

Photon Workshop

User Manual

Team **ANYCUBIC**

Copyrighted by "Shenzhen Anycubic Technology Co., Ltd ", all rights reserved.

Contents

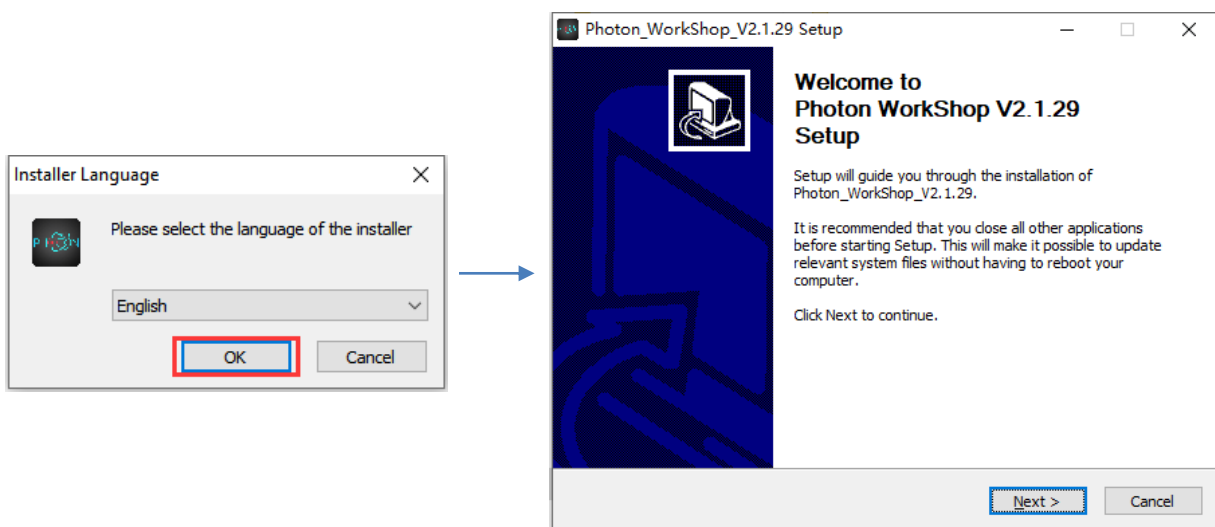
1. Installation -----	3
2. Overview -----	4
3. Settings -----	5
(1) Import -----	5
(2) Machine & Resin Settings -----	5
(3) Slice Parameter -----	6
4. Introduction to Functions -----	11
(1) View Changing -----	11
(2) Model Changing -----	11
(3) Hollow and Fill -----	12
(4) Punching -----	13
(5) Text Paste -----	14
(6) Split Model -----	15
(7) 3D Face Reconstruction -----	16
5. Support Settings -----	18
(1) Shape Editing -----	19
(2) Support Adding -----	22
(3) Export Support Information -----	26
6. Export Sliced File -----	27

Photon Workshop Installation

1. Photon Workshop Installation

Photon Workshop installation package is located in memory stick. (You may have to close the anti-virus software before installing the slicing software.) Double click “Photon_WorkShop_V2.1.29.exe”, and then follow the installation guide as shown below.

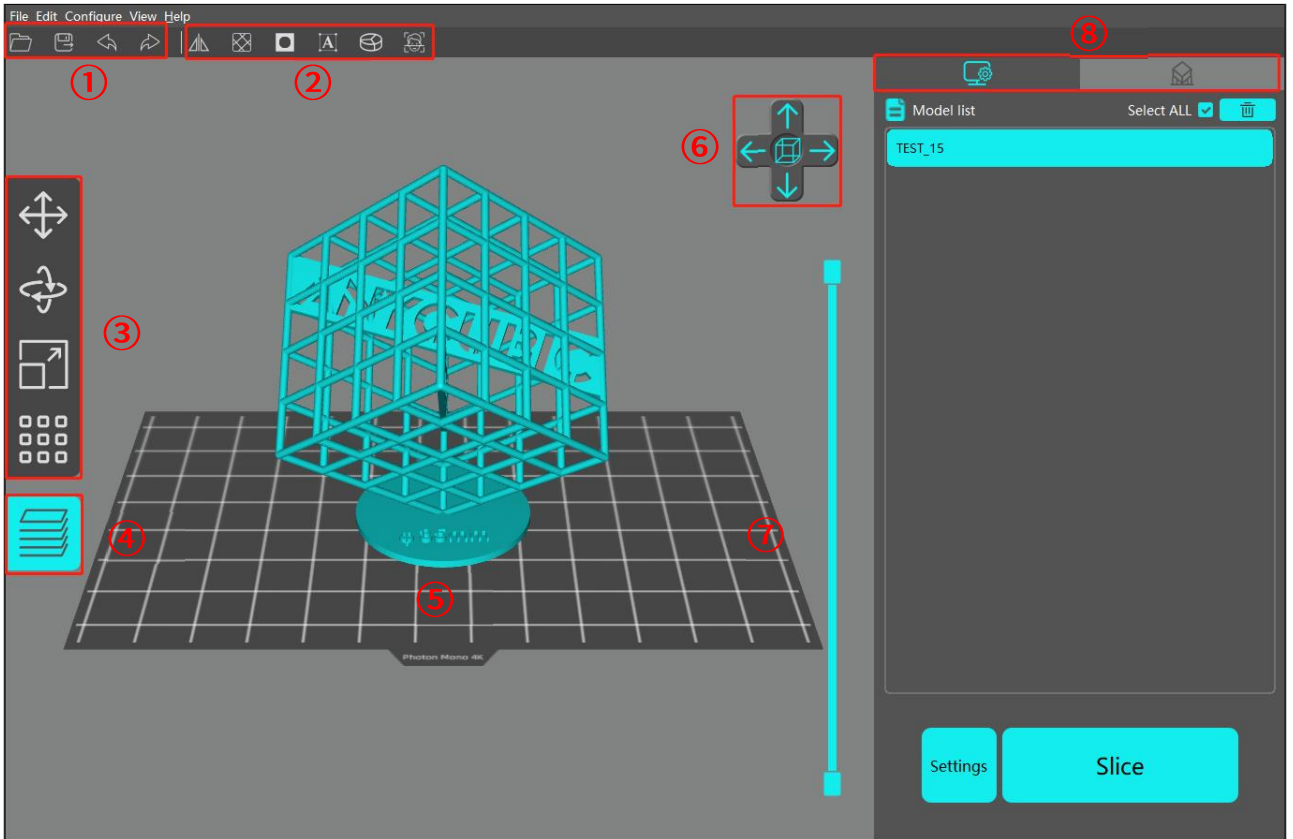
For Mac PC, double click “Photon_WorkShop_V2.1.29.dmg” to install the slicing software.



Note: ANYCUBIC may update the software and firmware without notification. Please visit www.anycubic.com for the latest updates.

Overview

2. Overview



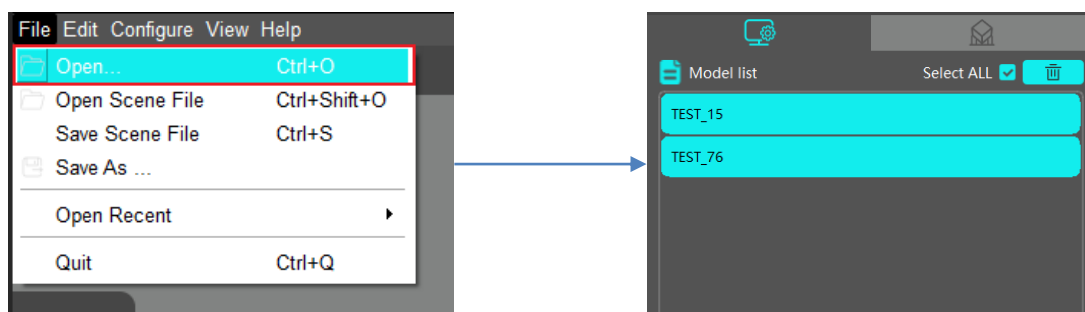
- ① From left to right are open, save, undo and redo.
- ② From left to right are model mirror, hollow and infill, punching, text paste, spilt model and 3D Face Reconstruction.
- ③ Manipulations of model include move, rotate, scale and layout.
- ④ Click to export sliced file.
- ⑤ 3D model preview.
- ⑥ Click to switch views.
- ⑦ Drag the slider to preview each layer of the model.
- ⑧ Switch to model settings or support parameters.

