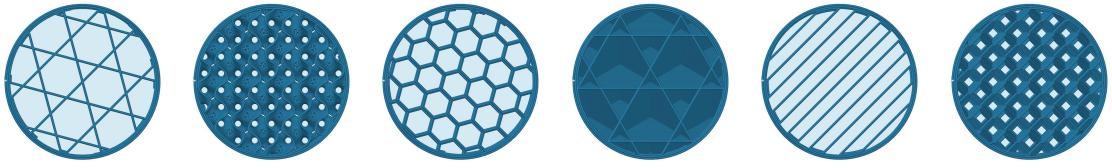
0% hollow　33%　66%　100% solid

【Sparse infill pattern】

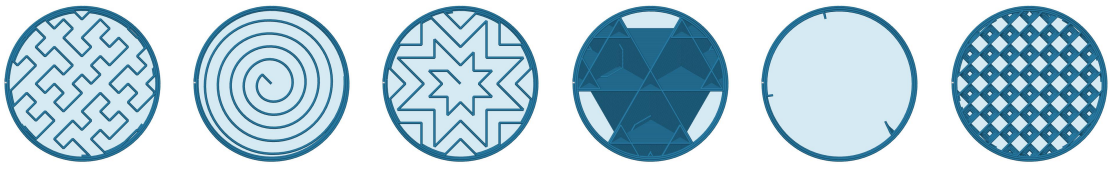
Infill can be set to different patterns, as shown in the illustrations below:



Concentric　Rectilinear　Grid　Line　Cubic　Triangles



Tri-hexagon　Gyroid　Honeycomb　Adaptive Cubic　Aligned Rectilinear　3D Honeycomb

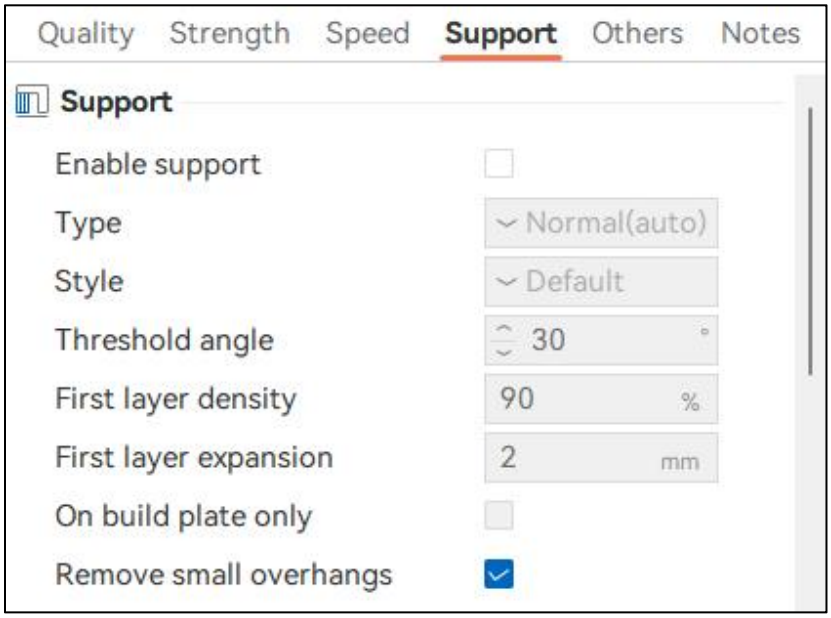


Hilbert Curve　Archimedean Chords　Octagram Spiral　Support Cubic　Lightning　Cross Hatch

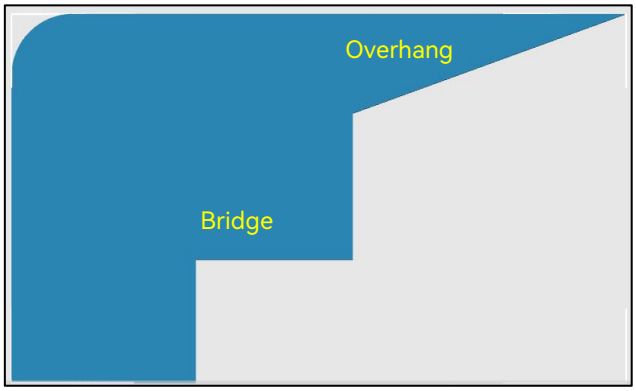
【Internal solid infill pattern】

This refers to the pattern used for solid infill inside, which is the same as the top shell pattern.

