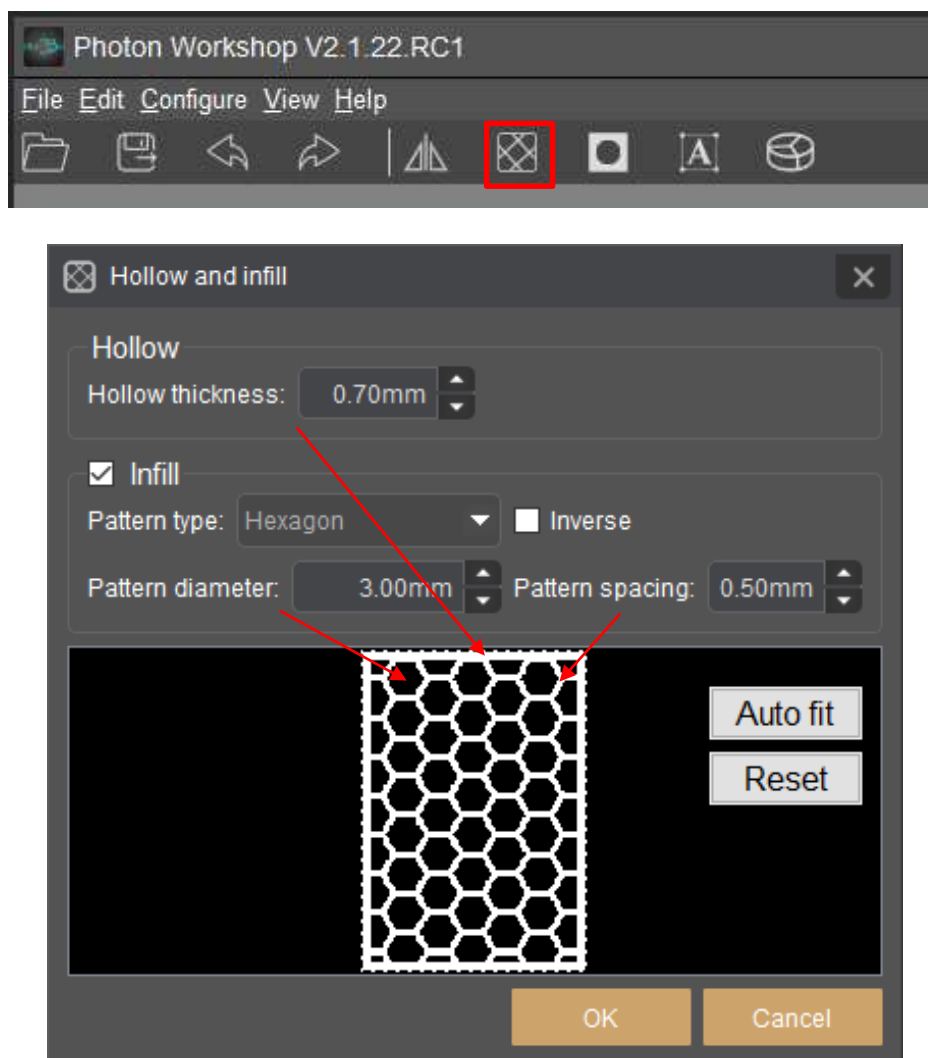


Introduction to Slicing Software

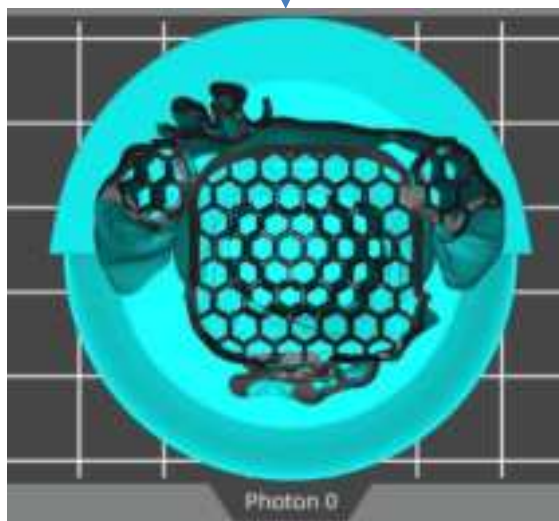
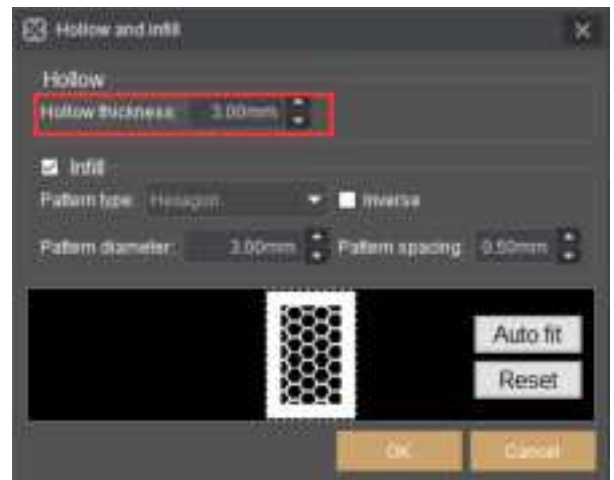
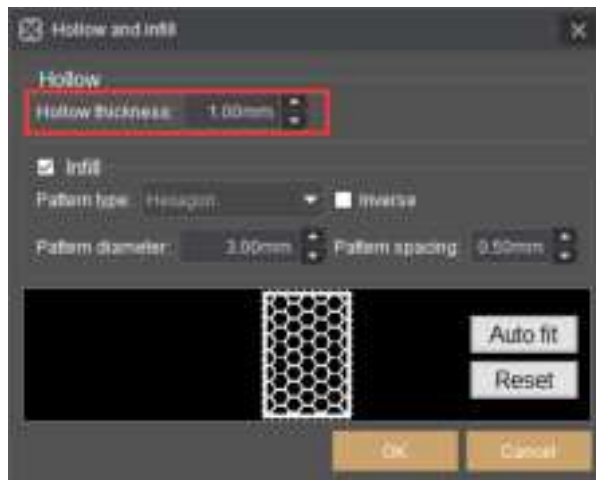
(4) Hollow And Fill

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

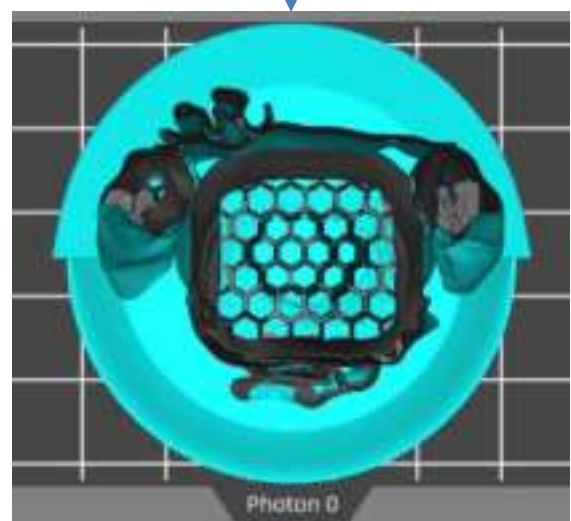


Please check and choose the most suitable parameters for hollowing and infilling to fit your requirement.

Introduction to Slicing Software



Hollow thickness: 1mm



Hollow thickness: 3mm

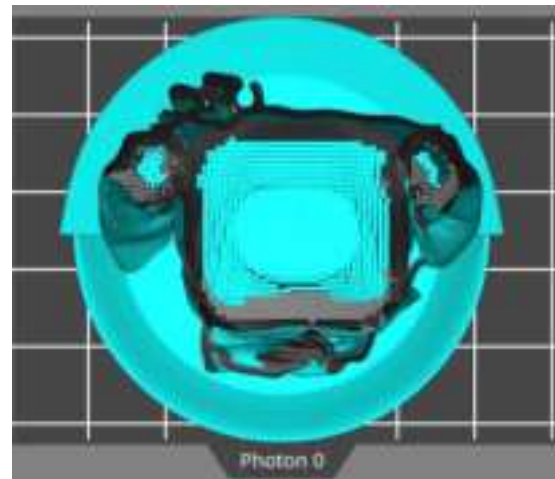
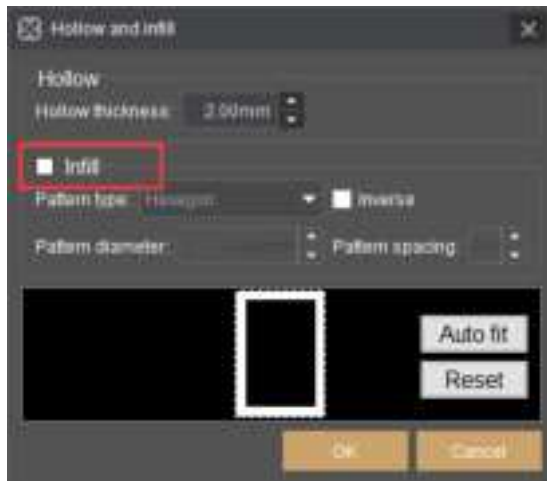


The model name: MIA

The author of the model: Fabio Nishikata

You may drag the slider to see the internal structure of the model after hollowing.

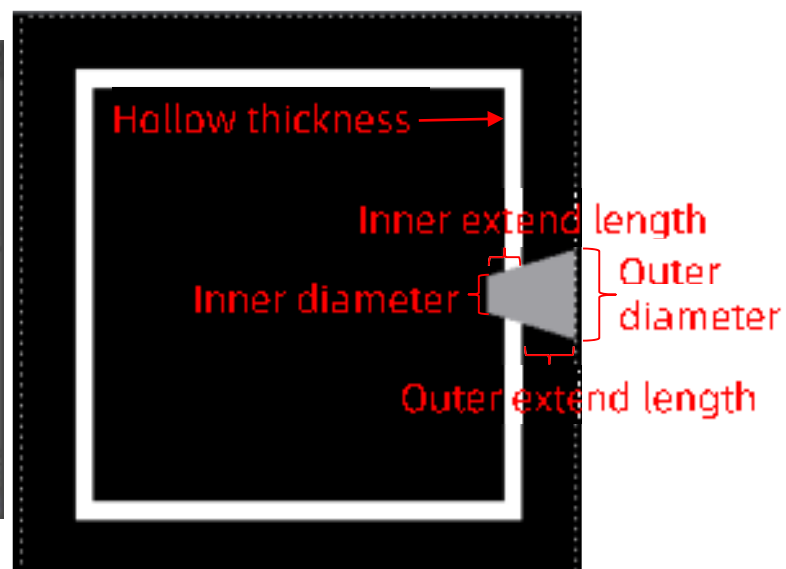
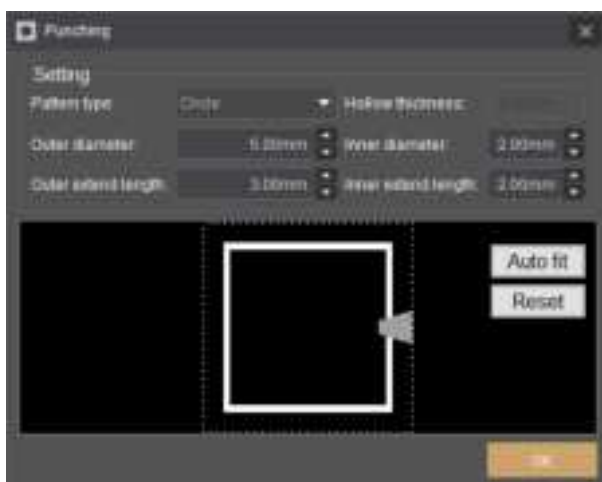
Introduction to Slicing Software