Settings

3. Settings

(1) Import

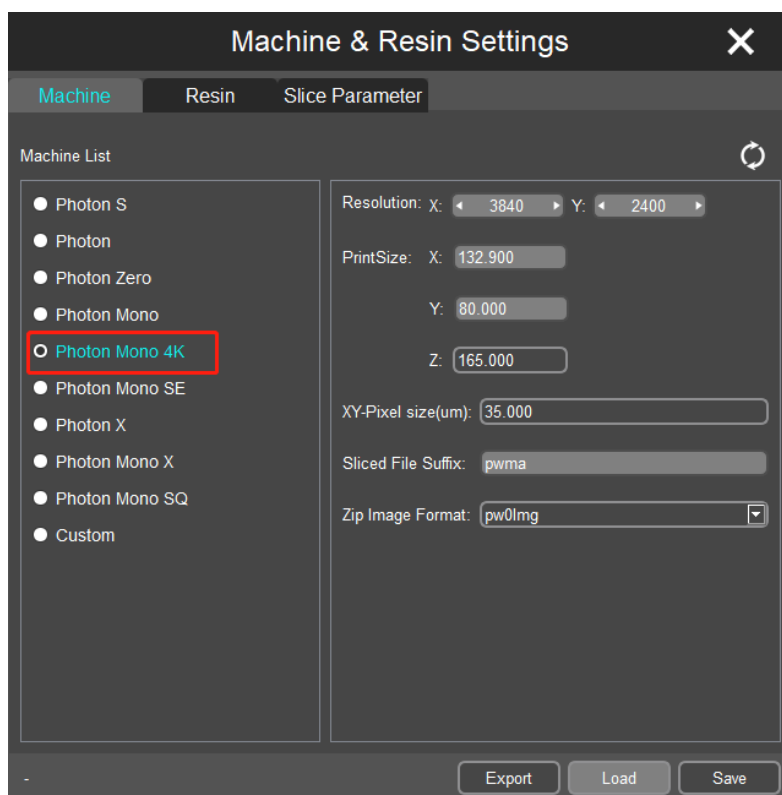
On the menu bar, click “File” → “Open” to import your own three-dimensional format model, i.e. STL file or OBJ file, and then select or delete the models from model list.



(2) Machine & Resin Settings

① Machine settings

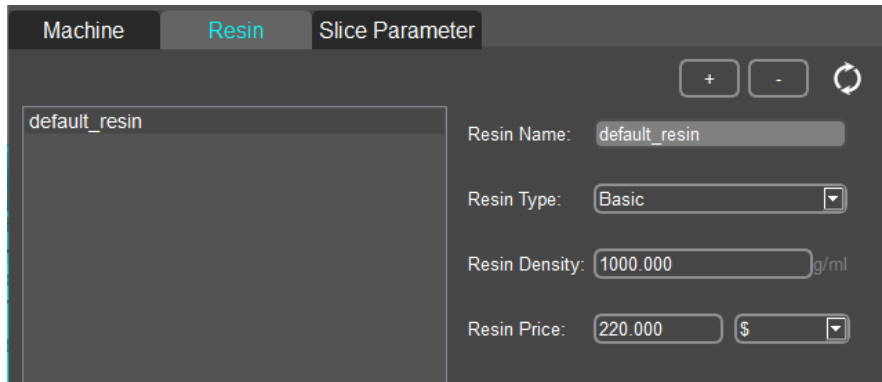
Click “Settings” in the lower right corner and select the type of your machine in machine interface. Different machine types have different parameters, please choose the machine you use to avoid print failure.



Settings

② Resin Settings

In resin interface, choose the resin type you use from “Basic”, “Plant-based” and “Dental Castable” or click “+” to add your resin type. Then click “Save” at the bottom right corner.



(2) Slice Parameter

Set slice parameters according to your requirement in this interface. Then click “Save” to apply it, or click “Export” to export profile.

① Slice parameter instruction

- **Layer Thickness:** The thinner the layer, the better the accuracy of Z-axis direction. The thicker the layer, the longer the exposure time for each layer.
- **Normal Exposure Time:** The length of normal exposure time depends on UV power, complexity of model, resin materials and so on. Underexposure may cause uncured detail, overexposure may affect accuracy of model.
- **Off Time:** The UV light will be off for a certain time between each layer. The longer off time allows resin with poor fluidity to reflow.

Settings

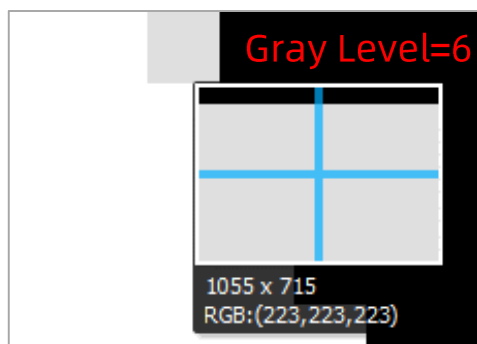
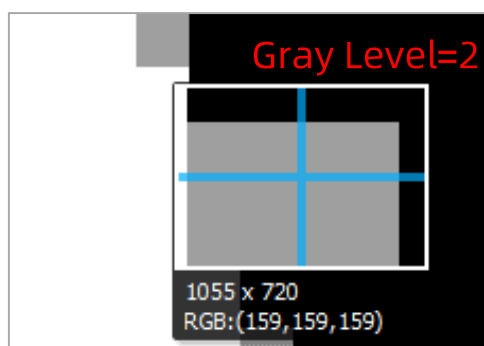
- **Bottom Exposure Time:** The longer the bottom exposure time is, the easier the bottom layers of the model stick to platform.
- **Bottom Layers:** The bottom layers need to be exposed for longer time to stick model to platform tightly. The bottom layers may be larger than normal layers.
- **Z Lift Distance:** The model requires enough distance to be separated from the FEP film.
- **Z Lift Speed:** If the lift speed is too fast, the model will be broken and supports may also be damaged due to the separation force.
- **Z Retract Speed:** If the retract speed is too fast, the printing quality may be damaged.
- **Anti-alias:** A higher grade of anti-alias value could enhance the ability to smooth the edges for each layer during printing, thereby resulting better surface of the printed objects. A higher grade of anti-alias value also means longer slicing time and larger files.
- **Surface Abrazine:** Only when anti-alias value is 1 can you check this option to get a matte surface.

Note: The recommended parameters is shown in User Manual:
[Technical Specification](#)→ [Recommended Printing Parameters](#).

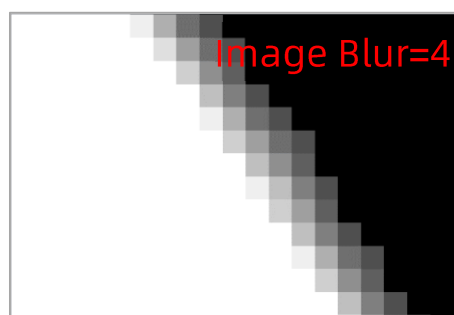
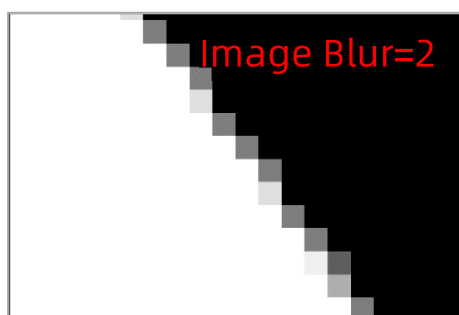
When anti-alias value is larger than 1, you can set the grades of gray level and image blur according to the requirement.

Settings

- **Gray Level:** The higher the gray level, the brighter the pixels of anti-alias are.



- **Image Blur:** It blurs the edge of image to achieve the natural cohesion. The higher the grade of image blur, the more blurry it is.

