Anycubic Photon Workshop

User Manual

Team ANYCUBIC

Contents

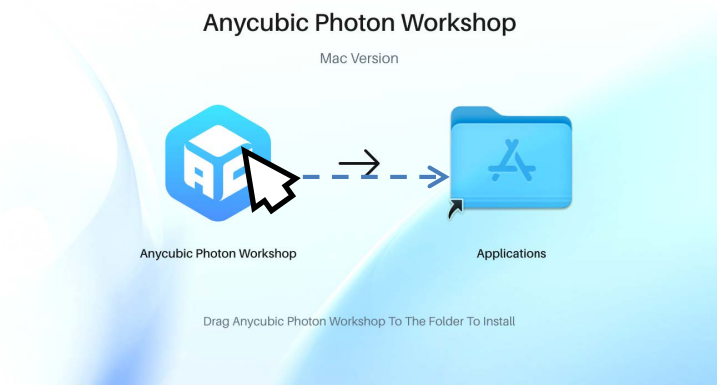
Anycubic Photon Workshop installation package is located in memory stick, please install and update the software as following steps. Do not run the older versions of Anycubic Photon Workshop in case of installation failure.

1. Installation

- **Windows**

Close the anti-virus software before installation. Open the suitable package and then follow the guide as shown below.

- **Mac**



Double click the installer and then drag Anycubic Photon Workshop to the applications as shown above

Anycubic Photon Workshop V4.X.X System Requirements

Windows

CPU	Intel® Core™ i5 6600K or higher AMD Ryzen™ 5 1600 or higher
RAM	≥ 16 GB
Free Disk Space	2 GB
Display Resolution	≥ 1920*1080 ≥ 2560*1440 (suggested)
GPU	NVIDIA GeForce GTX1050 or higher AMD Radeon RX480 or higher
GPU RAM	≥ 1GB

Mac OS

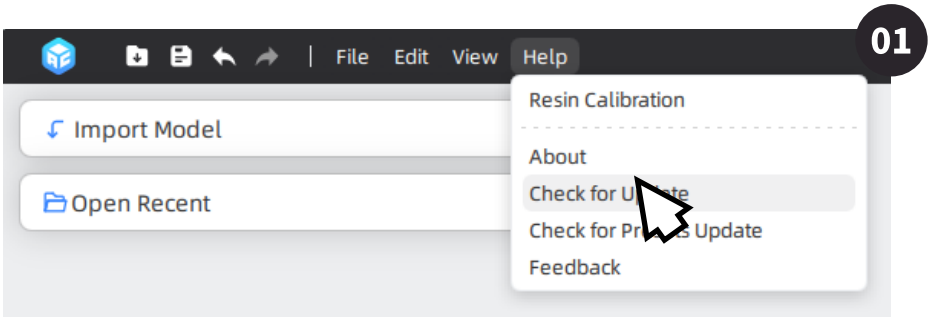
CPU	Intel® 4-Core (OS version 10.15) or higher Apple M1 4-Core (OS version 13.0) or higher
RAM	≥ 16 GB
Disk Space	≥ 64 GB
Display Resolution	≥ 2560*1440

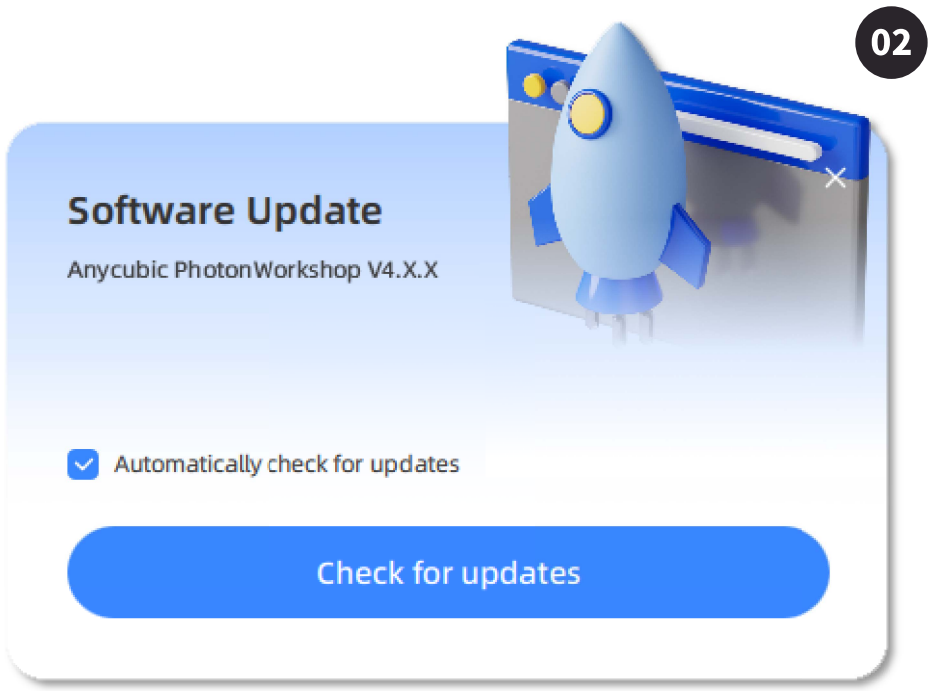
2. Language

Click “偏好” - “偏好设置” - “语言” to switch the language to English if the language is set as Chinese.