Hollow thickness: 2mm, no infill

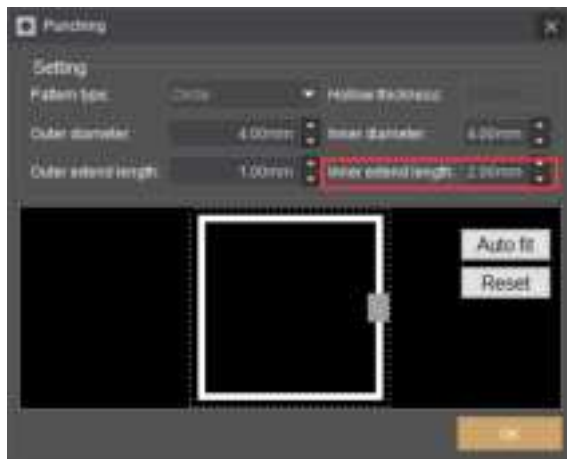
(5) Punching

Although the model is hollowed, there will still be part of the resin remaining in the model after printing. By punching the model, the resin in the model can flow out, which reduces the weight of the model and the resin consumption.

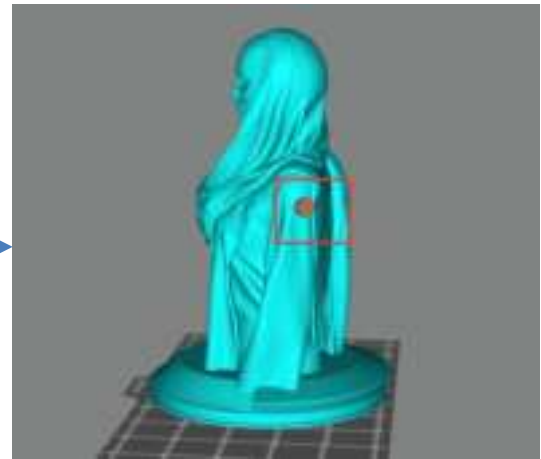


The "Inner extend length" must be greater than the "Hollow thickness", so that the model can be pierced when punching and the resin can flow out of the model.

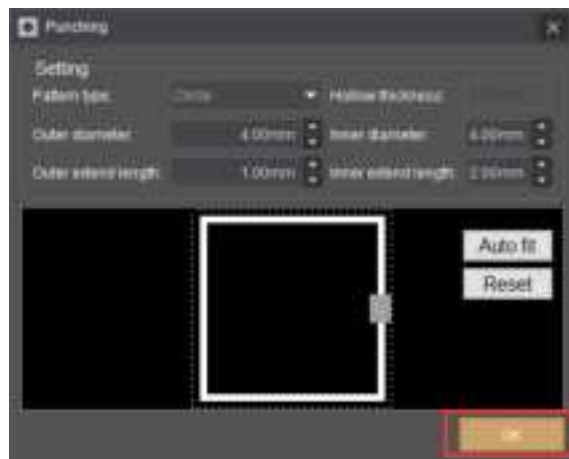
Introduction to Slicing Software



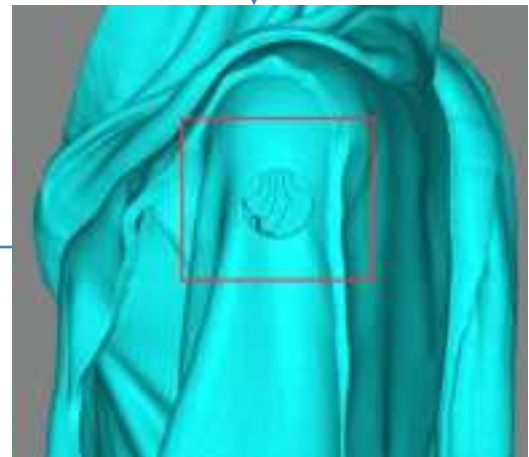
Set the parameter of the hole



Left click on the model



Click "OK" to finish



The model name: MIA

The author of the model: Fabio Nishikata

(6) Text Paste

You can paste text on the model with this feature.

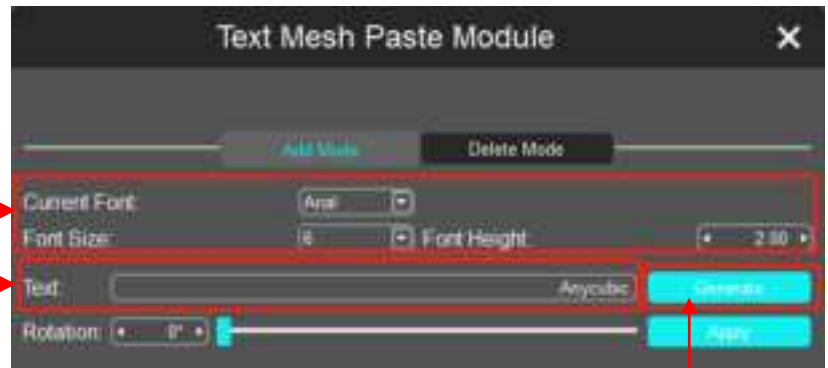


Slicing Software

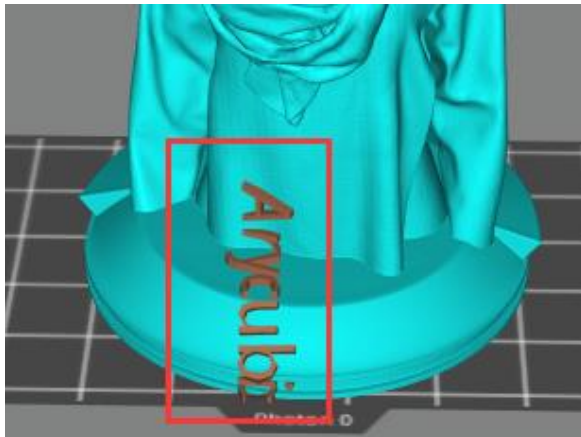
- Add mode

① Set the text style.

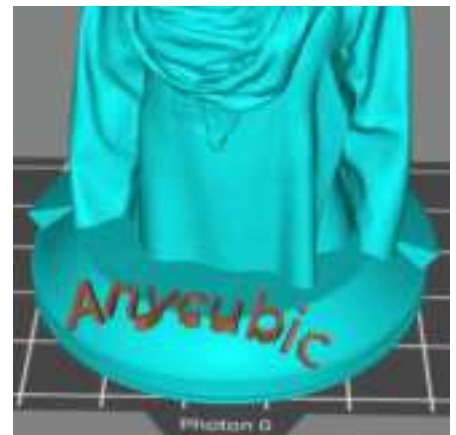
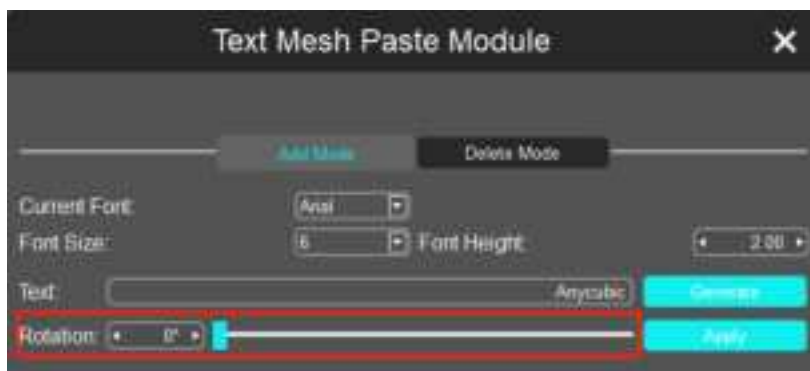
② Input the text.



③ Click to generate mesh.



④ Left click on the model.

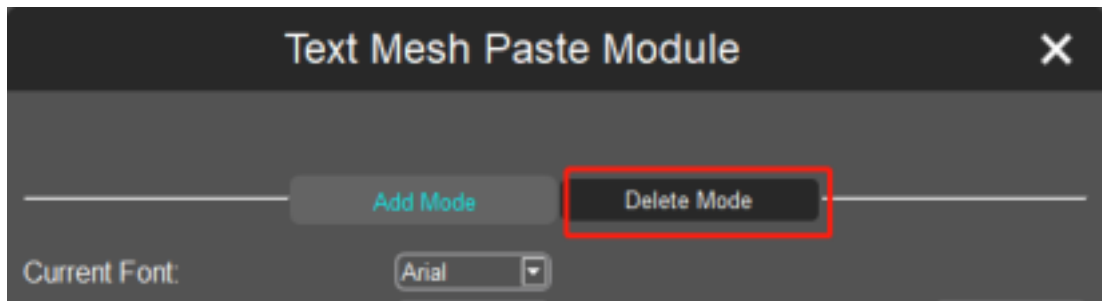


⑤ Drag the slider or input the number to rotate the text.

Slicing Software

- **Delete mode**

① Switch to the delete mode.



② Click the text and then click "Delete selected" to delete it.



③ Click "Delete All" to delete all texts added before.



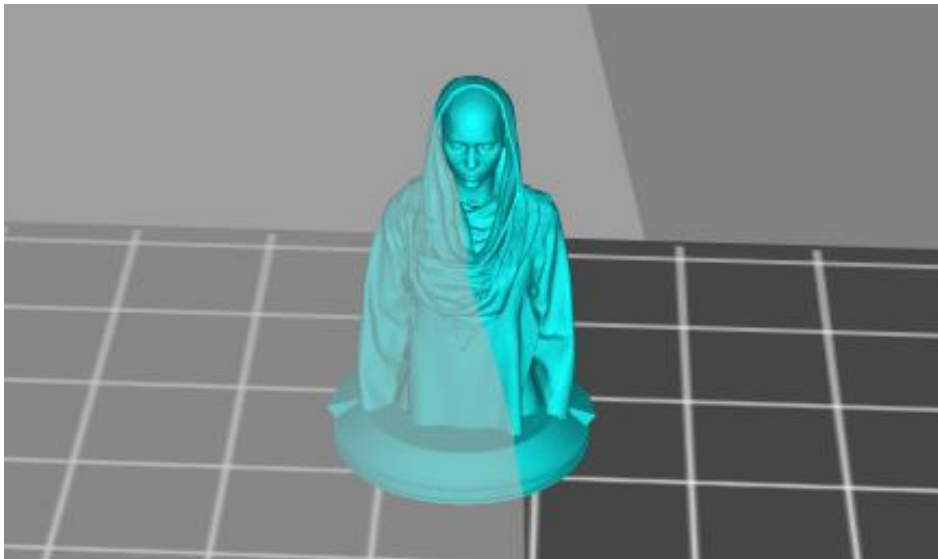
(7) Split model

You can split the model into several parts and then cut off the unwanted parts.