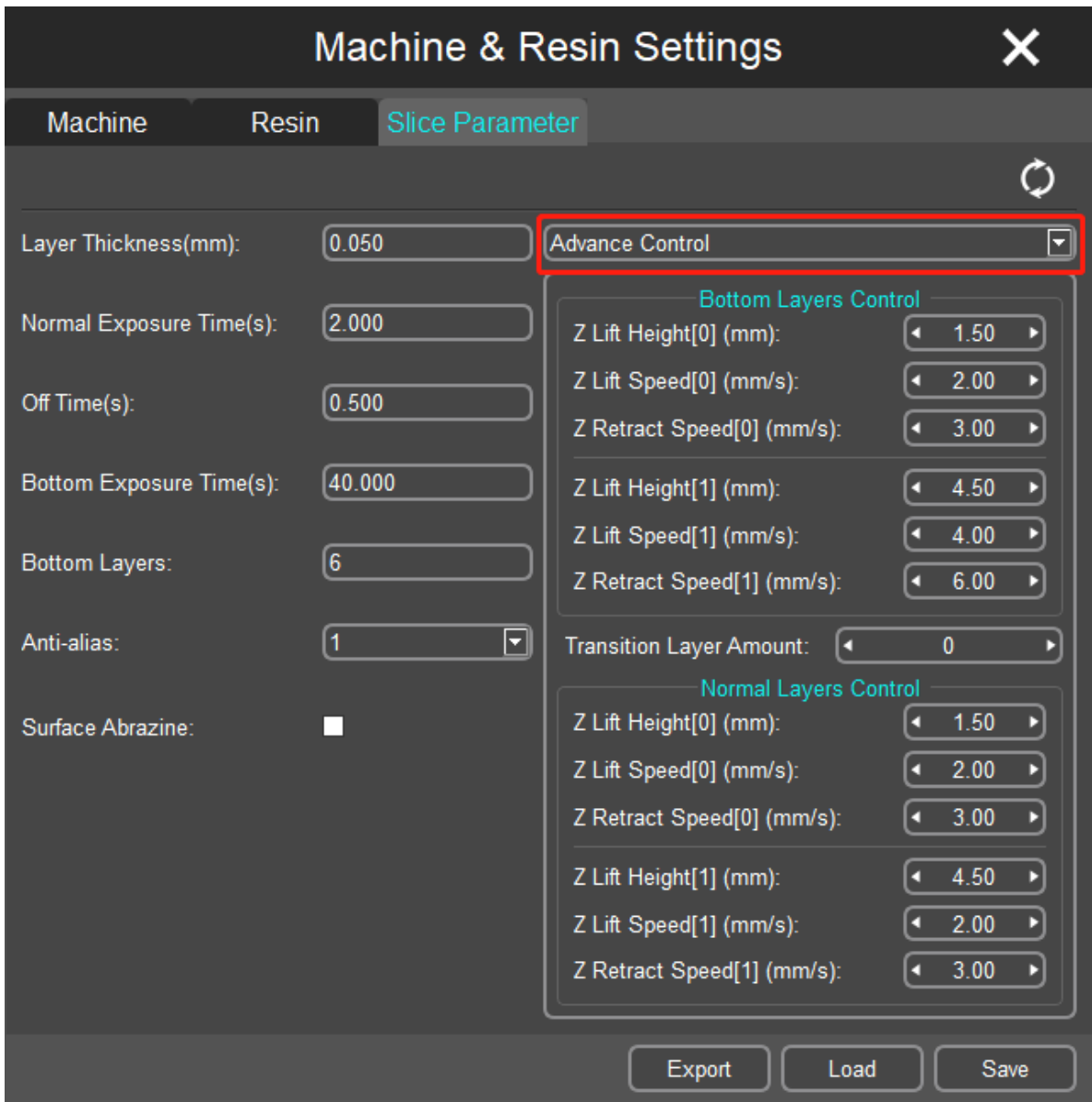


In the process of printing, please comprehensively consider the grade of anti-alias, gray level and image blur according to the actual requirement to get the best surface quality.

② Advance control mode

In basic control mode, Z lift time, Z lift speed and retract speed of the bottom layers are the same with those of normal layers. However, if you want to reduce the printing time or achieve better print effect, switch to advance control mode to set different parameters of Z-axis moving in different stages and layers.

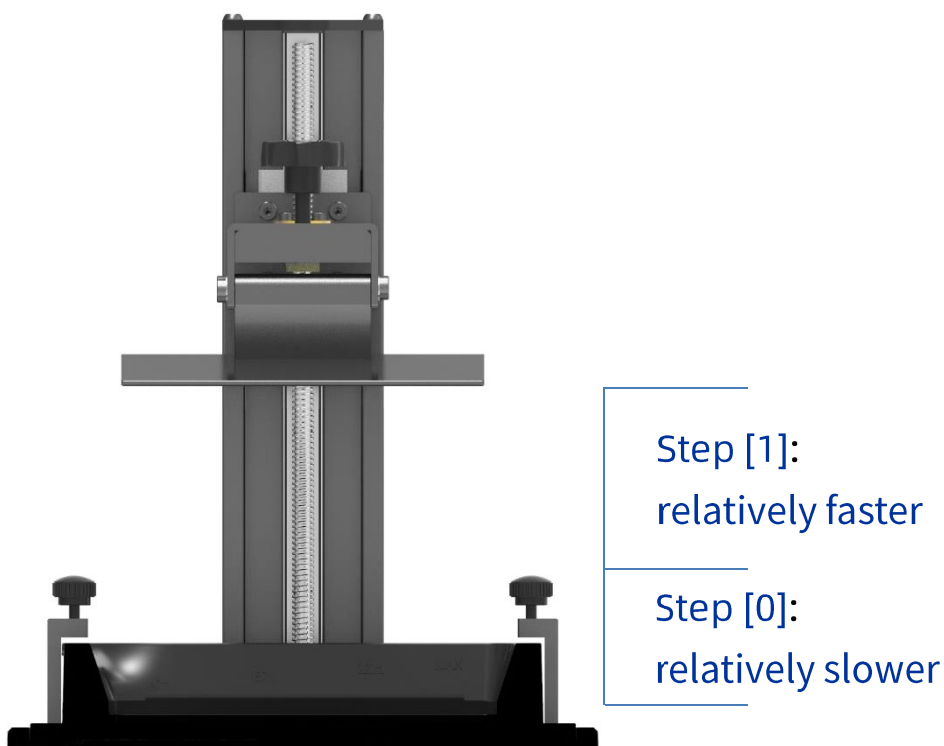
Settings



- **Bottom layers control:** To set Z lift height, Z lift speed and Z retract speed of the bottom layers.
- **Normal layer control:** To set Z lift height, Z lift speed and Z retract speed of the normal layers.
- **Transition layer count:** The transition layers between the bottom layers and normal layers. The more the transition layers are, the longer the time transition costs.

Settings

- **Step [0]:** The stage when printing platform is moving near the curing face. The speed of this stage is relatively slow to avoid affecting the printing.
- **Step [1]:** The stage that printing platform is moving away from the curing face. The speed of this stage is relatively fast to shorten the printing time.



Tips:

1. Currently, only **Photon Mono 4K** support advance control.
2. Each Z lift height in Step[0] and Step[1] corresponds to distance of two printing platform's motion.

Introduction to Functions

4. Introduction to Functions

(1) View Changing

① View changing by mouse: Scroll the mouse wheel to zoom in/out; left click the platform and drag to move it; right click the platform and move to change view angle.

② View changing by interface controls: click the arrow to show different views.

(2) Model Changing



Move selected: click “move” icon, input a number or manipulate the controls can move the model. You also can center or reset the model.

Rotate selected: click “rotate” icon, input a number or manipulate the controls can rotate the model. You also can reset the model.

Scale selected: click “scale” icon, input a number or percentage or manipulate the controls can scale the model. You also can set the model to its maximum size.

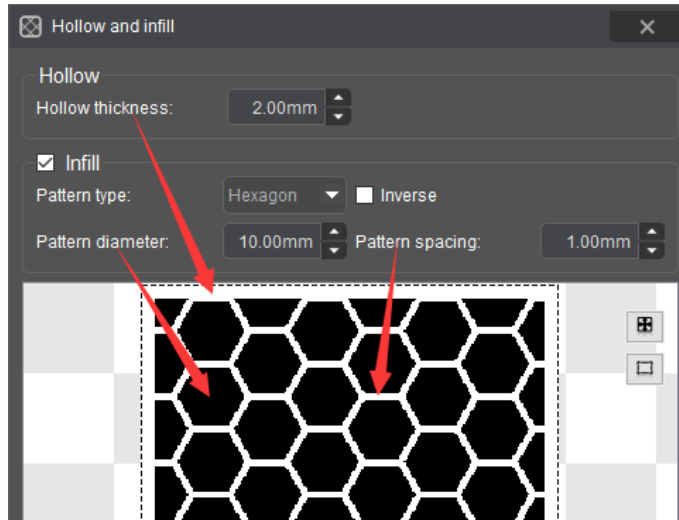
Layout models: click “layout” icon, you can duplicate the model and arrange the models in X or Y direction.

NOTE: The part out of the print range will turn dark grey, which is not printable.