3. Update

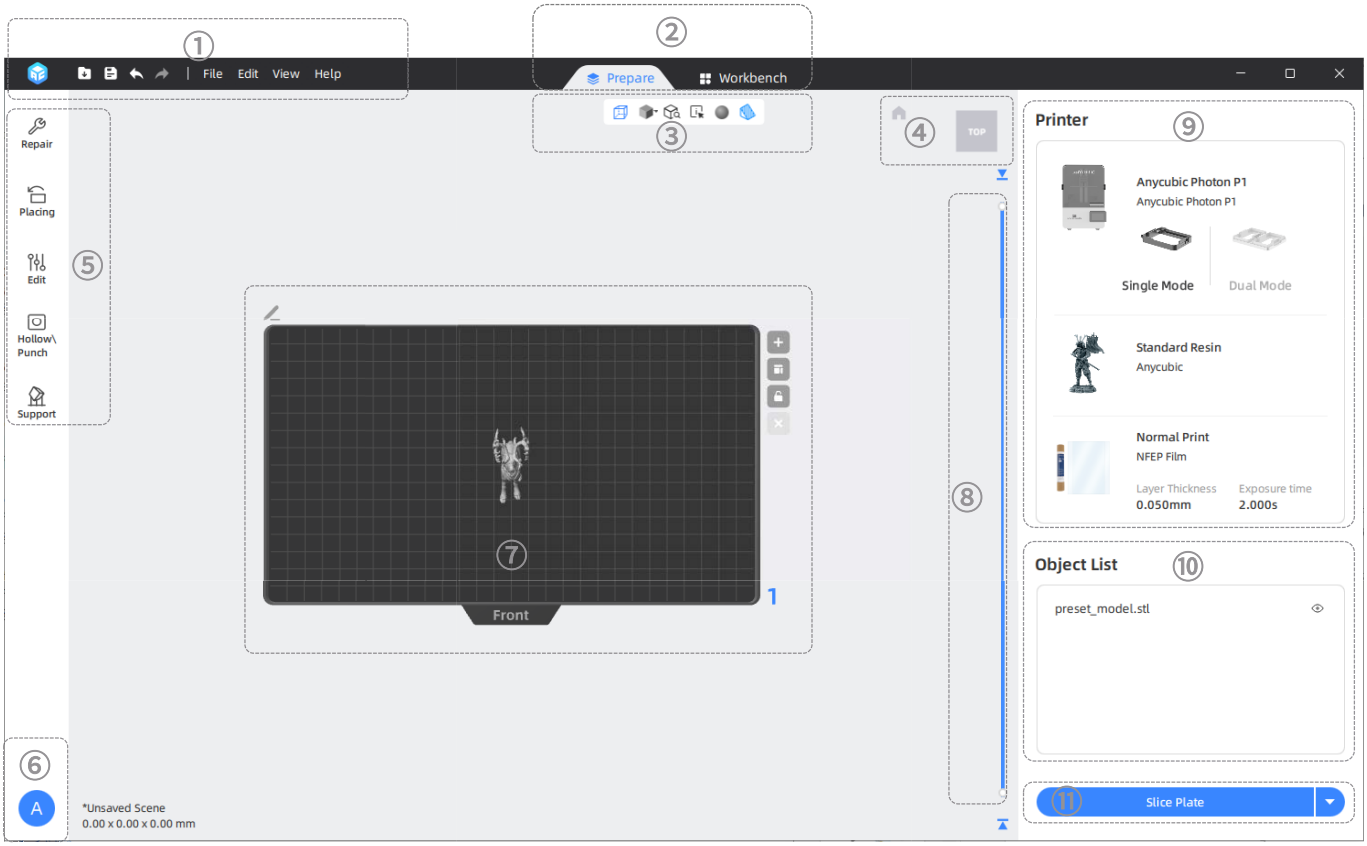
If a new version is released, it will be a popup asking you to update when you open the software. You can also click Help - Check Update to check whether the software update to the latest version. If the automatic update is unnecessary, cancel the selection of “Automatically check for updates”.





Notice: Anycubic Photon Workshop and its instructions may be updated irregularly. Please visit www.anycubic.com for the latest updates.

Overview



① Open/save file, undo/redo

② Switch to Prepare/Workbench interface

③ View mode

④ View switcher

⑤ Repair, placing, edit, hollow/punch and support settings

⑥ Login/logout Anycubic Cloud

⑦ 3D model preview

⑧ Drag the slider to preview each layer of the model

⑨ Configuration of machine, resin and slice parameters

⑩ Object list

⑪ Slice button: slice plate/slice all

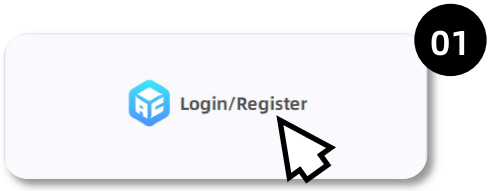
1. Add Printer

Cloud /LAN mode are only available on printers that supports these features.

Check the product detail page to confirm.

Before starting, ensure the printer is connected to the network.

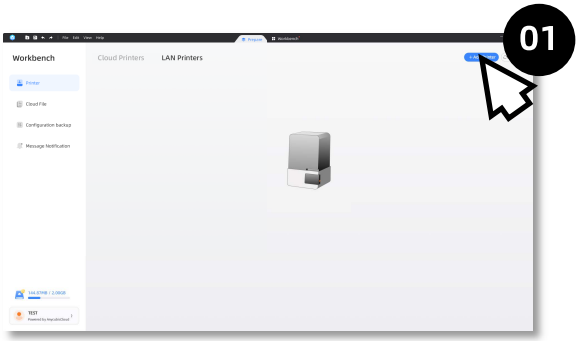
① Log in Anycubic Cloud.



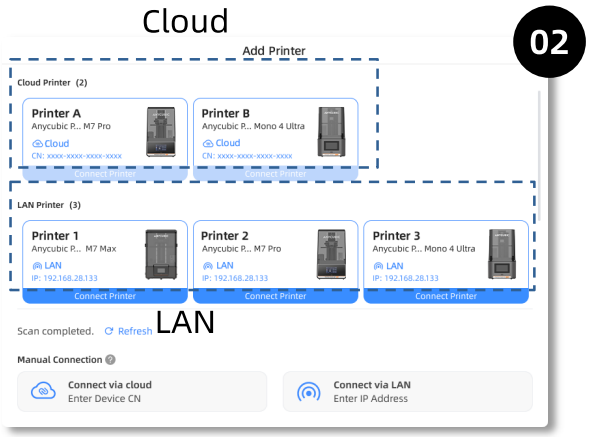
Click to enter in log in interface

Enter your account and password then click log in

② Add printer.