Introduction to Slicing Software

① Click the split icon, as shown in the red square above. Then make a cut across the model.

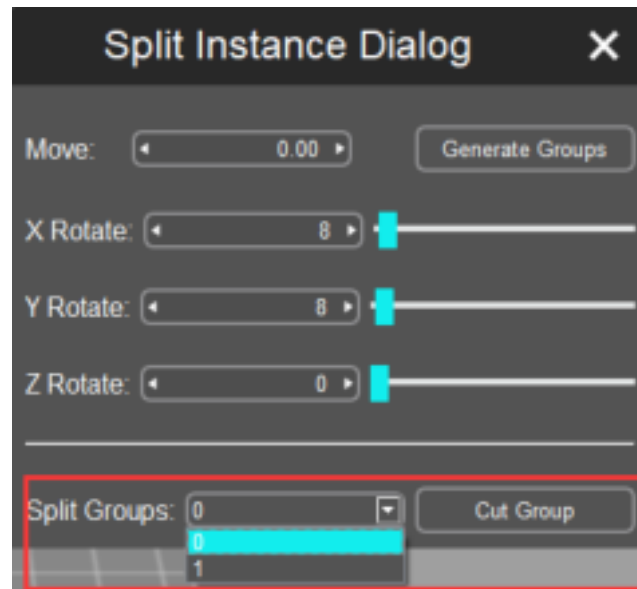


② Drag the slider to rotate the cutting surface along the XYZ axis. Then click “Generate Groups” after adjusting the cutting surface.

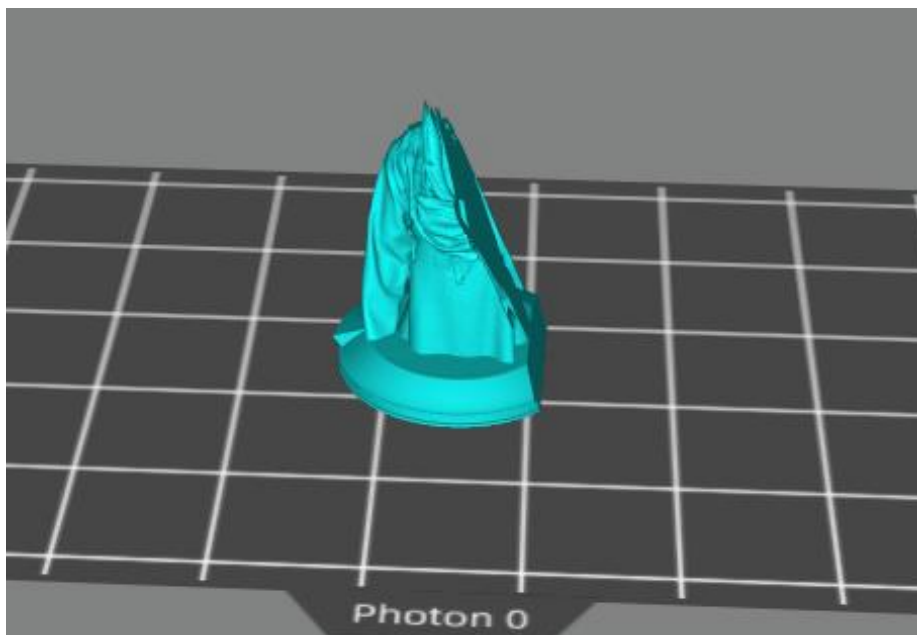


Introduction to Slicing Software

③ Select the unwanted group from the "Split Groups". The selected group will be shown in red in the model. Click "Cut Group" to remove it.



④ The effect picture after splitting the model.



Introduction to Slicing Software

(8) 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.

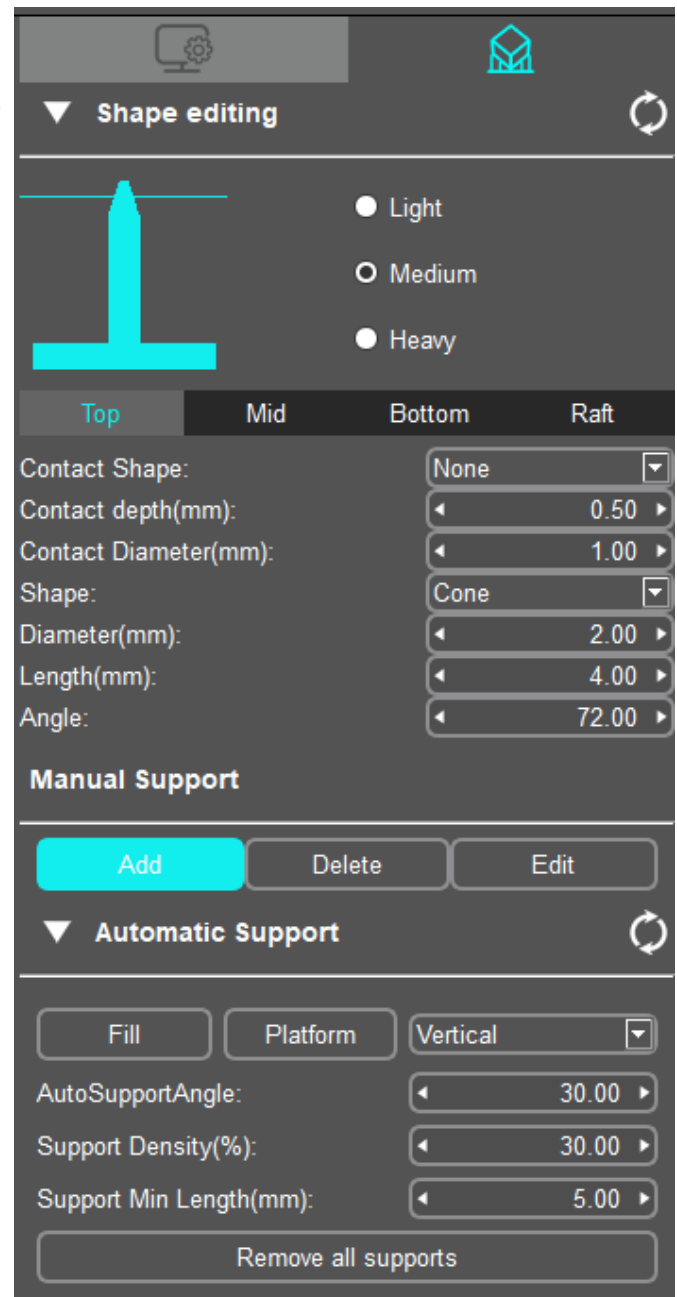
Before adding support, you can edit the shape of the support.

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 using the default settings.



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

Introduction to Slicing Software

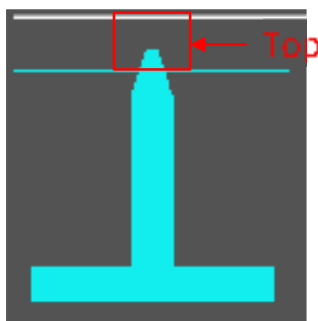
Step 1: Shape editing

Click on one of these types, such as Medium. As shown below, the support is divided into three parts, namely "top", "middle" and "bottom". The settings of those three parts are described in detail below.



Click to refresh to preset parameters

① **Top:** Set the parameters for the top of the support.



Top	Mid	Bottom	Raft
Contact Shape:	None		
Contact depth(mm):	0.50		
Contact Diameter(mm):	1.00		
Shape:	Cone		
Diameter(mm):	2.00		
Length(mm):	4.00		
Angle:	72.00		

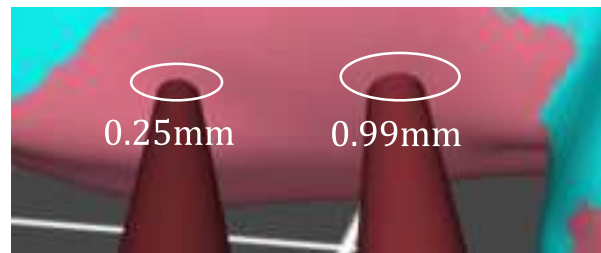
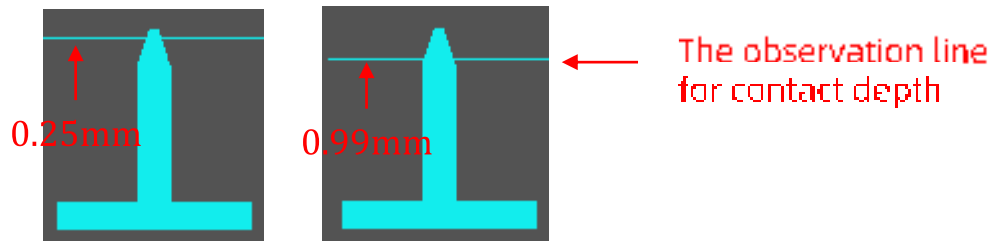
Contact Shape: Select the "Sphere" as the contact point between the top and the model can increase the contact area between the support and the model.



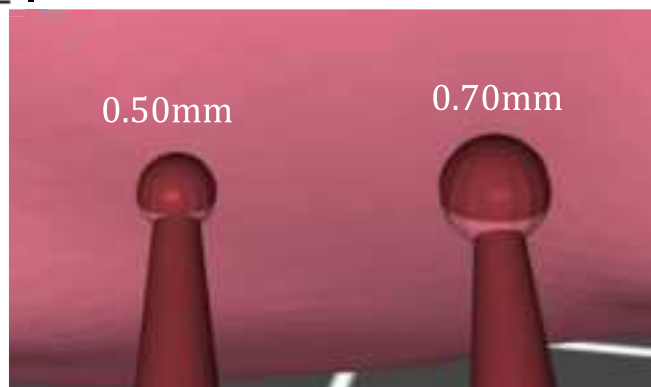
The author of the model: ZenMaster Maker

Introduction to Slicing Software

Contact depth: The contact depth between the support top and the model.



Contact Diameter: The contact diameter is valid when the contact shape is "Sphere".