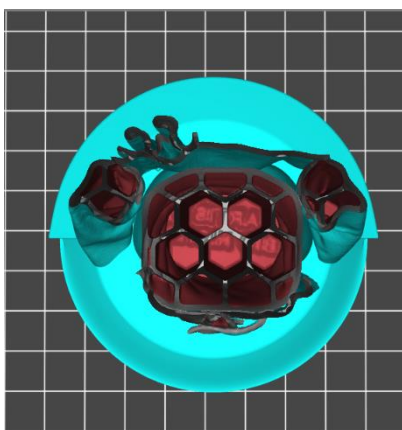
Introduction to Functions

(3) Hollow and Fill

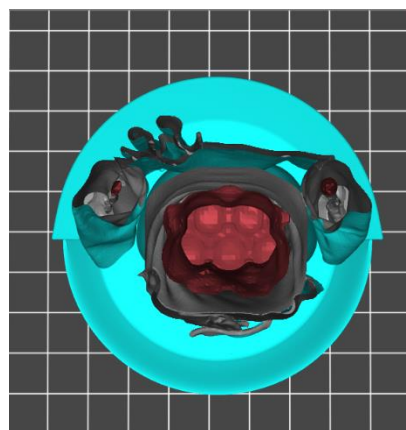
In some cases, you don't need the model to be completely filled. Before you start slicing a model, you can hollow to reduce resin consumption.



Please check and choose the most suitable parameters for hollowing and infilling to fit your requirement.



Hollow thickness: 1mm

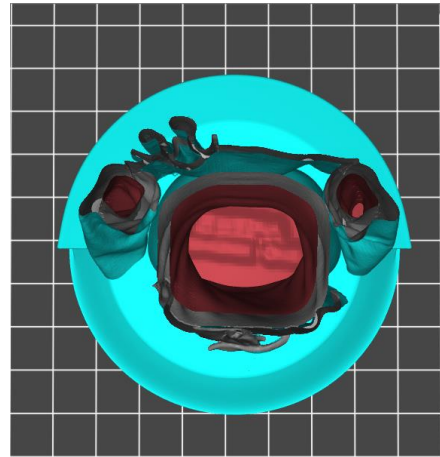
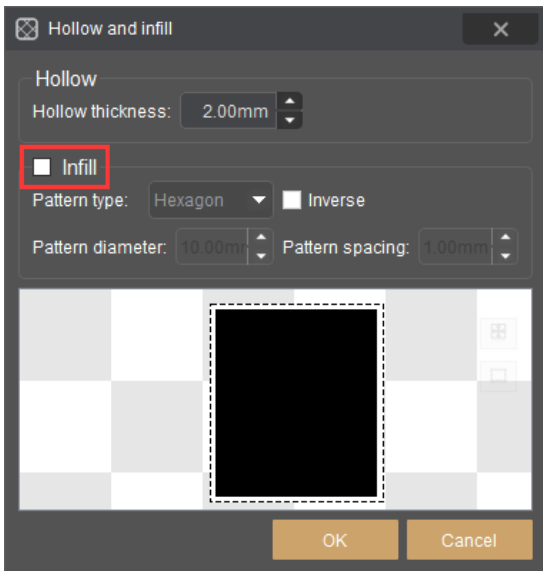


Hollow thickness: 3mm

The model name: MIA

The author of the model: Fabio Nishikata

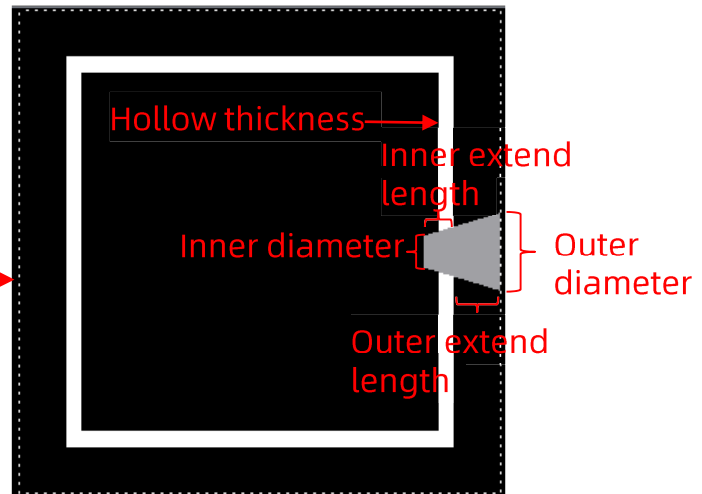
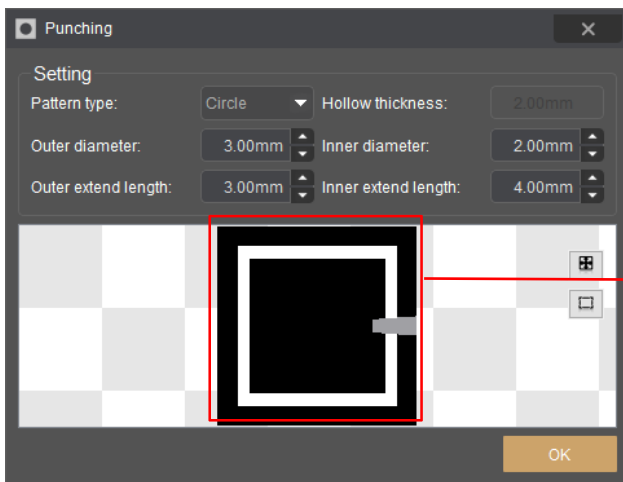
Introduction to Functions



Hollow thickness: 2mm, no infill

(4) Punching

Although the model is hollowed, there will still be part of the resin remain in the model after printing. Pinch to discharge the resin inside the model can prevent model from cracking.



The "Inner extend length" must be larger than the "Hollow thickness", so that the model can be pierced when punching and the resin can discharge from the model. Set the parameter of the hole first, then left click on the model to pinch. Click "OK" to finish.

Introduction to Functions

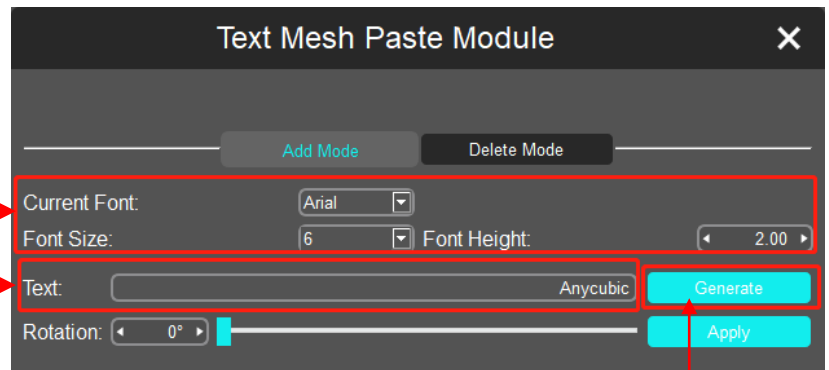
(5) Text Paste

You can paste text on the model with this feature.

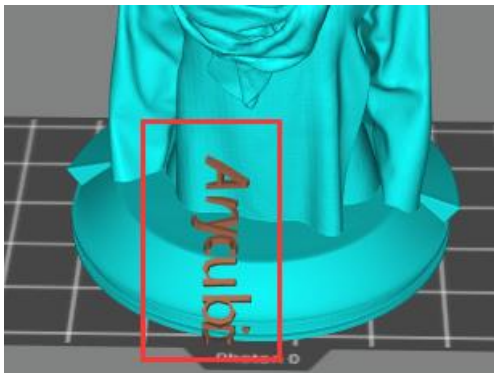
● Add mode

Set the text style

Input the text



Click to generate mesh

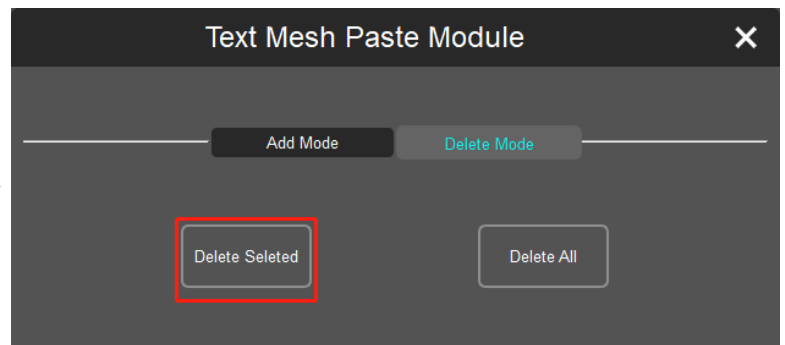
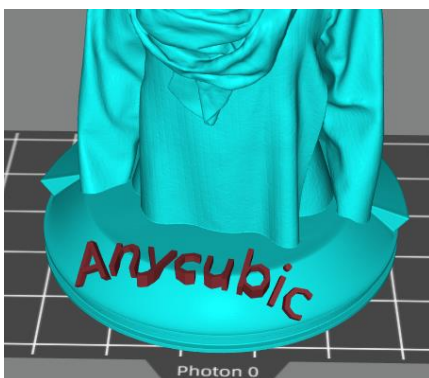


Left click on the model.