Cloud



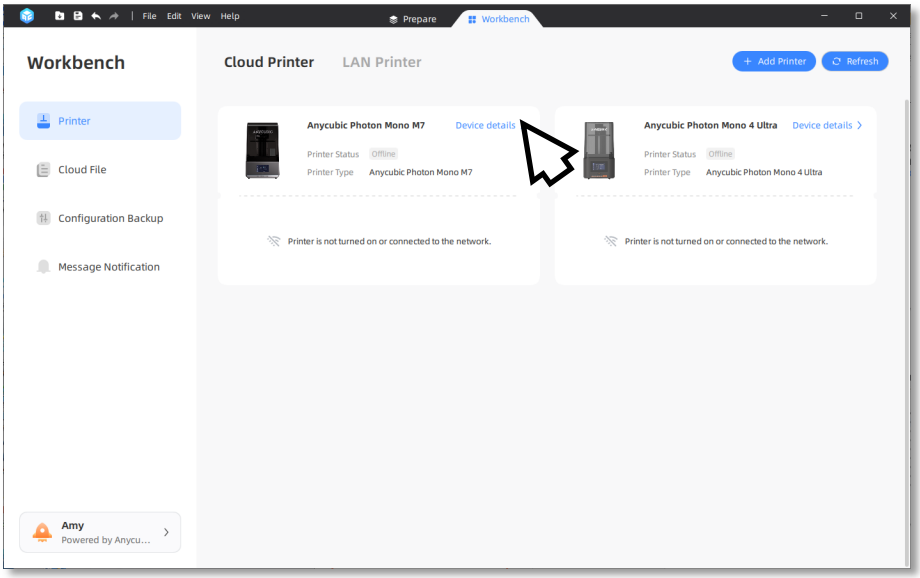
LAN

LAN mode and Cloud mode cannot be used at the same time. When a printer links via LAN, it will be disconnected in Cloud.

③ Slice the model and start the print job on cloud printer/ LAN printer.

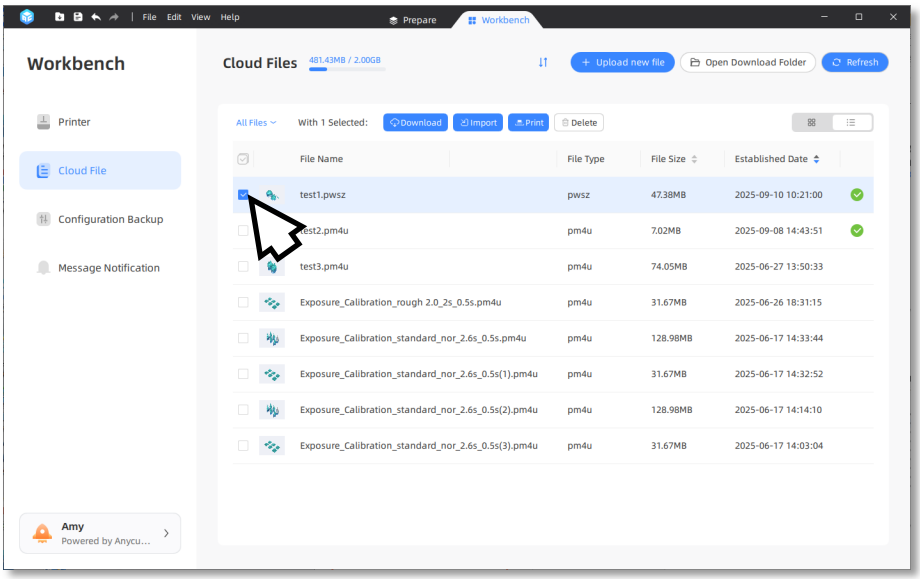
2. Printer Management

View the connection status and print progress in Cloud Printer/ LAN Printer.



3. Cloud Files Management

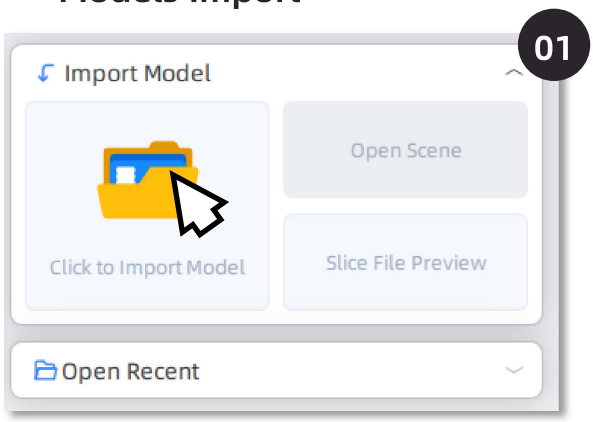
You can save the sliced files to the cloud here.