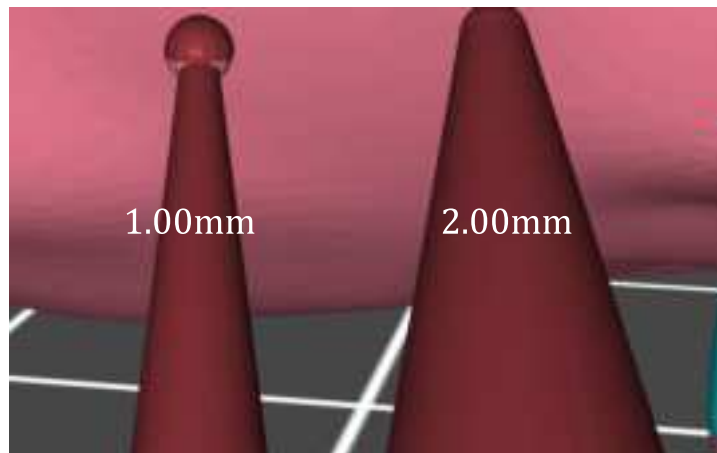


Shape: There are two options for the top shape, "Cone" and "Pyramid".

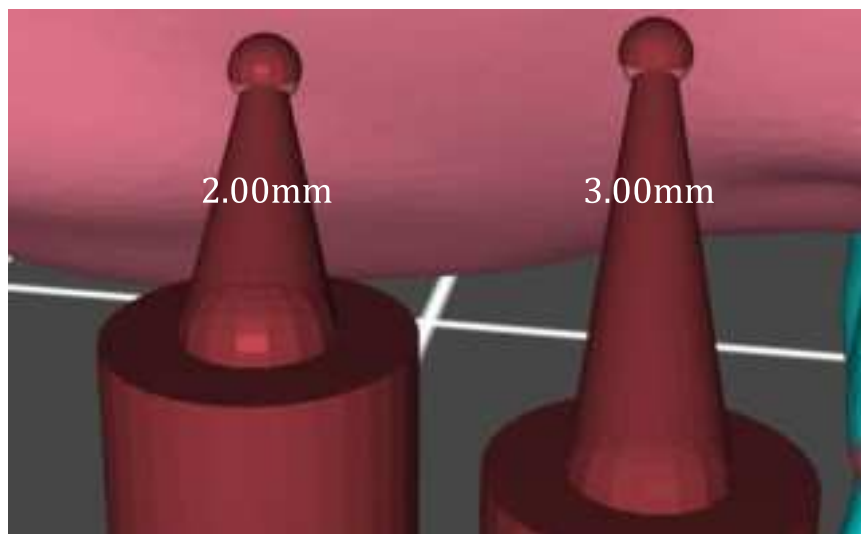


Introduction to Slicing Software

Diameter: You can input number to change the top diameter.

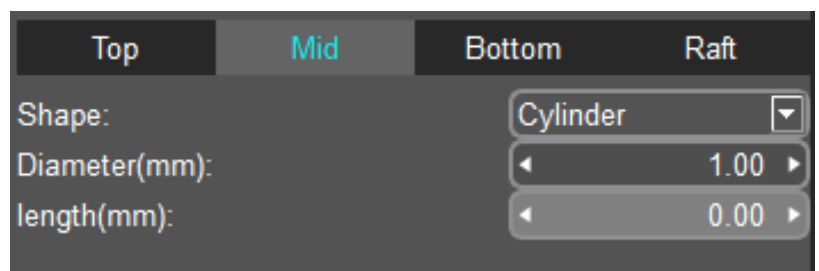
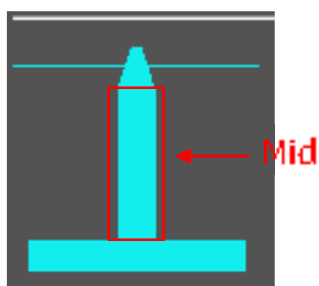


Length: You can input number to change the top length.



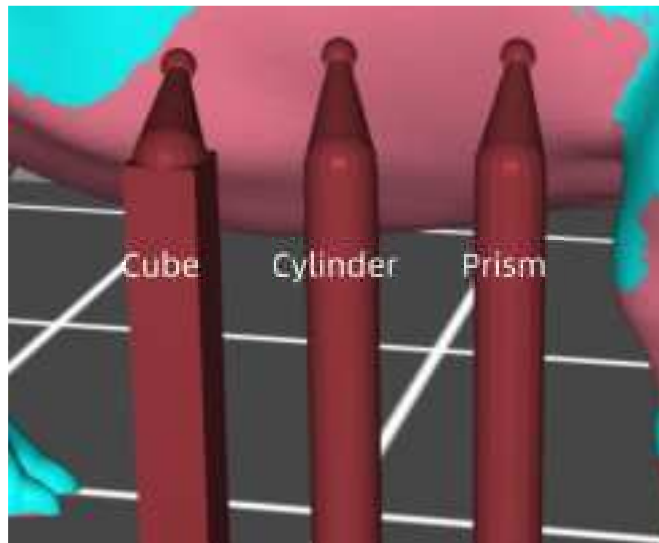
Angle: Use the default parameter.

② **Mid:** Set the parameters of the mid of the support.

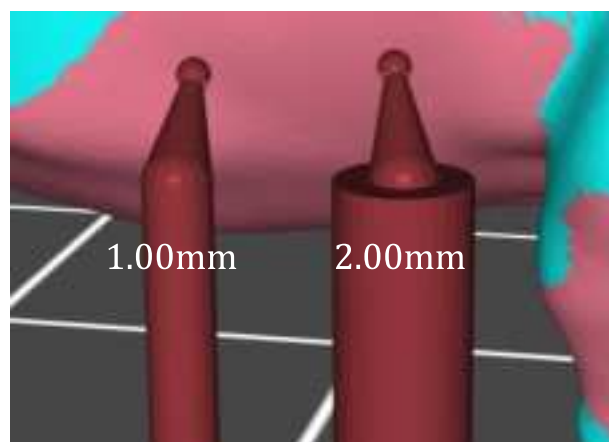


Introduction to Slicing Software

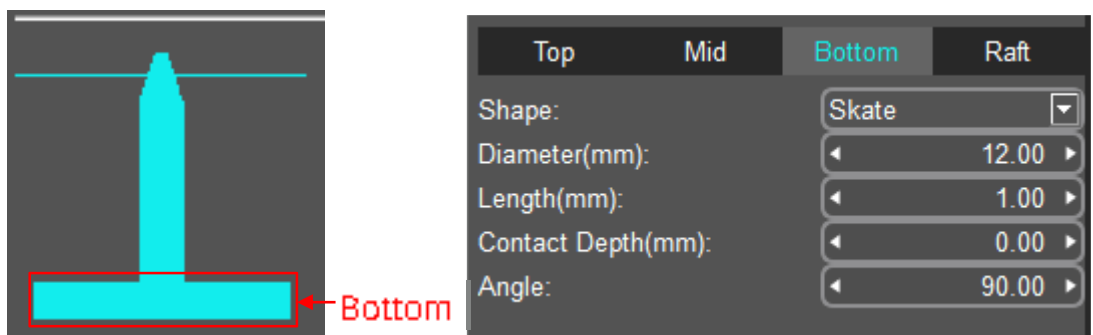
Shape: There are three options for the mid shape, "Cube", "Cylinder" and "Prism".



Diameter: You can input the number to change the mid diameter.

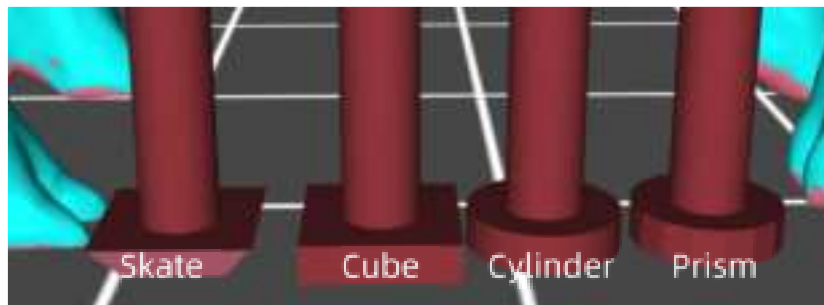


③ **Bottom:** Set the parameters on the bottom of the support.

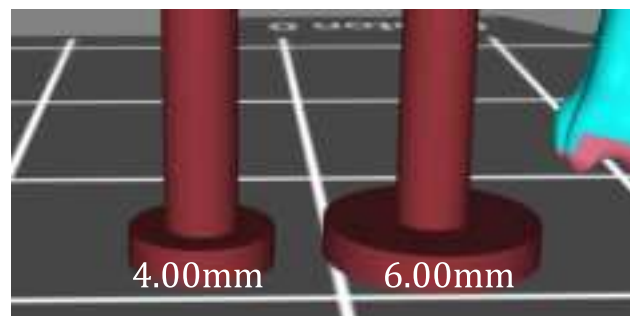


Introduction to Slicing Software

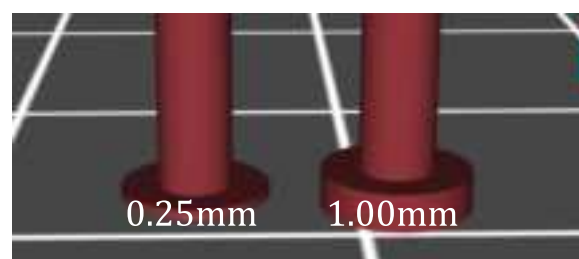
Shape: There are four options for the top shape, "Skate", "Cube", "Cylinder" and "Prism".



Diameter: You can input the number to change the bottom diameter.



Length: You can input the number to change the bottom length.



Contact Depth: The depth of contact between the bottom of the support and the model when the support is added inside the model.

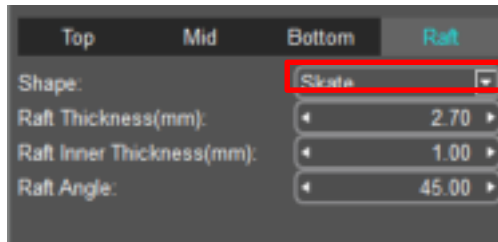
Angle: Use the default parameter.