Drag the slider or input the number to rotate the text.

● Delete mode

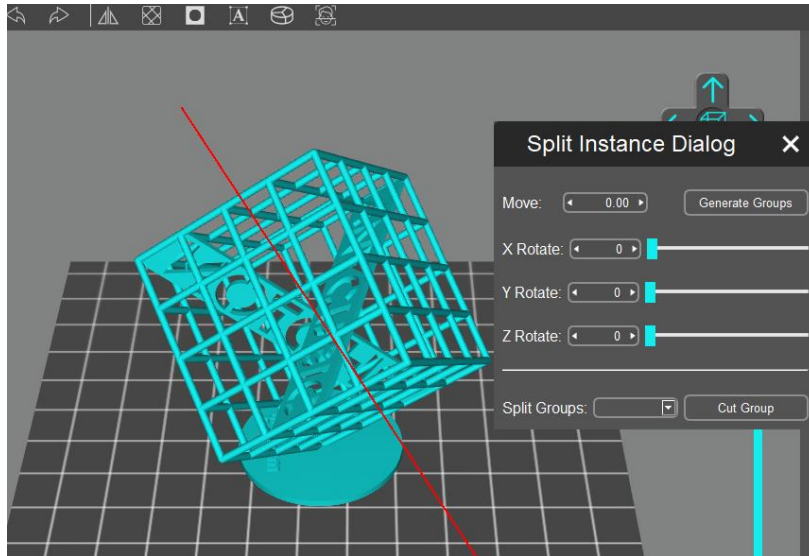
Switch to the delete mode. Click the text and then click "Delete Selected" to delete it. Or click "Delete All" to delete all texts.



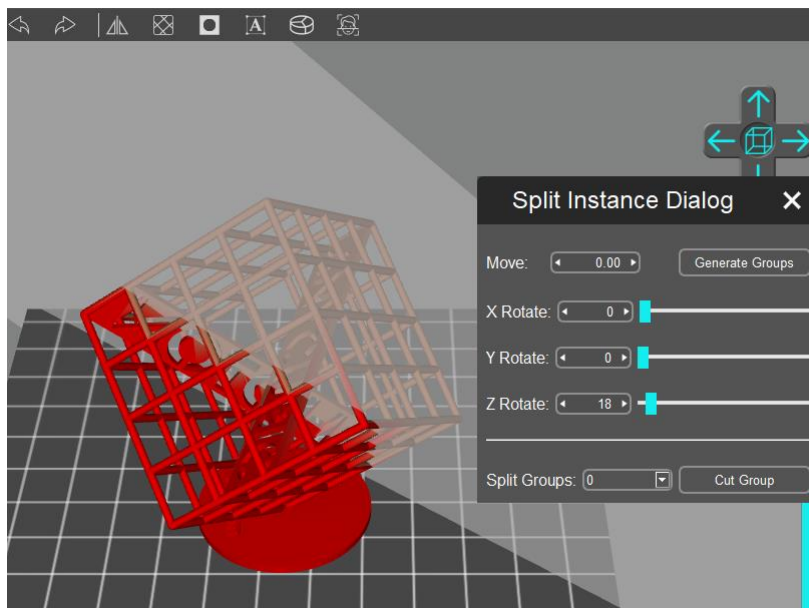
Introduction to Functions

(6) Split Model

You can split the model into several parts and then cut off the unwanted parts. Select split icon and make a cut across the model.




Drag the slider to rotate the cutting surface along the XYZ axis. Then click "Generate Groups" after adjusting the cutting surface. Select the unwanted group from the "Split Groups". The selected group will become red. Click "Cut Group" to finish the splitting.



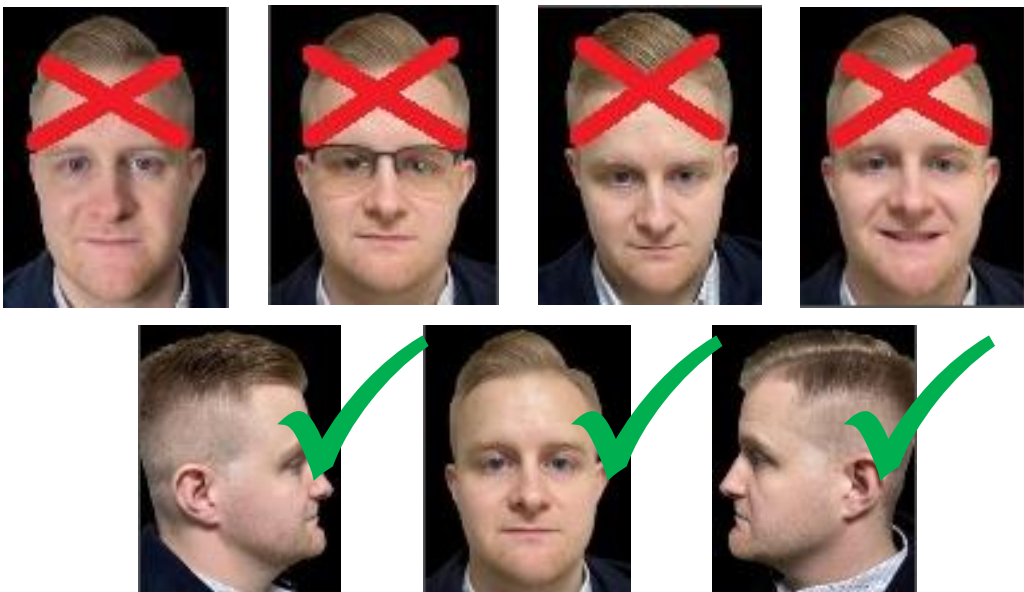
Introduction to Functions

(7) 3D Face Reconstruction

3D face reconstruction is the function reconstructing a face from the front and lateral view of the face into a 3D form.

Click “  ” and upload appropriate photos according to the photo requirements shown below.

- Environment: The light should be even and balanced to avoid shadows on the face. The facial contour should be clearly visible.
- Background: The background of the photo should be single color. The dark background is better. (black>blue>red>white)
- Dimensions: Minimum acceptable dimensions are 84 pixels (width) ×112 pixels (height)
- Facial Requirements: Please present the front and side views with the entire head and face clearly visible. The facial expression should look natural with eyes open and mouth closed. Do not let eyeglasses, hat or other object obscure the facial features.
- Note: The side views should show the point between the eyebrows.

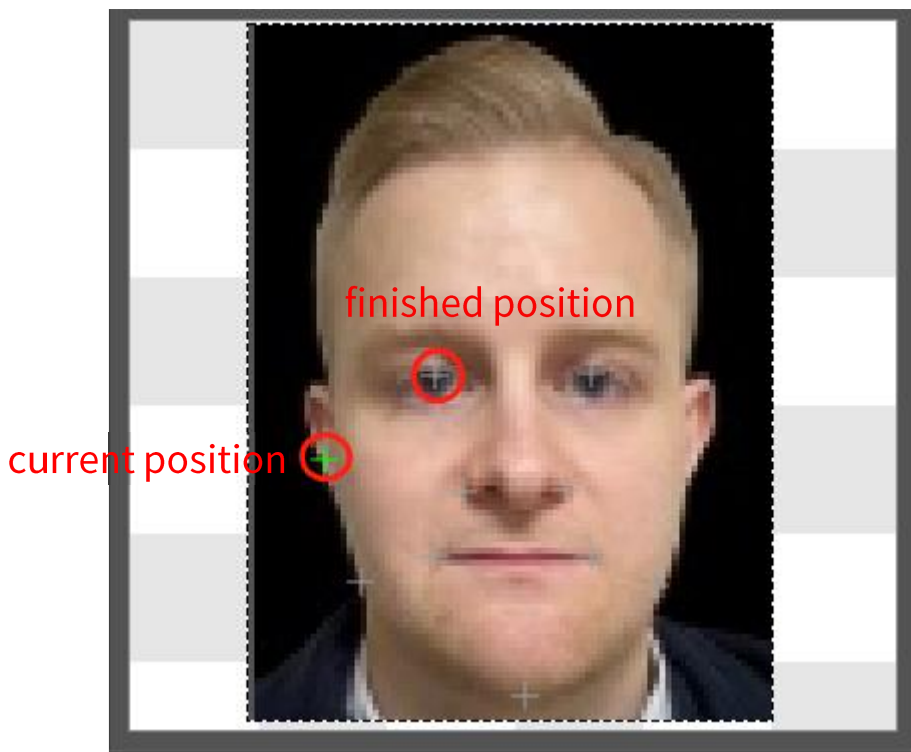


Introduction to Functions

After uploading photos, click “Next” and finish the face localization and detection. Click the corresponding points in accordance with the illustration and prompt.