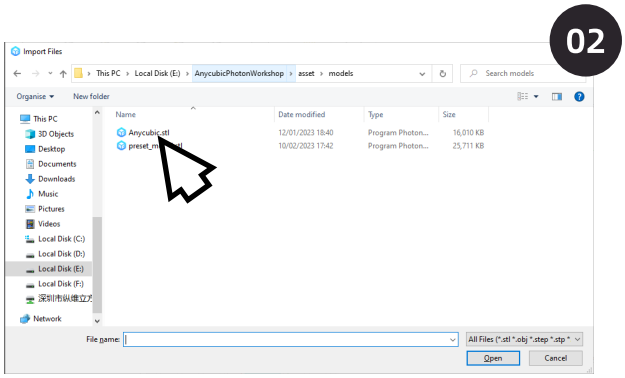


Edit cloud files

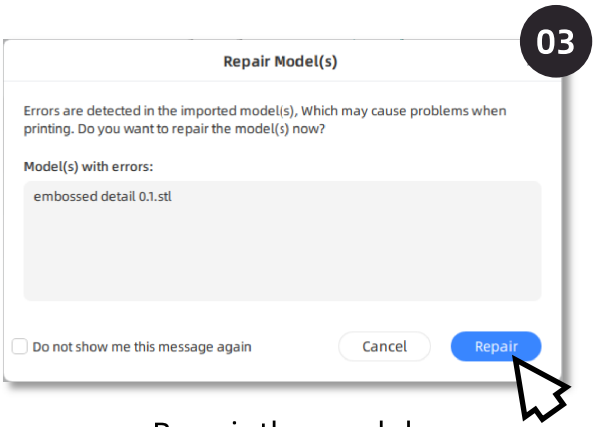
1. Import

- **Models Import**

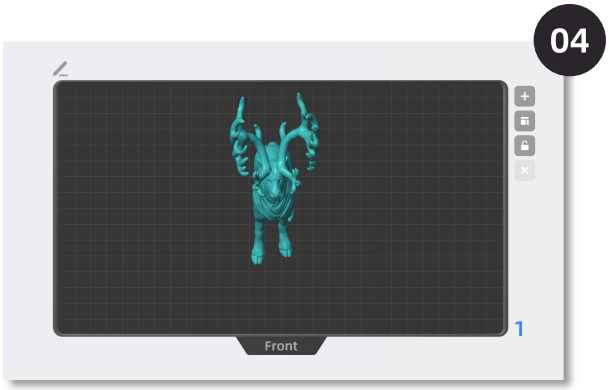




Select a file



Repair the model

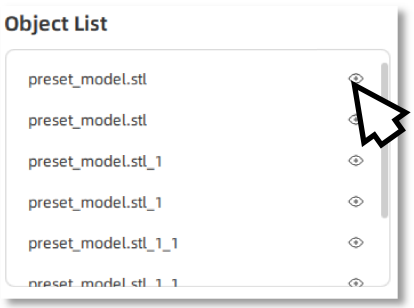


The object is imported

If the automatic models quality detection is unnecessary, cancel it in File-Preferences-Model Repair.

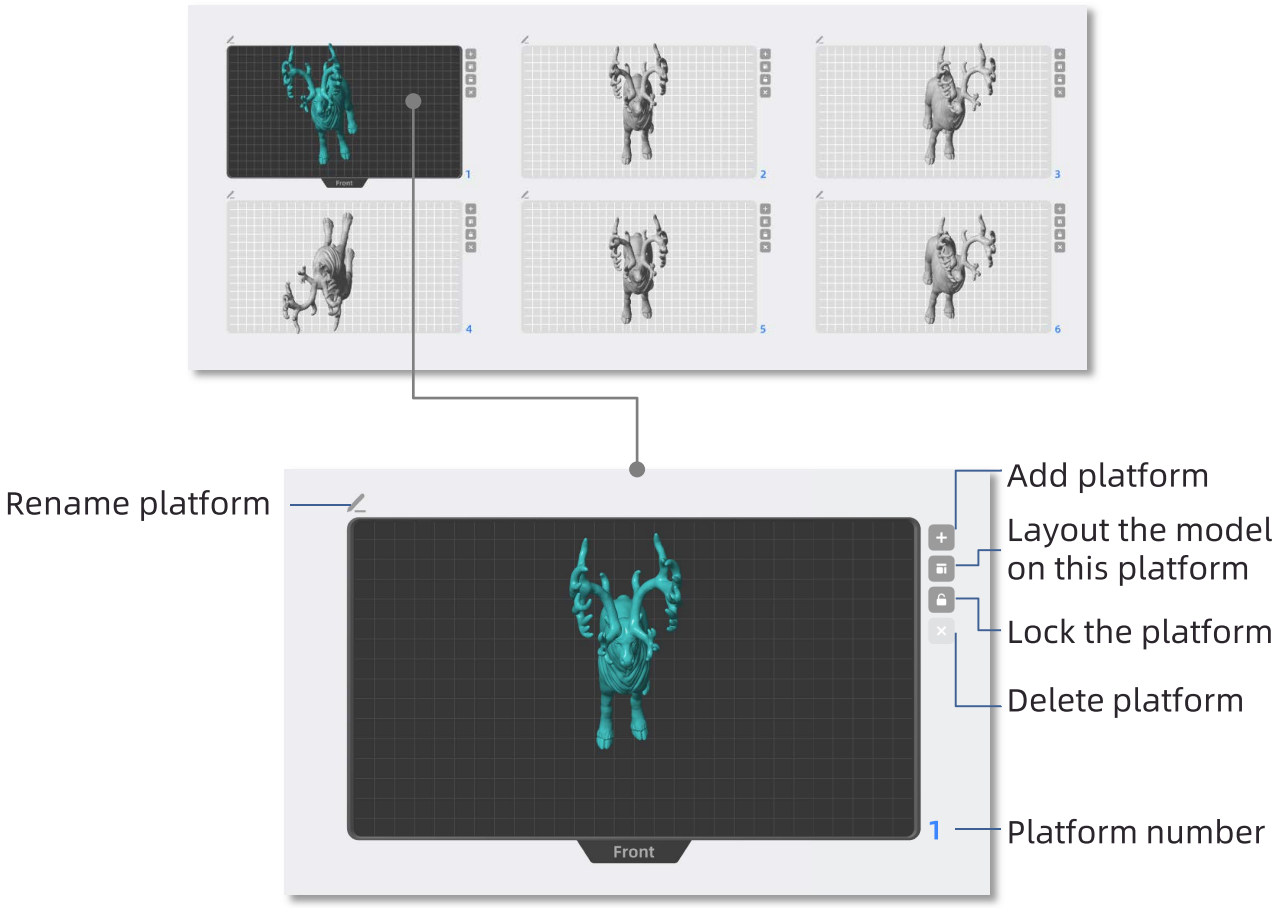
- **Object List**

When models are selected, they are highlighted. Click the eye icon to hide a model. Hidden models cannot be edited, have supports added, or be sliced.