Introduction to Slicing Software

④ Raft

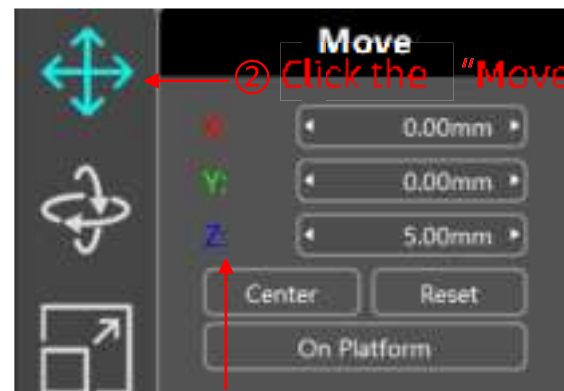
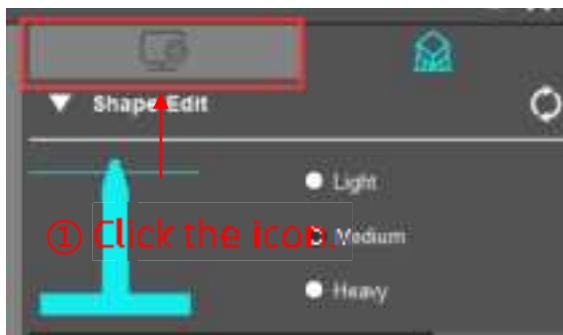
In addition to the three types of support set by software, raft can be added to the model. Adding raft will increase the adhesion between model and build platform, thereby minimizing the print failure or warping risk.

Select the shape of the raft as "Skate" 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.

- Lift the model up by 5mm (suggested) in the z-axis direction:



③ Set the Z to 5mm.

After lifting the model up, click the "  " icon to add support.

Step 2: Support adding

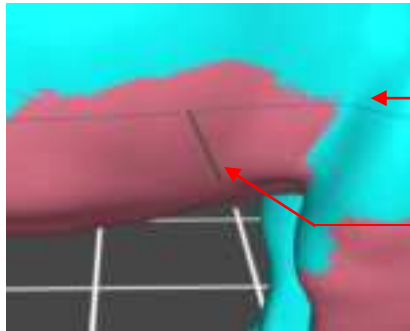
You can add the support to the model manually or automatically after setting up the shape of the support.

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

Introduction to Slicing Software

① Manual Support

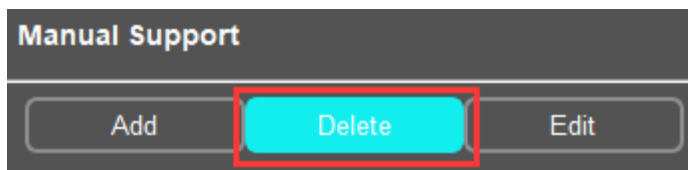
Add: Only after click the "Add" button then can you add the support to the model.



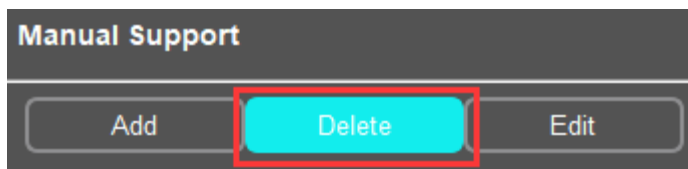
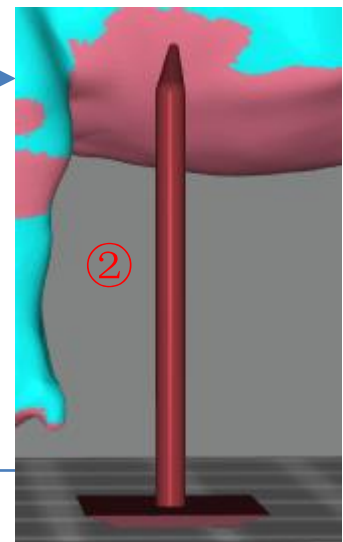
contour line, it can be used as reference line when you are adding support

When the mouse moves on the model, the green short line can be clicked to add support; the red short line means it can not be clicked to add support

Delete: Click the "Delete" button first, and then click the support on the model and click "Delete" button to remove the support.



①



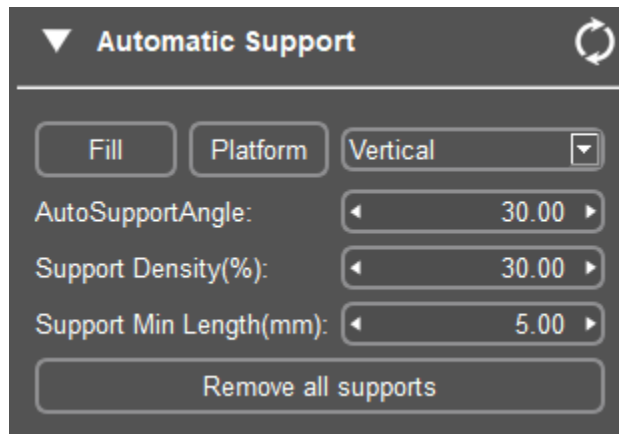
③

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. Besides, left click the model, hold on and move the mouse can change the position of the support.

Introduction to Slicing Software

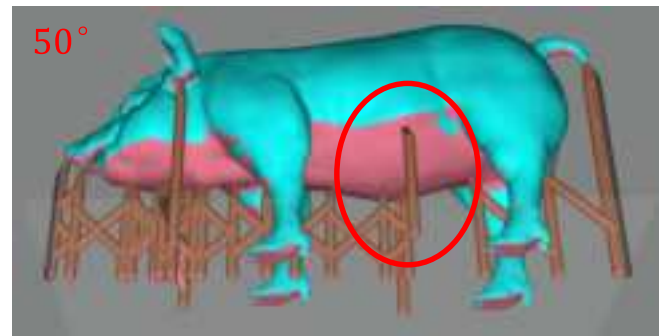
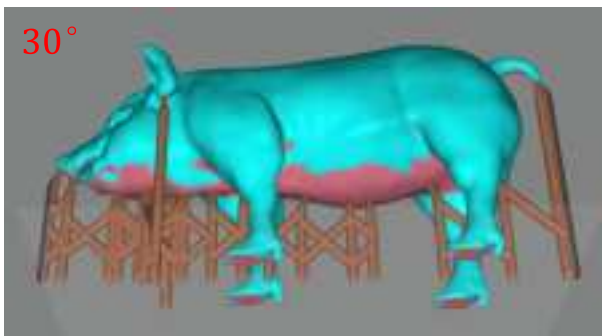
② Automatic Support

After setting "Auto Support Angle", "Support Min Length" and "Support Density", click "Fill" or "Platform" can automatically add supports for the model.



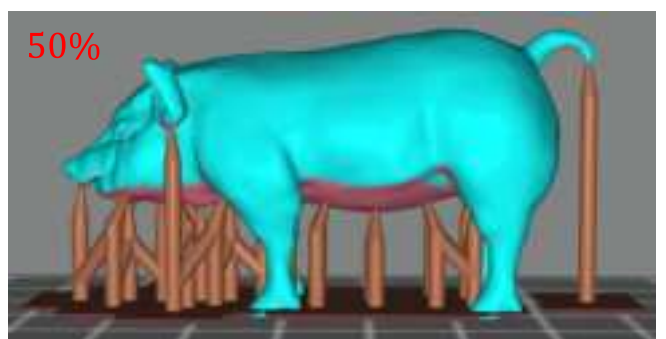
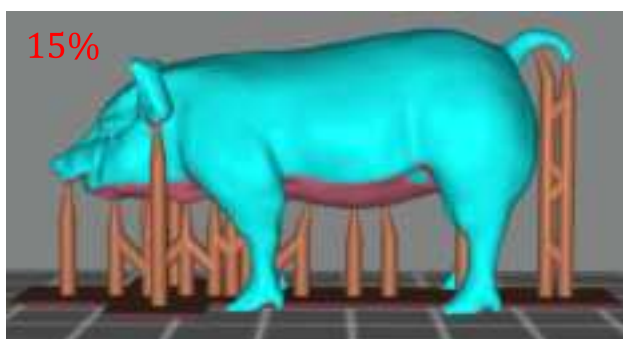
Click to refresh to the preset parameters

Auto Support Angle: The tangent angle between the model (small triangular facets) and the printing platform.



With the same "Support Density", the larger "Auto Support Angle" is, the more supports can be added.

Support Density:

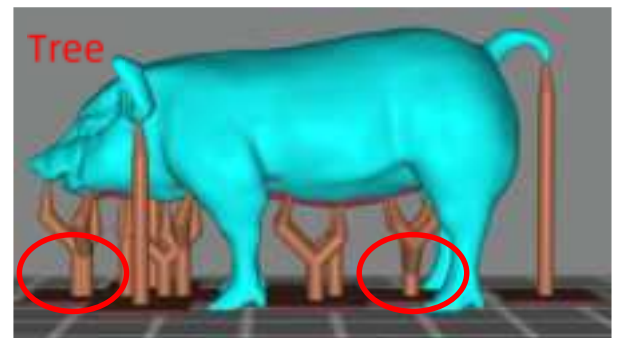
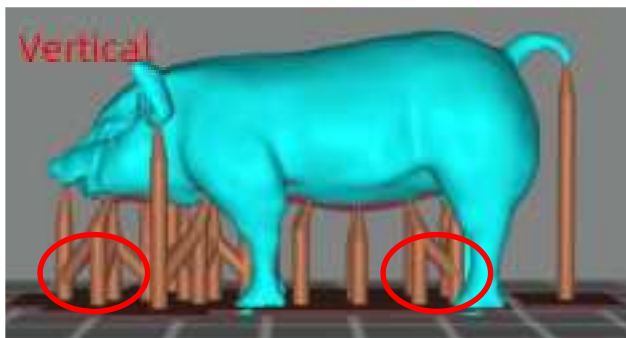
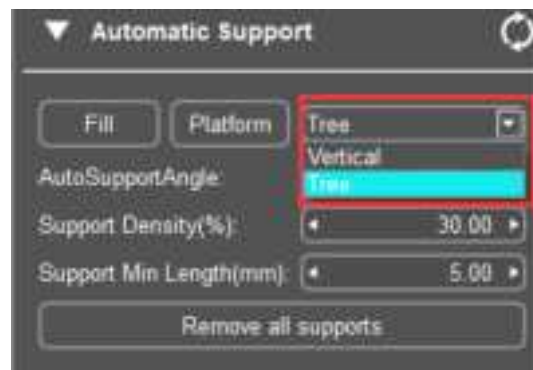


The author of the model: ZenMaster Maker

Introduction to Slicing Software

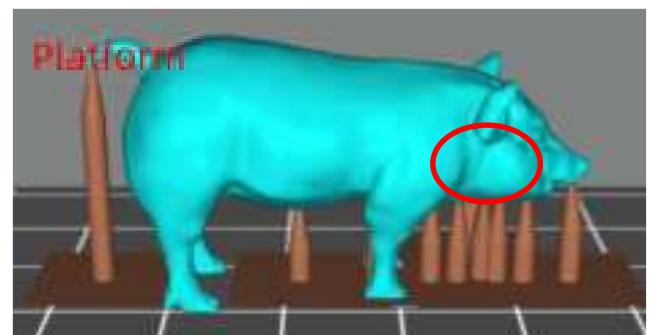
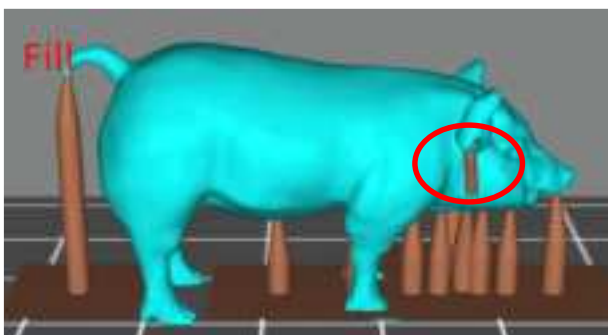
Support **Min Length**: Use the default parameter.