The finished position is grey and the current position is green. If you make a mistake, click the green cross to cancel.



Click the corresponding button to rotate the uploaded photo. Click the scaling buttons or roll the scroll wheel to scale the photos. When you finish the localization and detection, click “Next” to reconstruct the 3D model of the face.

Support Settings

5. Support Settings

When the model has obvious suspended parts or overhang, it needs to add support to minimize the printing failure.

Click on the model and then click support tab to edit the support for the model.

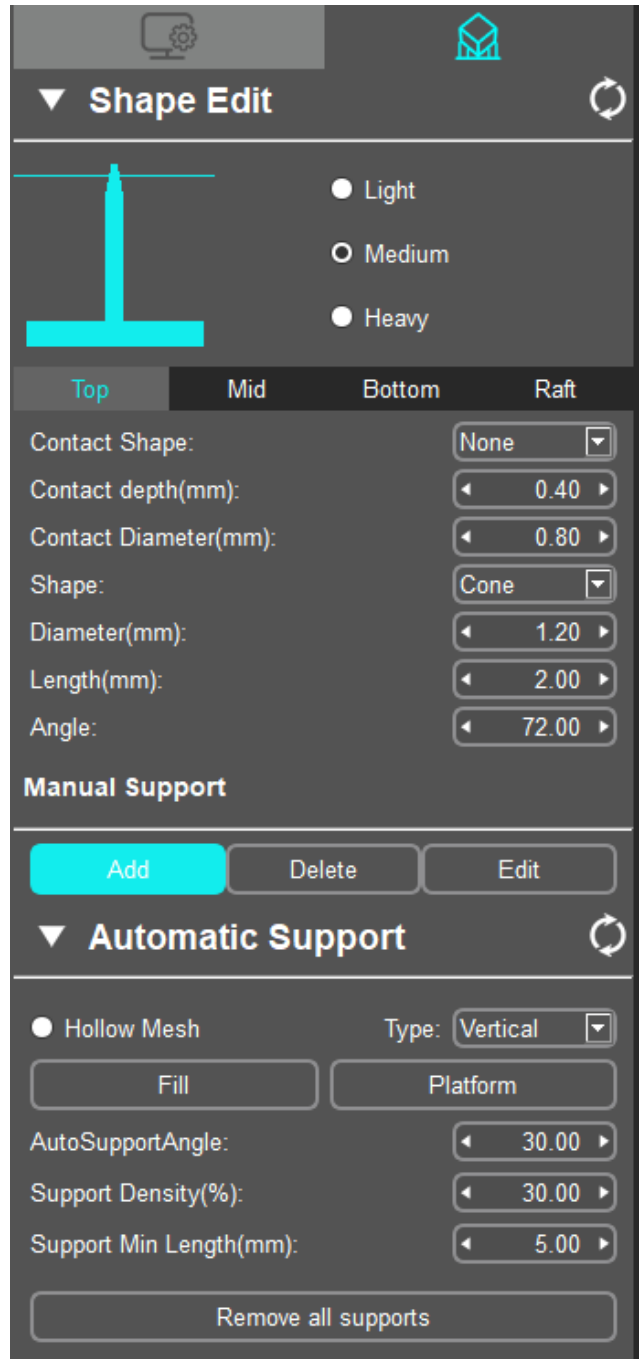
Before adding support, you can edit the shape of the support.

There are three types of support, Light, Medium and Heavy.

Light: Contact area between the support and model is small, and the support is easy to remove;

Heavy: Support contact with the model area is large and solid.

It is recommended to try the “Medium” first, and use the default settings.



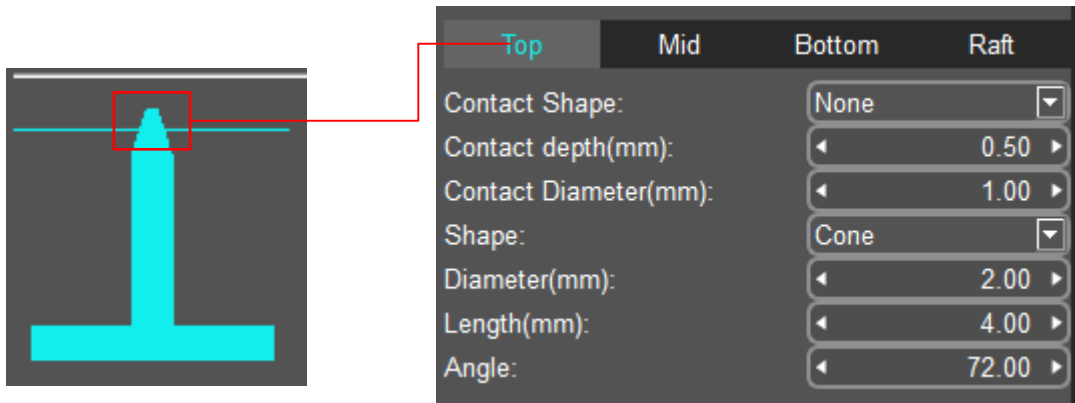
And you can always modify the support settings to fit your requirement.

Support Settings

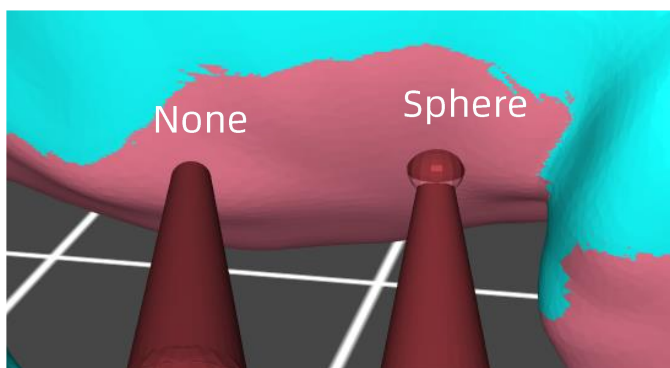
(1) Shape Editing

The support is divided into three parts, namely "top", "middle" and "bottom".

① Top



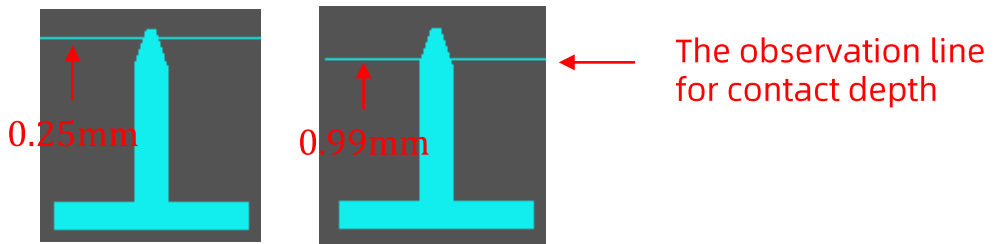
Contact Shape: Select the "Sphere" as the contact point between the top and the model can increase the contact area between the support and the model. The contact diameter is valid when the contact shape is "Sphere".



The author of the model: ZenMaster_Maker

Contact depth: The contact depth between the support tip and the model. An appropriate connection length can ensure that the support top is rough enough and easier to remove.

Support Settings



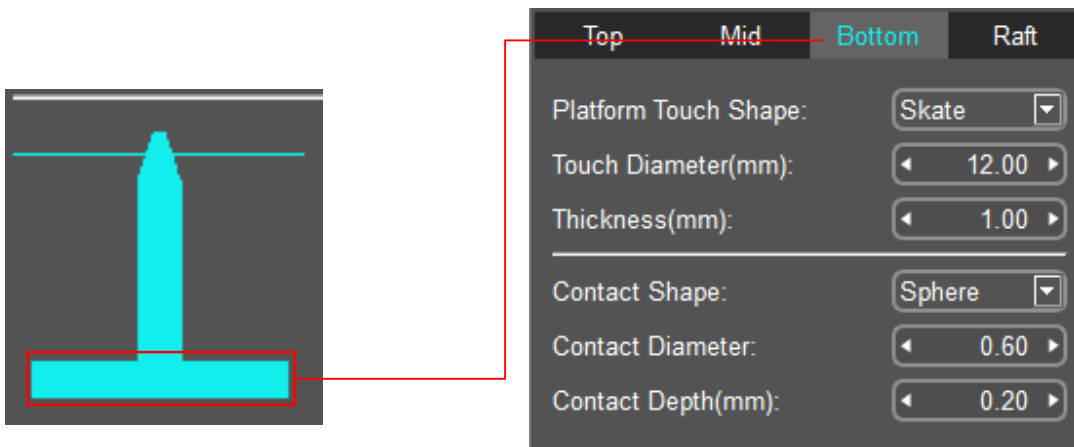
② Mid



Shape: There are three options for the mid shape, "Cube", "Cylinder" and "Prism".