- **Platform Management**

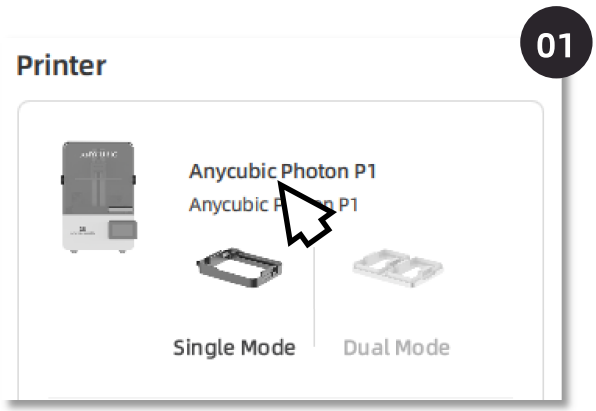
Add multiple platforms to place models for slicing.

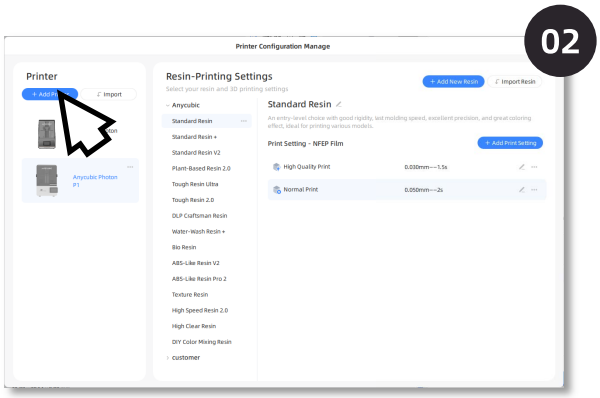


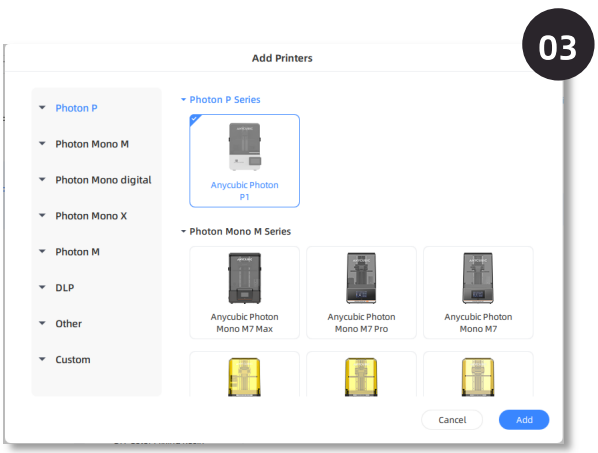
2. Machine Configuration Management

① Machine settings

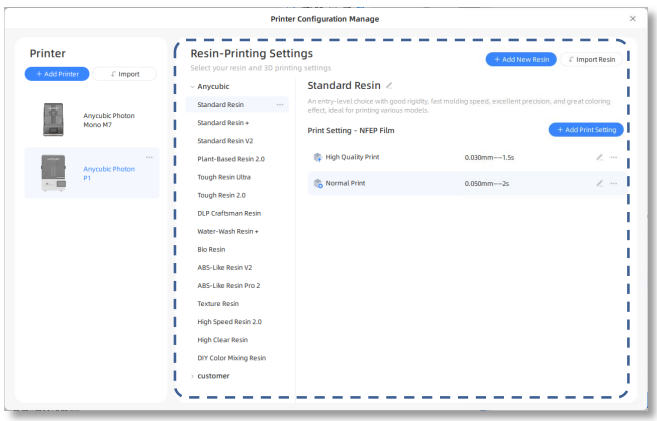
Enter in Machine Configuration Manage to select the printer model. Different printer types have different parameters, please choose the printer you use to avoid print failure.







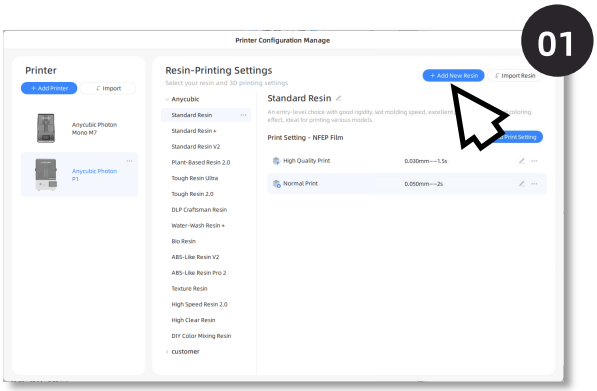
② Resin settings

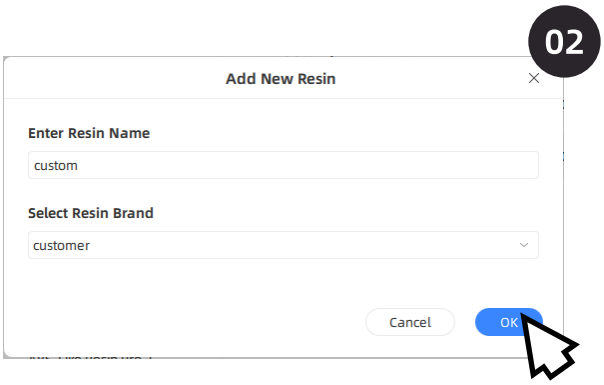


Select the parameters according to the resin type

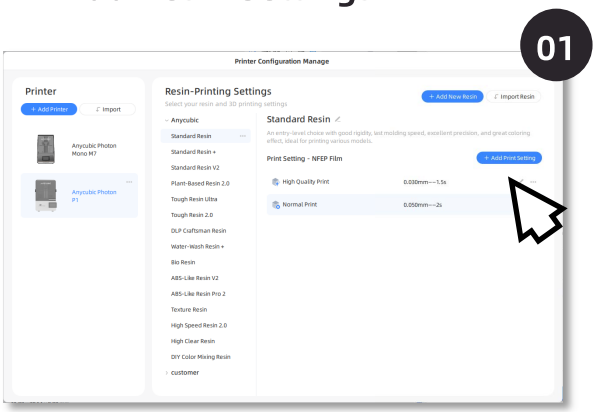
③ Resin Settings Configuration (optional)