There are two types of support added automatically, "Vertical" and "Tree". Choosing the "Tree" type, the supports can be combined and interlocked. It simplifies the supports and saves material.



After setting each parameter, click the "Fill" or "Platform" button to automatically add support.

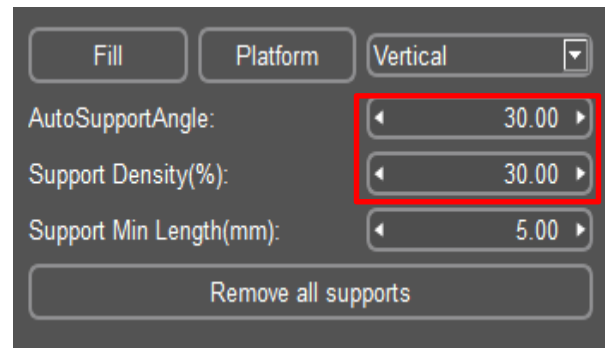
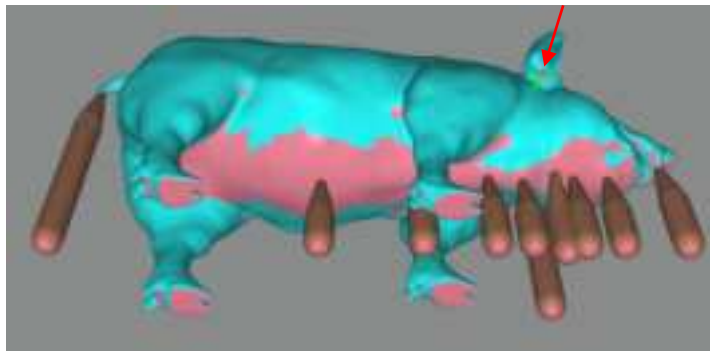
- "Fill": The support can be added between the platform and the model, and between the model and the model.
- "Platform": The support only can be added between the platform and the model.



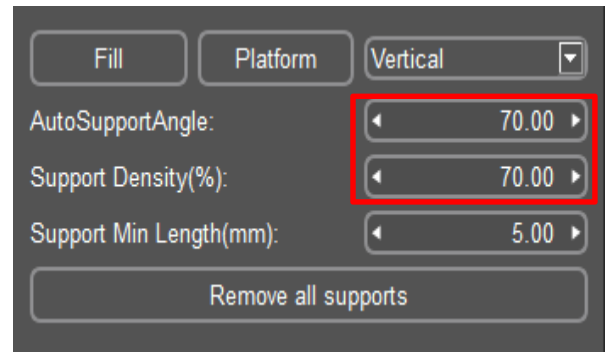
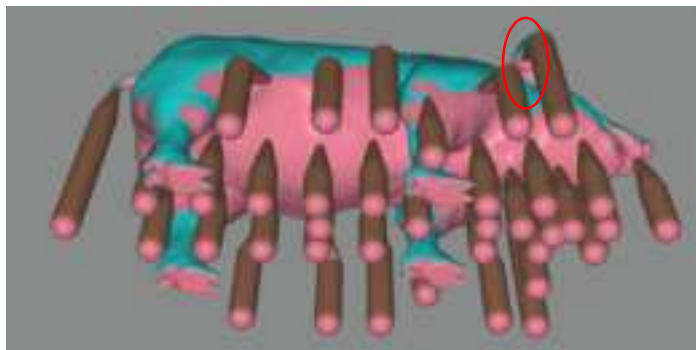
Introduction to Slicing Software

③ Automatic support adding skills (improve print success rate)

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



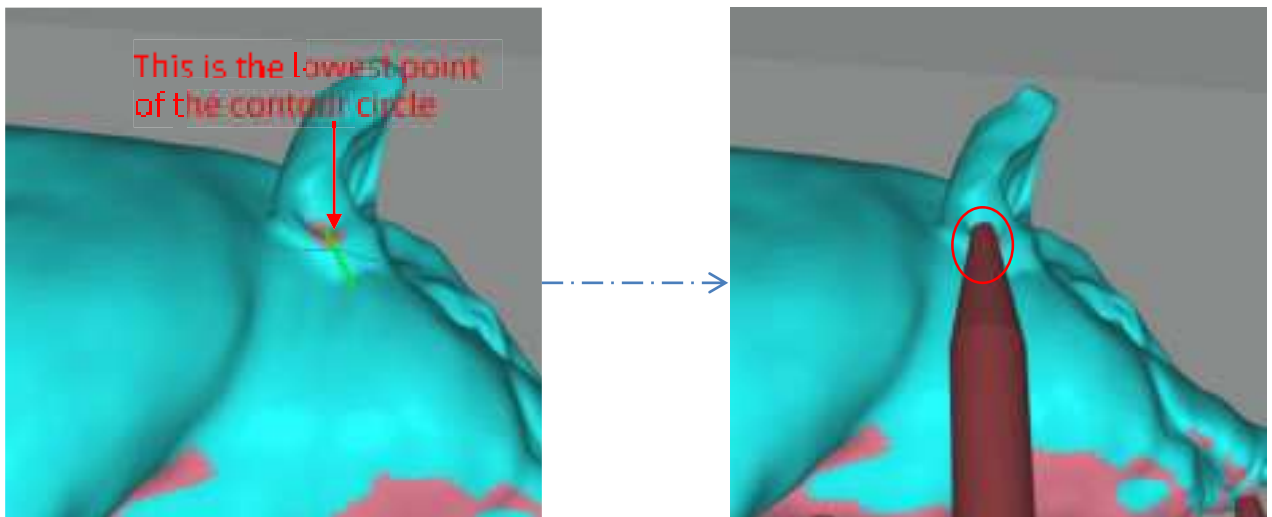
When browsing on the model, by observing the contour circle, it can be found that the model still has some weak points that have not been added supports properly (highlighted by red arrows).



If we increase the automatic support angle and support density (highlighted by red square), we can see from the picture below that more supports have been added to some of the weak points.

Tip 2: Manual support after Auto support (use the contour to find the weak points, add support to the local lowest point by checking the contour circle)

Introduction to Slicing Software



(9) Parameter Settings

① Slice Settings

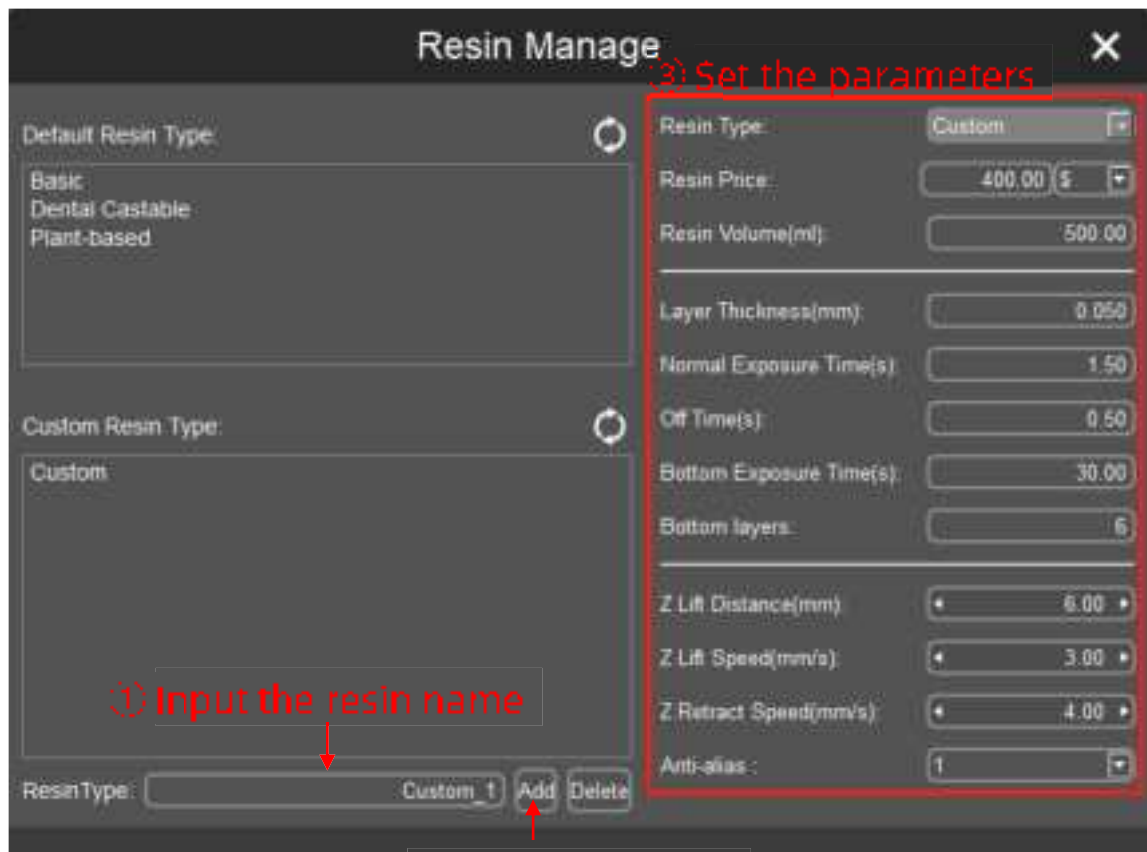
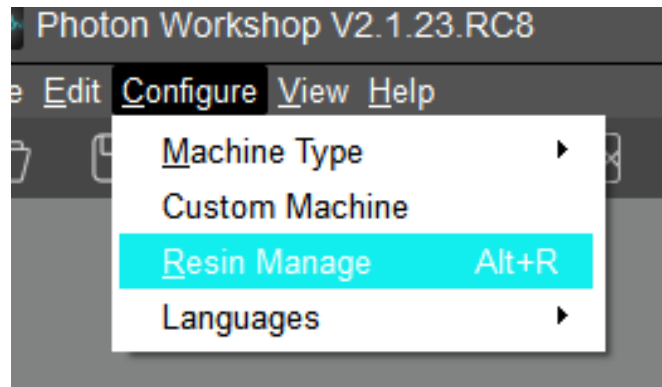
The default printing mode is "Normal Mode". In this mode, four resin types are preset—"Basic", "Plant-based", "Dental Castable" and "Custom". Different resin types have different printing parameters. You can directly select the "Resin Type" according to which Anycubic Resin you are using.