【Support】

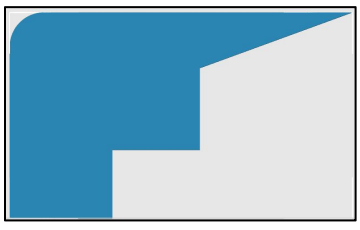


When there are significant overhangs or bridges in the model, it is essential to add proper supports during printing to prevent print failures caused by the collapse of these sections.

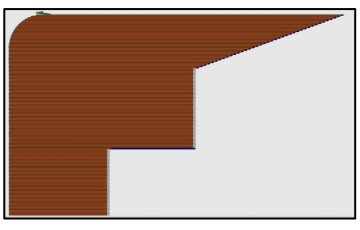


【Enable support】

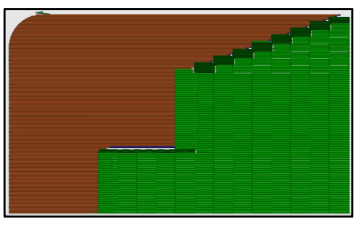
Check to enable support.



Original model



Without support



With support

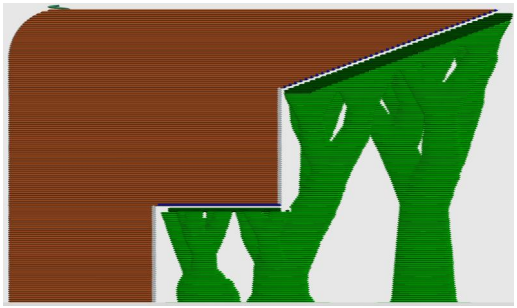
【Type】

Auto/manual

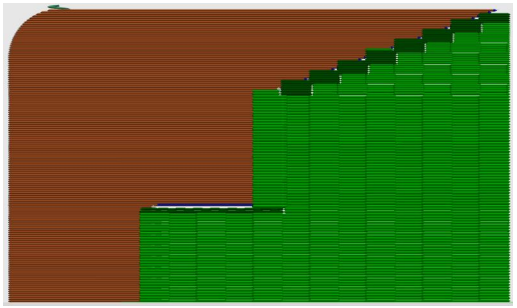
Auto: Supports are automatically generated by the software based on threshold angles.

Manual: Supports are only generated on enforcer faces (as described earlier in  “Support painting”).

Tree/normal support



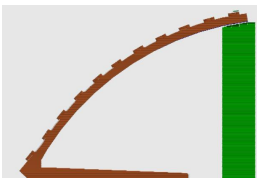
Tree support



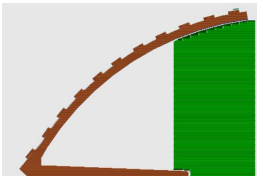
Normal support

【Threshold angle】

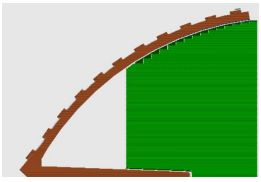
This setting determines the angle at which supports are generated for overhangs.



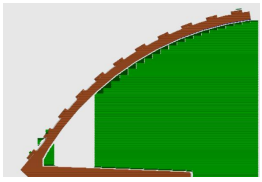
10°



20°



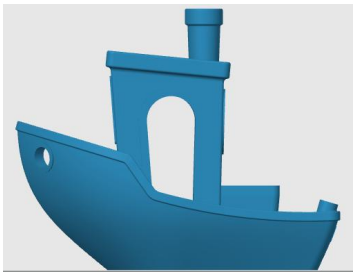
30°



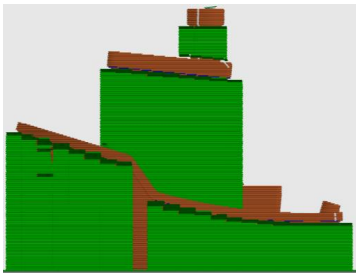
40°

【On build plate only】

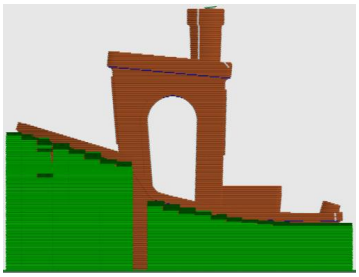
Support structures are generated only between the platform and the model, not within the model itself.