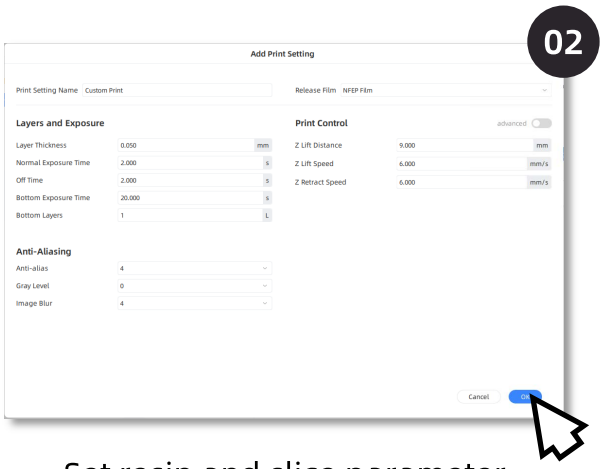
- **Add Resin**





- **Add Resin Settings**





Set resin and slice parameter
Then click OK to finish

④ Slice parameter

Set the slicing parameters according to the resin characteristics and model geometry to improve the printing success rate.

Layer and Exposure

- **Layer Thickness:** The thinner the layer, the higher the Z-axis resolution. The thicker the layer, the longer the exposure time for each layer.
- **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.
- **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.
- **Bottom Exposure Time:** The longer the bottom exposure time is, the easier the bottom layers of the model stick to platform.

The wait time allows low-fluidity resin to reflow.

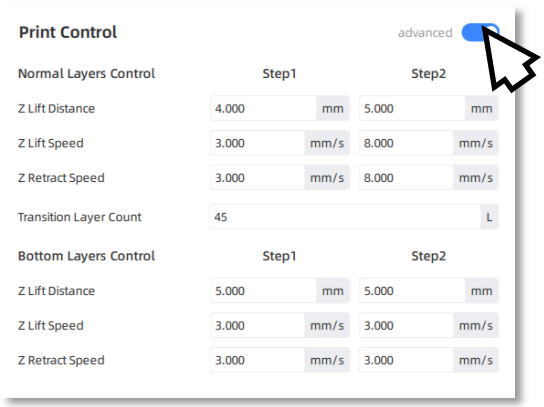
- **Off Time:** The UV light will be off for a certain time between each layer. The longer off time allows resin with poor fluidity to reflow.
- **Wait Before Lift:** The time gap starting from the end of exposure to when the platform starts to rise.
- **Wait After Lift:** The time gap starting from when the platform stops rising to when it starts to lower down.
- **Bottom Off Time:** The UV light will be off for a certain time before starting exposure each bottom layer.
- **Bottom Wait Before Lift:** The time gap of bottom Layers starting from the end of exposure to when the platform starts to rise.
- **Bottom Wait After Lift:** The time gap of bottom layers starting from when the platform stops rising to when it starts to lower down.

Print Control

- **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.

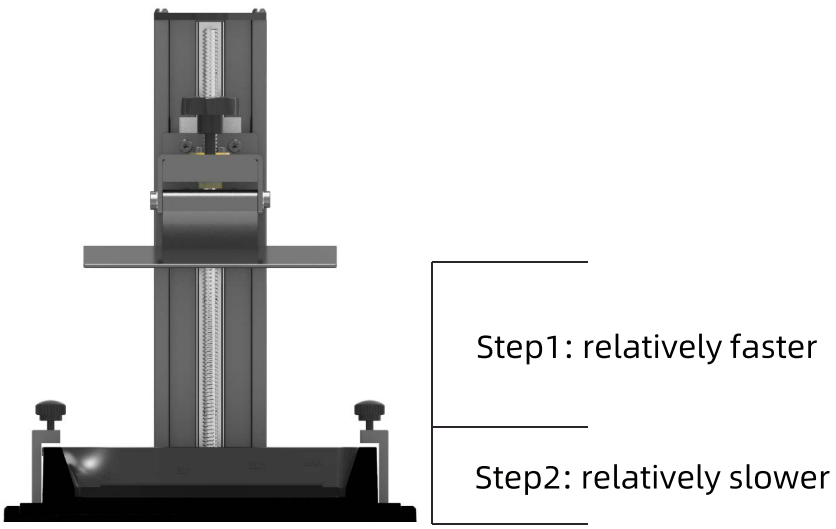
Print Control-Advanced

In basic mode, the movement parameters for the normal layers and the bottom layers are the same. By switching to advanced mode, you can configure the Z-axis movement in more detailed stages.



- **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.

- **Step1:** Movement near the curing surface. The speed of this stage is relatively slow to avoid affecting the printing.
- **Step2:** Movement away from the curing surface. The speed of this stage is relatively fast to shorten the printing time.



*Each Z lift height in Step1 and Step2 corresponds to distance of two printing platform's motion.

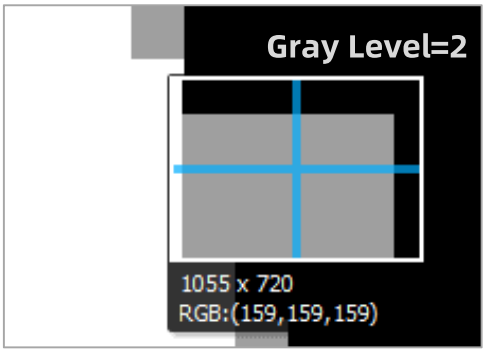
Advanced Parameters

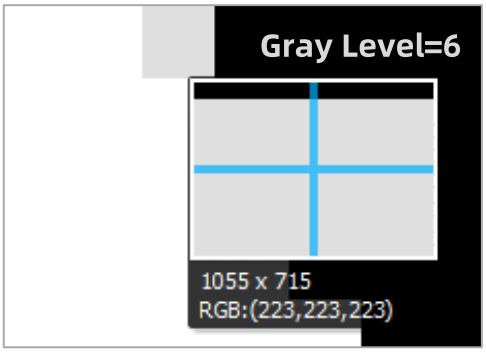
Anti-aliasing

- **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.
- **Use Random Erode Shell:** This option can only be enabled when the anti-alias value is 1 to get a matte surface.