Resin Type:	Basic	▼
Resin Price:	400.00	\$ ▼
Resin Volume(ml):	500.00	
<hr/>		
Layer Thickness(mm):	0.050	
Normal Exposure Time(s):	1.50	
Off Time(s):	0.50	
Bottom Exposure Time(s):	30.00	
Bottom layers:	6	
<hr/>		
Z Lift Distance(mm):	◀ 6.00 ▶	
Z Lift Speed(mm/s):	◀ 3.00 ▶	
Z Retract Speed(mm/s):	◀ 4.00 ▶	
Anti-alias :	1	▼

Introduction to Slicing Software

You can always modify the parameters.

And, you can add custom resin type. Click "Configure" → "Resin Manage".



② Click "Add" button

Note: Click refresh icon can restore the default parameters of the resin.

Introduction to Slicing Software

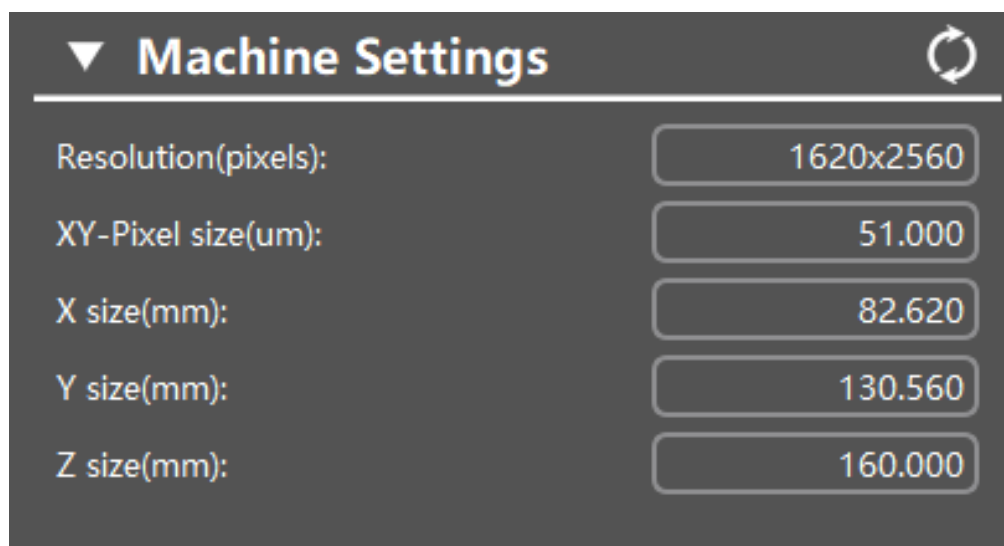
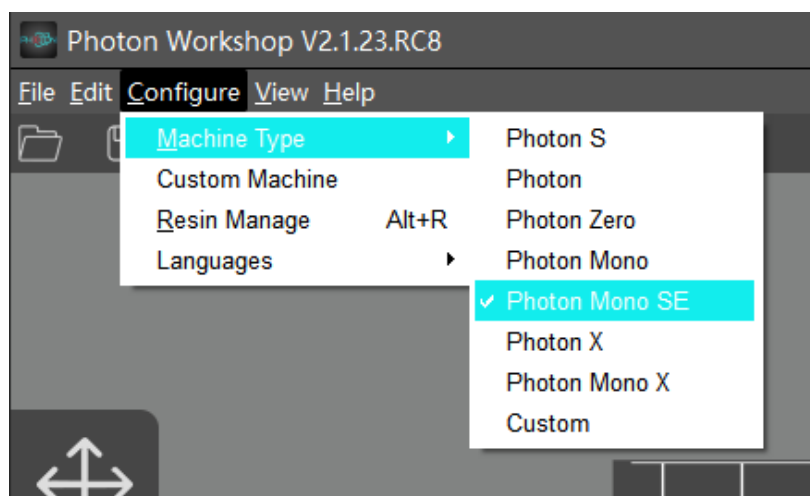
Slice parameters :

- ① **Resin Price:** Set the price according to the purchased resin.
- ② **Resin Volume:** Set according to the volume of a bottle of resin you purchased. During slicing, it will automatically calculate the resin consumption and price in total.
- ③ **Layer thickness:** The thicker the layer, the longer the exposure time for each layer.
- ④ **Normal exposure time:** Setting range: 1.5~8s, the exposure time is set according to the thickness of each layer, the complexity of the model details and the resin materials.
- ⑤ **Off time:** The UV light interval between each layer is ranged 0.5~3s.
- ⑥ **Bottom Exposure Time:** Setting range: 20~80s, the longer the bottom exposure time is, the easier the bottom layer of the model stick onto the build platform.
- ⑦ **Bottom layers:** Setting range: 3~10.
- ⑧ **Z Lift Distance:** It is suggested to set it to 6mm.
- ⑨ **Z Lift Speed:** It is suggested to set it to 3mm/s.
- ⑩ **Z Retract Speed:** It is suggested to set it to 4mm/s.
- ⑪ **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. The suggested value is 1.

Introduction to Slicing Software

② Machine Settings

These parameters seldom need modification. But if the printed model shows big dimensional error along a particular axis (X,Y or Z), you can modify the corresponding values for that axis proportionally.

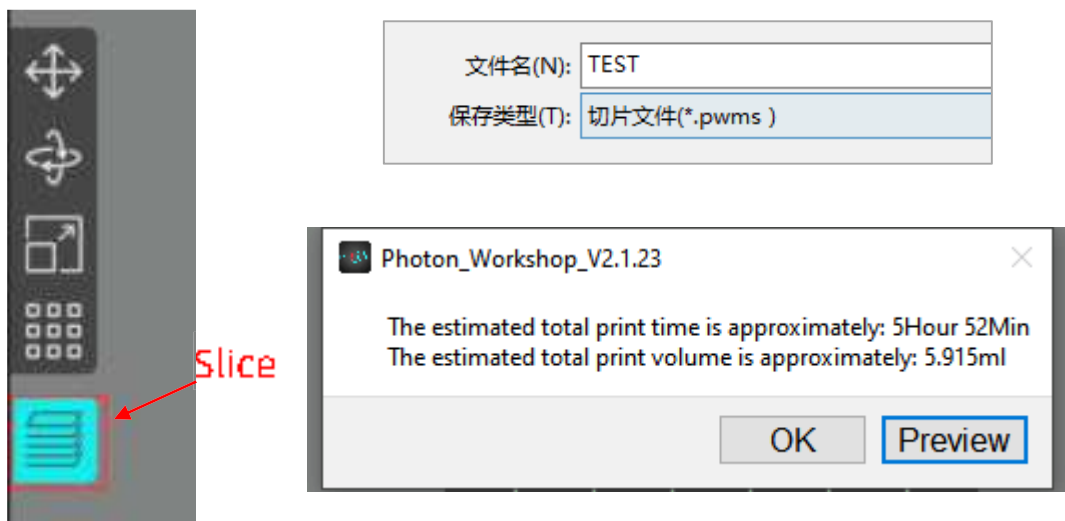


Introduction to Slicing Software

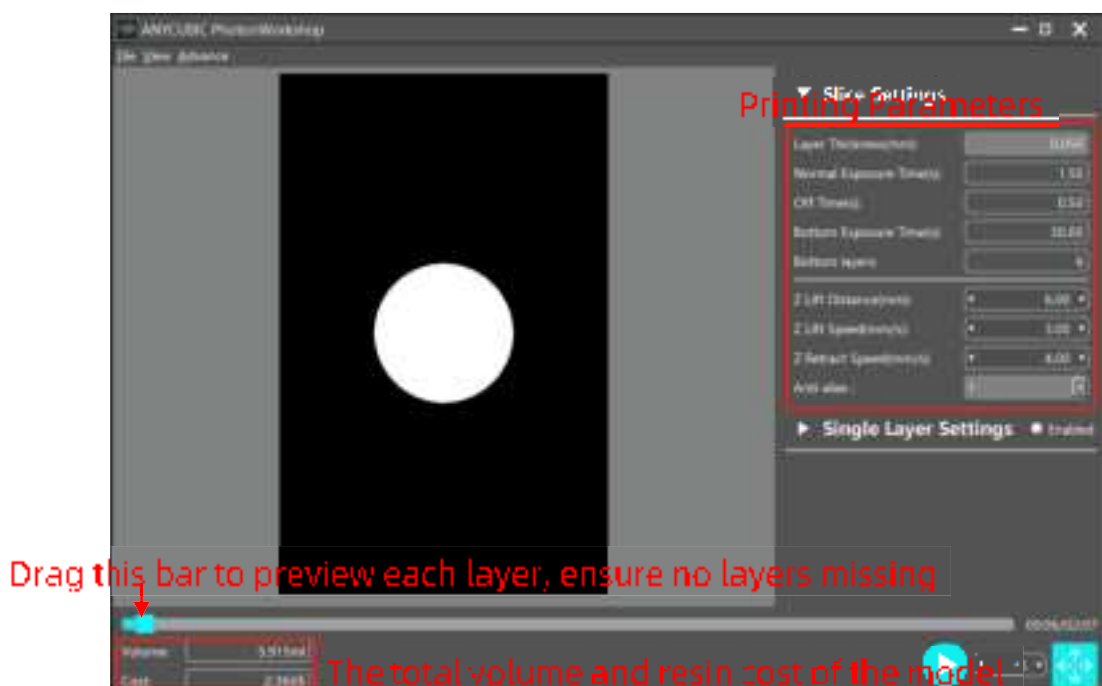
(10) Slicing

After confirm the slice settings, click the "Slice" icon at the top left (red square). You need to save the sliced file as ".pwms" so the Photon Mono SE would recognize the file.

Choose the file directory and save the ".pwms" file in the memory stick and then start slicing, and click OK to finish. You may click "Preview" to check each layer and the corresponding parameters.