Original model

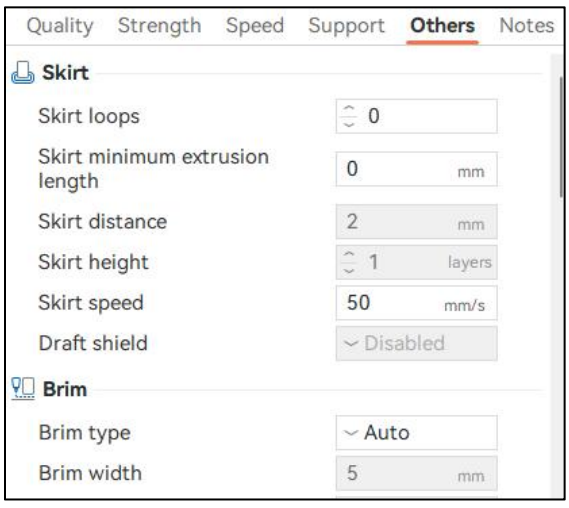


On build plate only: Off



On build plate only: On

【Others】

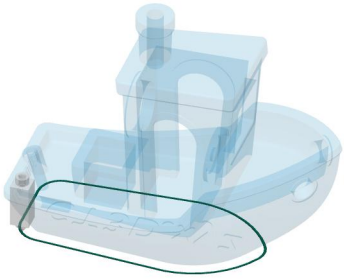


【Skirt】

【Skirt loops】

Before printing the model, a line or a series of loops is printed around the the model's outer perimeter without making direct contact, known as a skirt.

Example:



Skirt (green line)

Skirt and first layer

Skirt and model

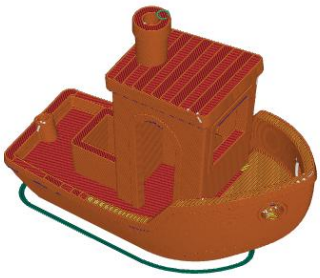
Purpose:

- Clean the Nozzle: Extrude the residual waste, ensuring a clean nozzle before actual printing.
- Prime the Nozzle: Pre-fill the nozzle for smoother subsequent extrusions.
- Indicate Orientation: Serve as a visual reference for model orientation and size.

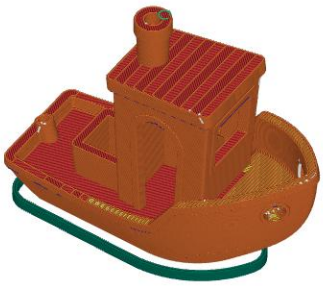
For most prints, using 1-2 skirt loops is typically sufficient.

【Skirt height】

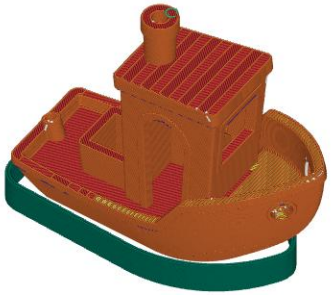
The number of skirt layers printed before the main print starts.



Skirt height: 1 layer



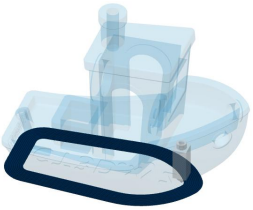
Skirt height: 10 layers



Skirt height: 30 layers

【Brim】

The brim refers to one or more additional layers of material printed directly attached to the base of the model.