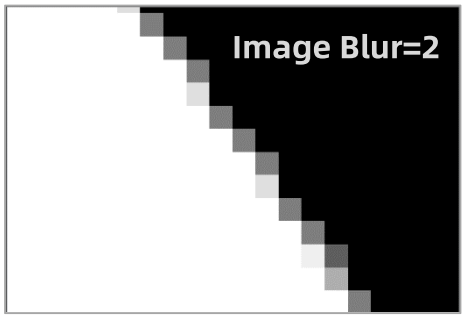
When anti-alias value is larger than 1, you can set the grades of gray level and image blur according to the requirement.

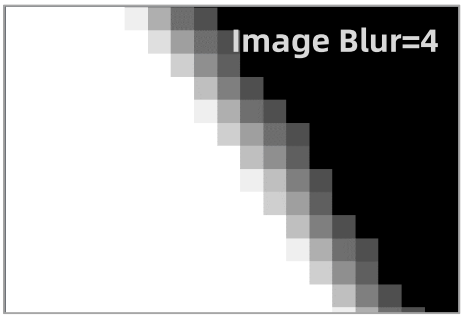
- **Gray Level:** The higher the gray level, the brighter the pixels of anti-alias.





- **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.

Shrinkage Compensation

Scale the model in the X/Y/Z directions to compensate for shrinkage or expansion caused by resin characteristics.

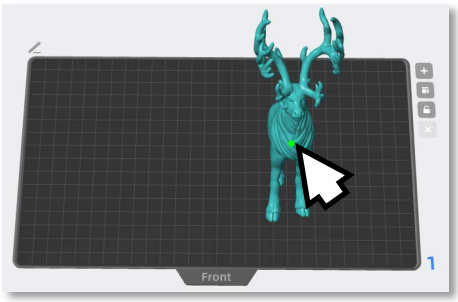
Some parameter groups have preset values. Turn off Auto to adjust.

The default value is 100%. Values above 100% enlarge, below 100% reduce.

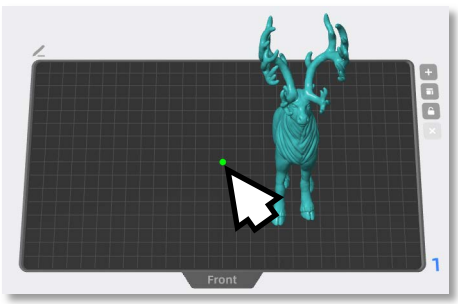
1. View Changing

① View angle

- **Mouse:** Scroll the mouse wheel to zoom in/out. Left-click and drag the platform to move it. Right-click and drag to rotate the view. Right-click and drag on the model to rotate the model.

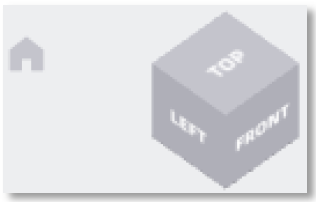


Model as the center

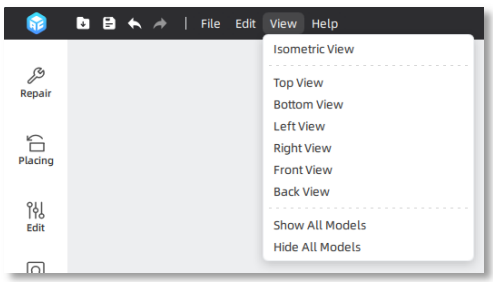


Platform as the center

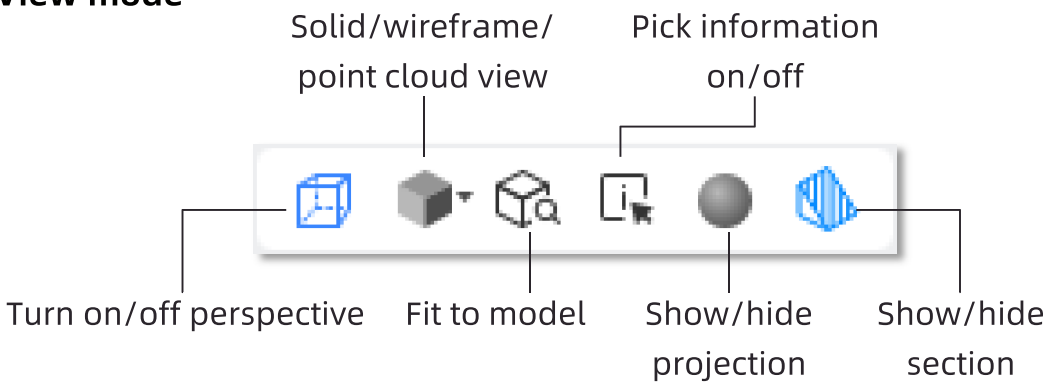
- **Interface controls:** click the surfaces of cube to change view angle; click home icon to switch to isometric view.



- **View menu:** Switch to different views.