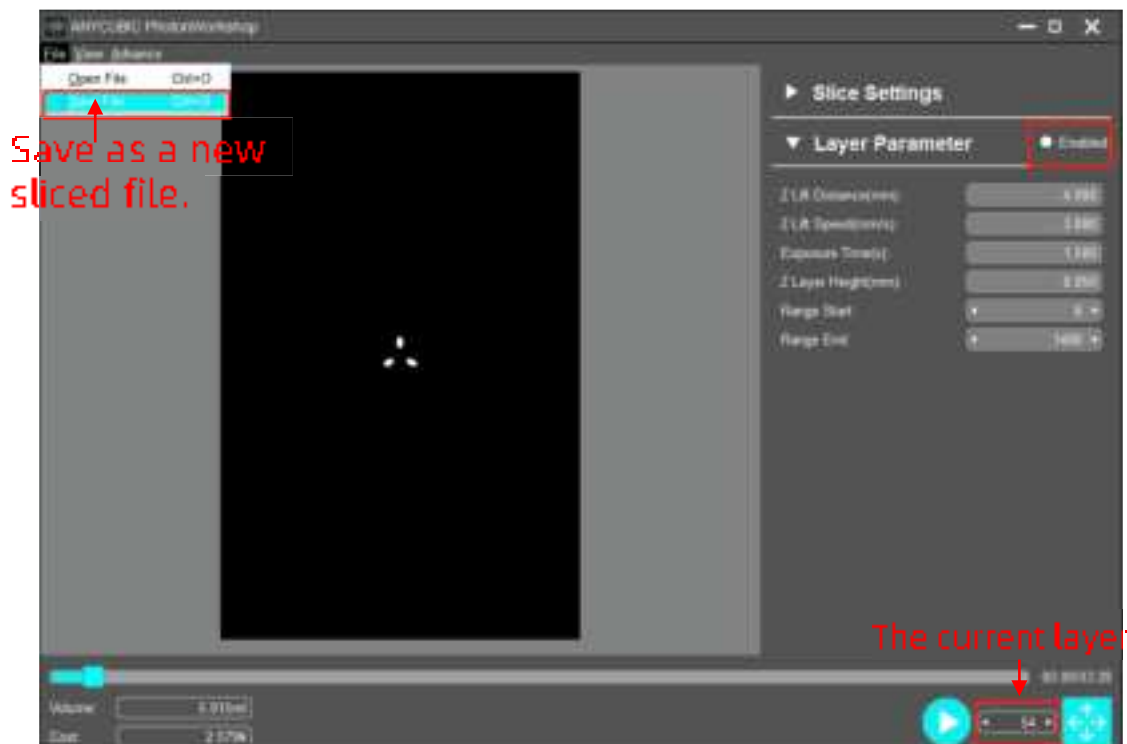


In the slice file view interface, you can preview related slice settings, machine settings, single layer settings and other information.



Introduction to Slicing Software

In the Sliced File View interface, check "Enabled" to set the exposure time, Z Lift distance and Z Lift speed of the current layer according to personal requirements. Upon finished, click "File" → "Save File" on the upper left corner to save as a new sliced file.



Note:

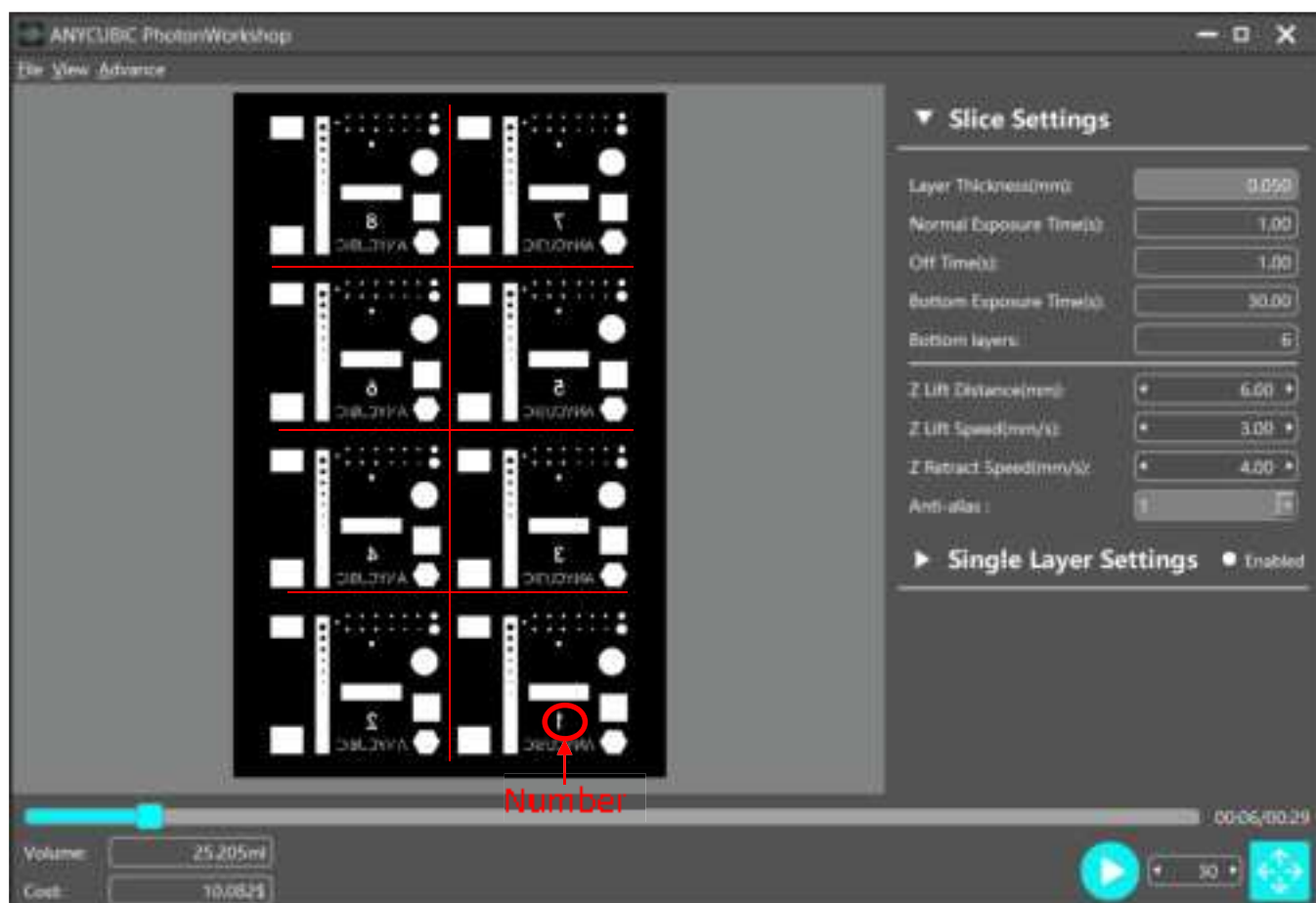
1. This function is invalid for the bottom layers. Do not use this function to bottom layers.
2. After changing the individual layer settings, the exposure parameters of the new file cannot be modified again via the printer touch screen during printing. Even if it has been modified, the change would only be valid for the current layer.

Introduction to Slicing Software

(11) R_E_R_F

“R_E_R_F” is an abbreviation for “Resin Exposure Range Finder”. This function is used to find out the optimal exposure parameters for different resins.

Import the “R_E_R_F.pwms” file into the slicing software (The file has been saved into the memory stick). Specifically, in the R_E_R_F mode, the curing screen will be divided into eight areas and each area is numbered, as shown in the picture.



Introduction to Slicing Software

- The exposure time for Area No.1 is equal to "normal exposure time (s)" in the slicing settings (exclude Bottom Exposure Time), and the exposure time for other areas will be increased by an increment of "0.4s" subsequently.
- For example, when "Normal Exposure Time (s)" is set to 1s in the slicer, the exposure time for Area No.1 is 1s, the exposure time for Area No.2 will be 1.4s, and so on, and the exposure time of Area 8 will be 3.8s.
- You can modify the exposure time of Area No.1 by modifying the "Normal Exposure Time (s)" parameter, and this action is also valid during printing.

The normal exposure parameter of "R_E_R_F.pwms" file attached to memory stick is 1, and users can print this file directly for testing.

After printing, take off the model, wash it with ethanol 95vol% concentration, and then observe it. The exposure parameters of the model with the best printing effect are the best exposure parameters of the resin.

Note:

DO NOT change the file name of "R_E_R_F", because the Photon Mono SE can only recognize THIS file name to run this function. Also, do not name other unrelated file as "R_E_R_F".

FAQ and Machine Maintenance

1. FAQ

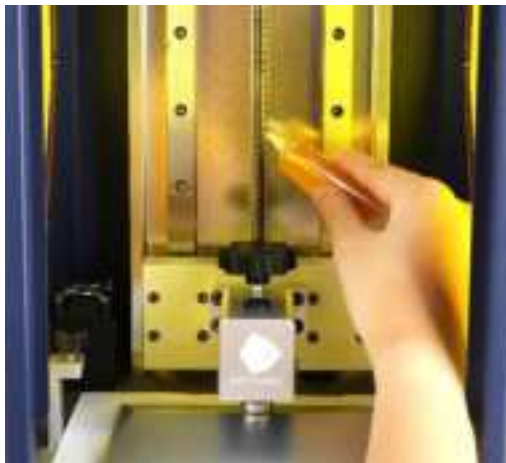
(1) Model not sticking to platform

- Bottom exposure time is insufficient, increase the exposure time.
- Contact area between the model and platform is small, please add raft.
- Bad leveling

(2) Layer separation or splitting

- The machine is not stable during printing
- FEP film in the vat is not tight enough or it need a change for new one
- The printing platform or vat is not tightened

2. Machine maintenance



(1) If Z axis make noisy sound, please apply lubricant to Z lead screw.



(2) Do not use sharp objects to scrape off the residues on the FEP film.

FAQ and Machine Maintenance



(3) Be careful when remove the platform, do not let it fall onto the 2K LCD screen.



(4) Do not left the resin in the vat for over two days if not using it. Please filter and store the resin properly.

(5) After printing, please clean up the platform (**wipe clean with paper towels or wash with alcohol**), and ensure no residue left before next print (**filter the residue with funnel**).

(6) If the outside of printer is stained with resin, use alcohol to wipe clean.

(7) To switch the resin colors, please clean the original resin vat first.

Thank you for purchasing ANYCUBIC products! Under normal usage and service, the products have a warranty period up to one year. Please visit ANYCUBIC official website(www.anycubic.com) to report any issue with ANYCUBIC products. Our professional after-sale service team would respond within 24 hours and solve the issues.

FCC Caution:

This device complies with part 15 of the FCC Rules. Operation is subject to the following two conditions: (1) This device may not cause harmful interference, and (2) this device must accept any interference received, including interference that may cause undesired operation.

Any Changes or modifications not expressly approved by the party responsible for compliance could void the user's authority to operate the equipment.

Note: This equipment has been tested and found to comply with the limits for a Class B digital device, pursuant to part 15 of the FCC Rules. These limits are designed to provide reasonable protection against harmful interference in a residential installation. This equipment generates uses and can radiate radio frequency energy and, if not installed and used in accordance with the instructions, may cause harmful interference to radio communications. However, there is no guarantee that interference will not occur in a particular installation. If this equipment does cause harmful interference to radio or television reception, which can be determined by turning the equipment off and on, the user is encouraged to try to correct the interference by one or more of the following measures:

- Reorient or relocate the receiving antenna.
- Increase the separation between the equipment and receiver.
- Connect the equipment into an outlet on a circuit different from that to which the receiver is connected.
- Consult the dealer or an experienced radio/TV technician for help.

Radiation Exposure Statement

This equipment complies with FCC radiation exposure limits set forth for an uncontrolled environment. This equipment should be installed and operated with minimum distance 20cm between the radiator and your body.

This transmitter must not be co-located or operating in conjunction with any other antenna or transmitter.