Brim width: 5 mm

Brim and first layer

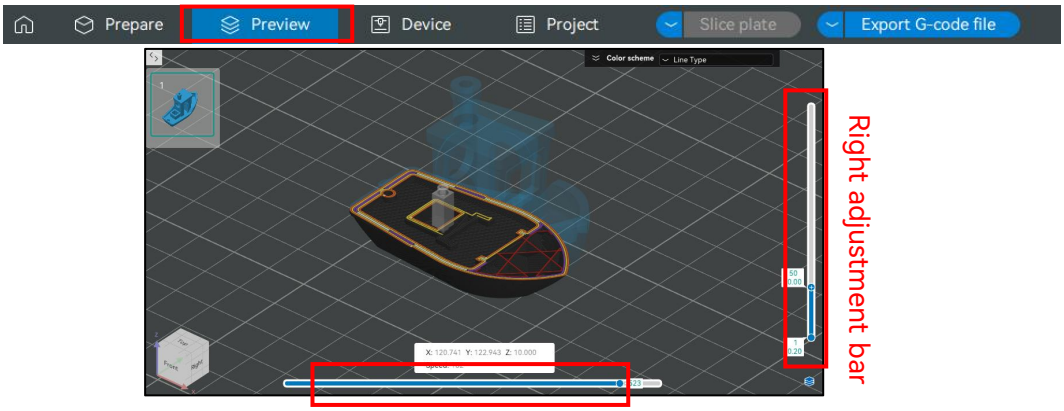
Brim and model

Brim width: 10 mm

Purpose: The primary function of a brim is to enhance adhesion during printing by increasing the model's contact area with the build plate. While it serves several purposes similar to the skirt, the key distinction lies in Brim's direct contact with the model, requiring manual removal post-printing. Typically set as a single layer with a width of 5 - 8 mm, this parameter can be adjusted based on the model size and specific adhesion needs.

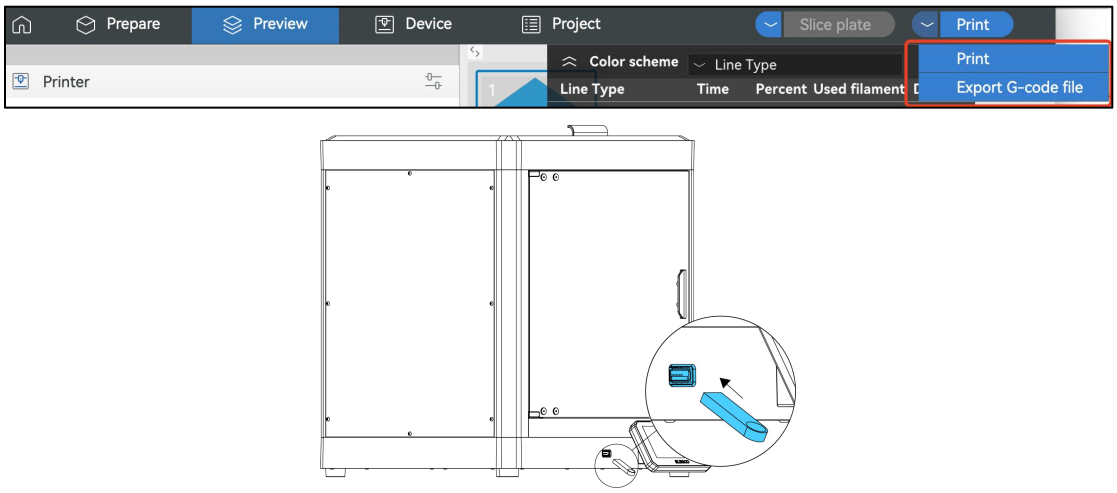
【Printing Method】

After setting the parameters, preview the model's printing status using the adjustment bars on the right and bottom.



Right adjustment bar: Preview printing at different heights.
Bottom adjustment bar: Preview printing at different positions on the current layer.

After confirming that the preview is correct, you can export the file to a USB drive and insert it into the printer to start printing. Alternatively, you can connect your printer to upload the file directly for immediate printing.



【Device Connection】

1. Click the button in the Prepare menu to open the connection window.
2. Select the appropriate host type that matches your printer. For example, if you are using the Centauri Carbon, choose “Elegoo Link.”
3. Enter the IP address of the connected printer, which can be found on the printer’s display. Make sure your computer is connected to the same network as the printer.
4. Click OK to complete the connection process.