Diameter: The mid diameter must be larger than top diameter.

③ Bottom



Platform Touch Shape: It is recommended to choose skate as bottom, which is easier to be removed from platform.

Contact Shape/Contact Diameter/Contact Depth : The bottom shape, diameter and depth of the supports added inside the evacuated model.

Support Settings

④ Raft

There also can add raft to supports. Raft will increase the adhesion between model and printing platform to reduce the risk of warping and print failure.

Select the shape of the raft as "Cell" and click "Fill" or "Platform" to add the raft and support.



Note: Before adding the raft, you need to lift the model up by a certain height in the Z-axis direction.

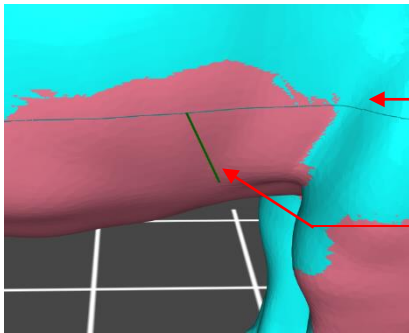
Support Settings

(2) Support Adding

You can add the support to the model manually or automatically after setting up the shape of the support.

① Manual Support

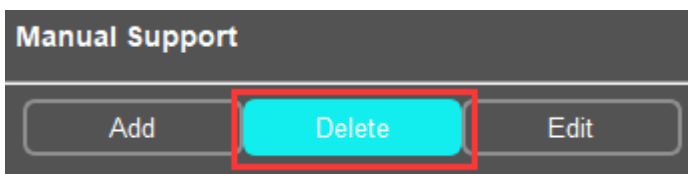
Add: In add mode, click the point on model which is needed support to add support.



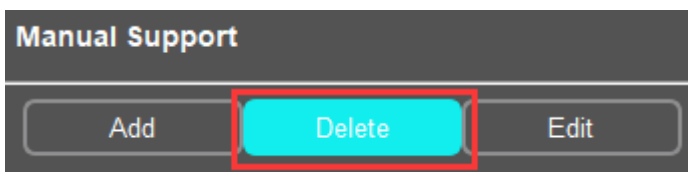
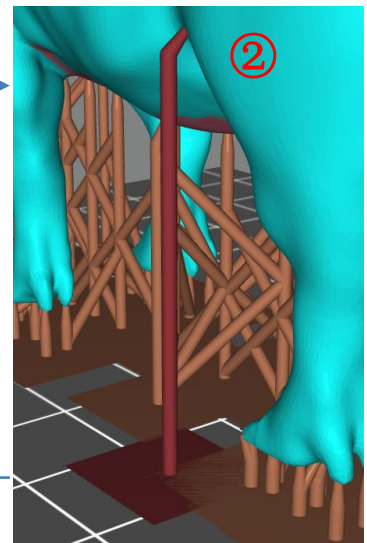
contour line, it can be used as reference line when you are adding support

When the mouse moves on the model, the green short line can be clicked to add support; the red short line means the place cannot be added support on.

Delete: Click "Delete" to switch to delete mode. Click the support you want to delete and click "Delete" again to remove it.



①



③

Support Settings

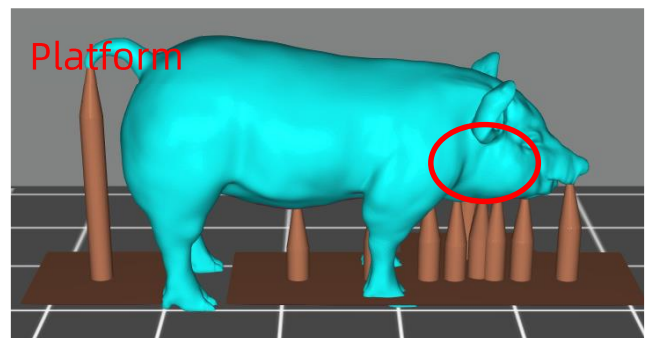
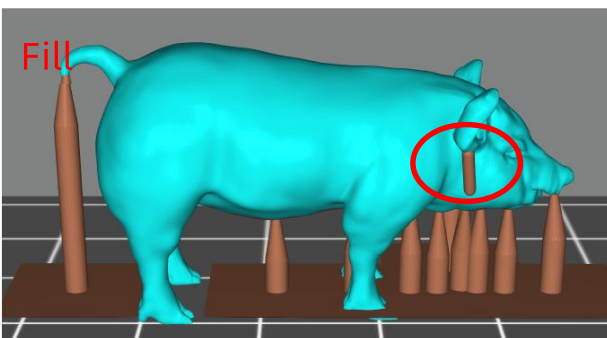
Edit: The support can be edited after clicking the “Edit” button. Click the support, it will become red. Its shape can be changed through editing the top, mid and bottom parameters.

② Automatic Support

Set "Auto Support Angle", "Support Min Length" and "Support Density", and choose support shape as "Vertical" or "Tree". When you choose "Tree" type, the supports can be combined and interlocked. It simplifies the supports and saves material. When “Hollow Mesh” is selected, the hollowed part will be added supports.

When all setting is finished, click "Fill" or “Platform” to automatically add supports for the model.

- "Fill": The support can be added between the platform and the model, and between the model and the model.
- "Platform": The support only can be added between the platform and the model.



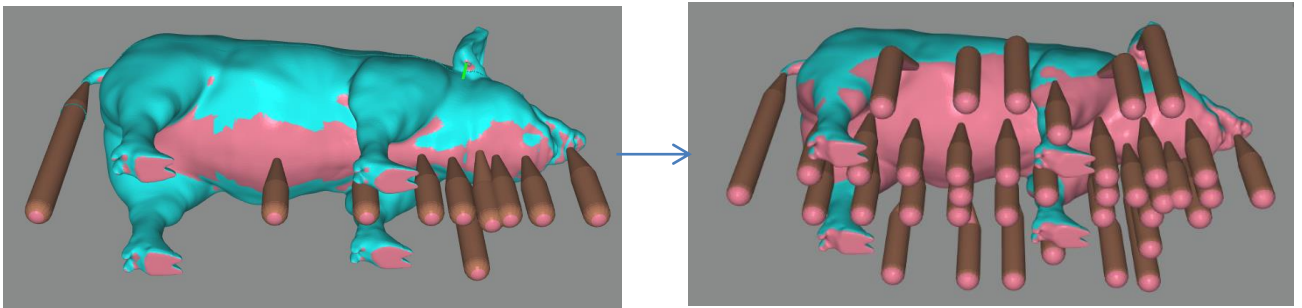
Note: The automatic support will override all the previously set supports.

Support Settings

③ Automatic support adding skills (to improve print success rate)

Tip 1: Properly increasing the support angle and density can optimize the support results and deliver better print quality.

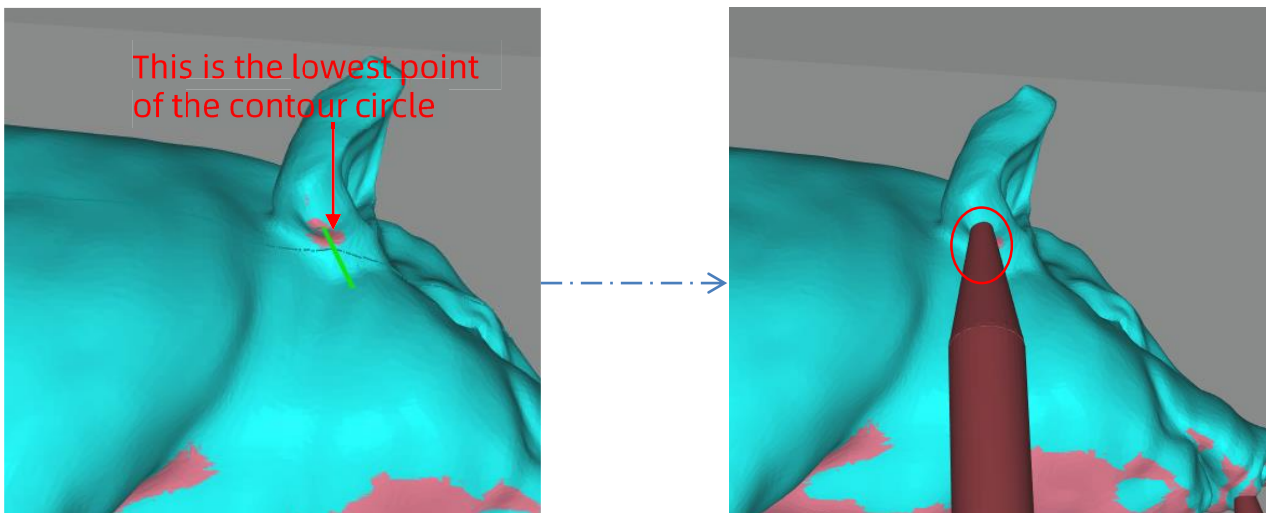
When browsing on the model, by observing the contour circle, it can be found that the model still has some weak points that have not been added supports properly.



If you increase the automatic support angle and support density, more supports can be added to some of the weak points.

Tip 2: Add manual support after automatic support.

Add support to the local lowest point by checking the contour circle.

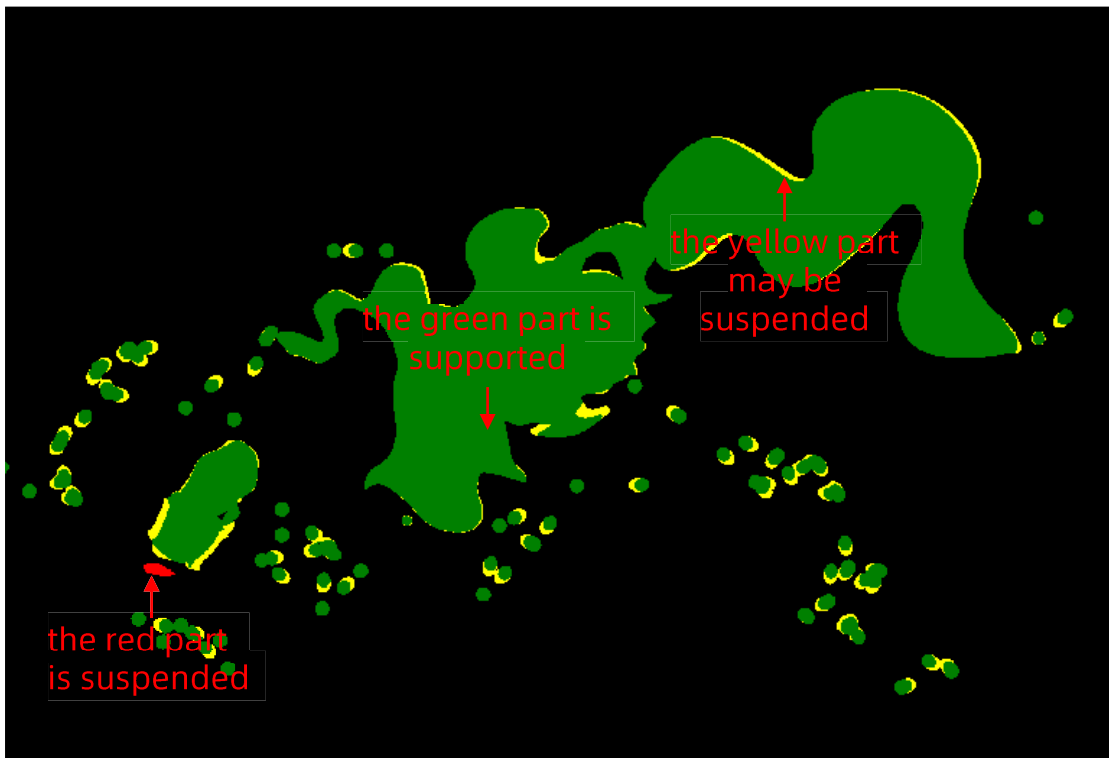


Support Settings

Tips 3 : Check Land.

It checks whether models remain suspending parts. Click “View”→“2D View” and “View”→“Check Land”, then drag slider to check the image of every layer.

The green part means there is support below; the yellow part is connected to other parts but may be suspended and may need supports; the red part is completely suspended and must be added supports.



You can also check land in sliced file preview interface “Advance”→“Check Land”.

Support Settings

(3) Export Support Information (when you need)

After adding supports, click “File” → “Save Scene File” to save the current scene as a file. The file can reproduce the support information and be available for different types of machine.

You can click “File”→ “Open Scene File” to reproduce the scene and edit the supports again, or sent the file to other computers to edit.

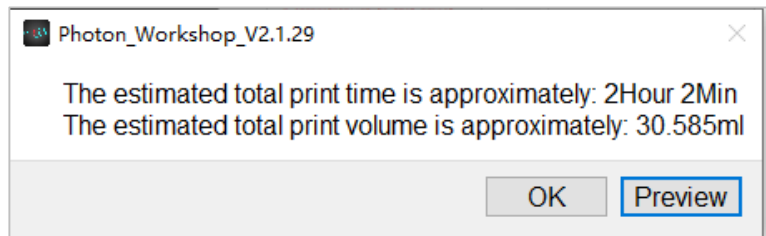
Export Sliced File

6. Export Sliced File

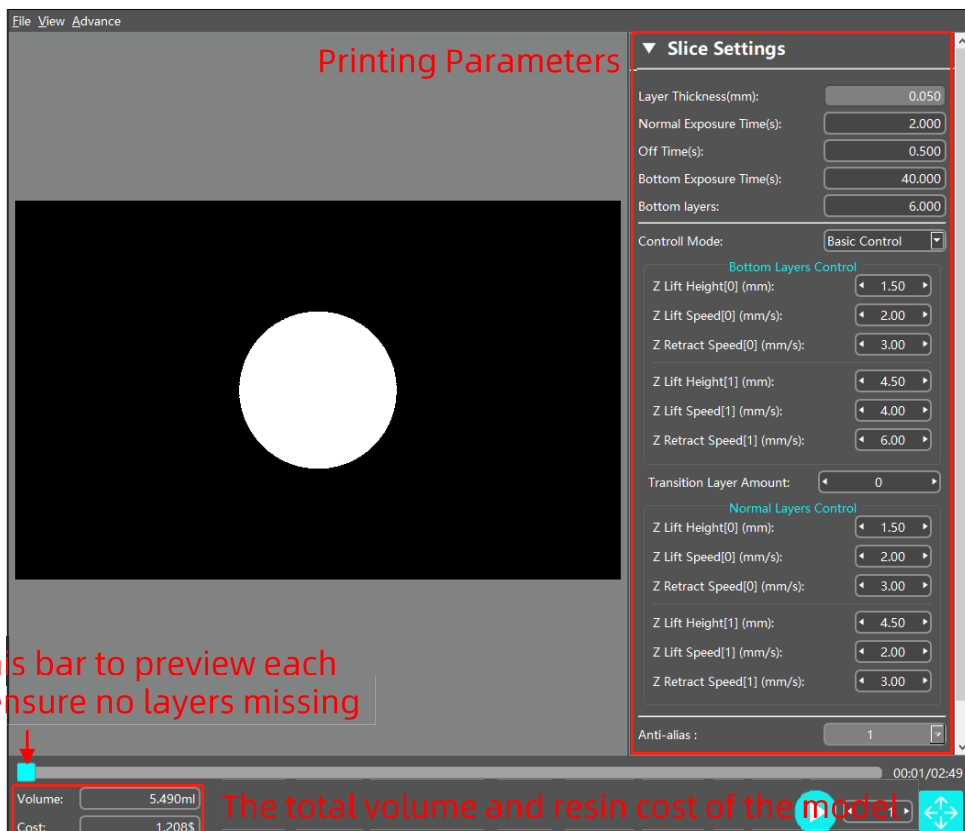
After confirming the slice settings, click the "Slice" icon at the lower left corner or click "Slice" at the lower right corner to save sliced file. Anycubic 3D printer can only read its corresponding sliced file formats, please choose the machine type you use at Machine Settings to ensure higher printing success rate. Wait for a while to generate sliced file and click OK to finish. Or click "Preview" to check each layer and the printing parameters.



"Slice"
icon



In the slice file view interface, you can preview related slice settings, layer parameter and other information.



Slicing

If necessary, adjust the slice settings. Or check to enable layer settings and set the parameters. Upon finished, click “File” → “Save File” on the upper left corner to save it as a new sliced file.



Note: After changing the individual layer settings, the exposure parameters of the new file cannot be modified again via the printer touch screen during printing. Even if it has been modified, the change would only be valid for the current layer.