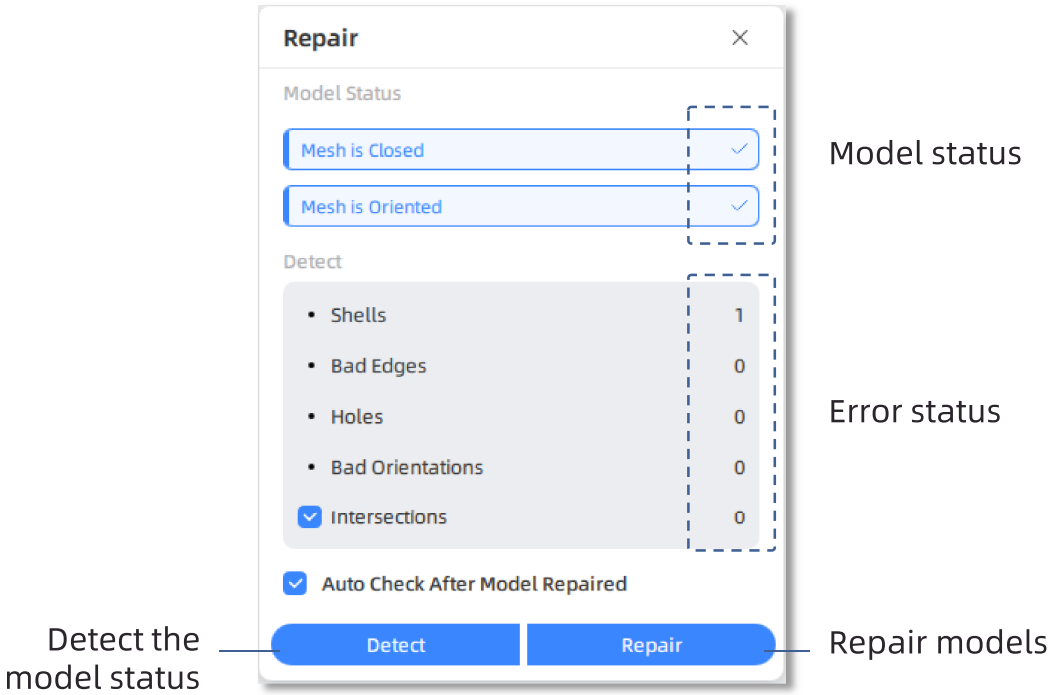


② View mode



2. Repair

Check if there are problems such as bad orientation, bad edge, hole, intersection and repair the models to increase the success rate of printing.



3. Placing

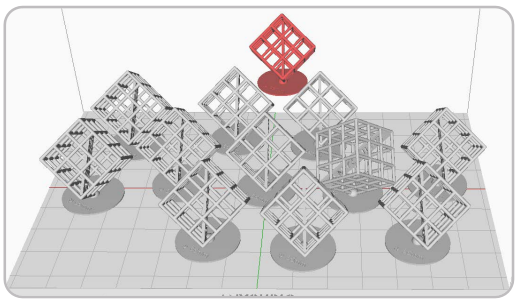
Move: Input a number or manipulate the controls to move the model.

Rotate: Input a number or manipulate the controls to rotate the model. Click “Rotate by Face” to select a face to align model to the build plate.

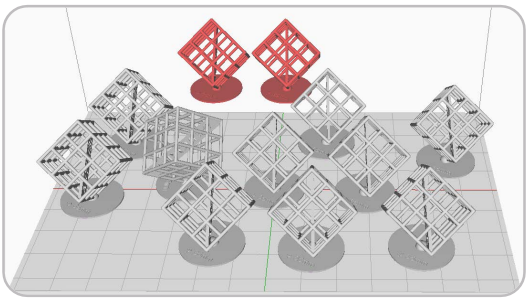
Scale: Input a number or manipulate the controls to scale the model. Click “Scale to Fit” to scale the model to its maximum size for the printer.

Layout: Place the models according to the settings of model interval, order, position and so on. It increases the space utilization to print more models in one time.

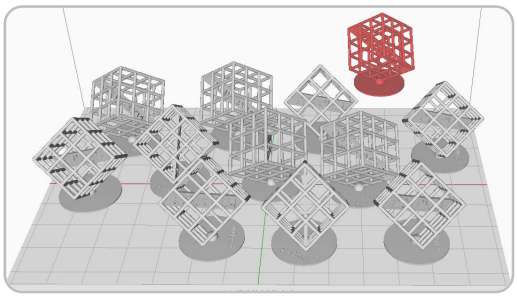
- The Bigger the model spacing, the less the models can be placed.
- Rotate the models on Z-axis to increase the space utilization.
- Set the placing order according to the personal requirements.



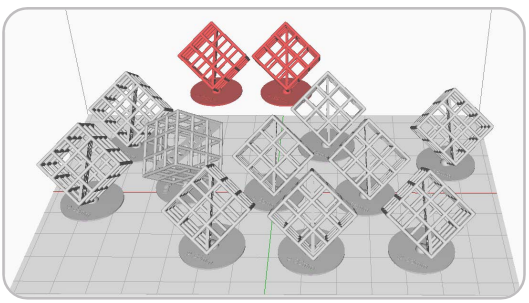
Model Spacing 2mm



Model Spacing 3mm



Rotate 45°

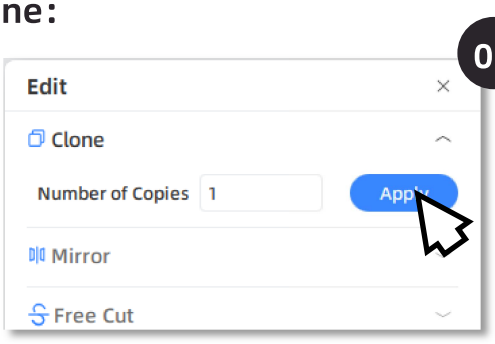


No Rotation

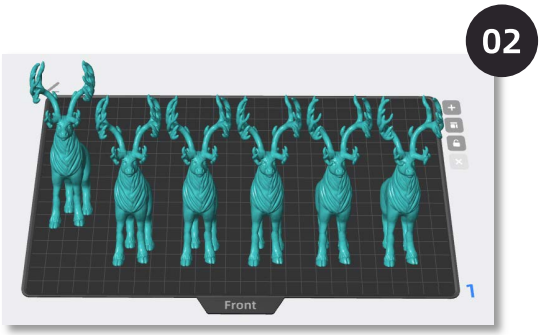
Notice: The red part out of the print range is not printable.

4. Edit

Clone:

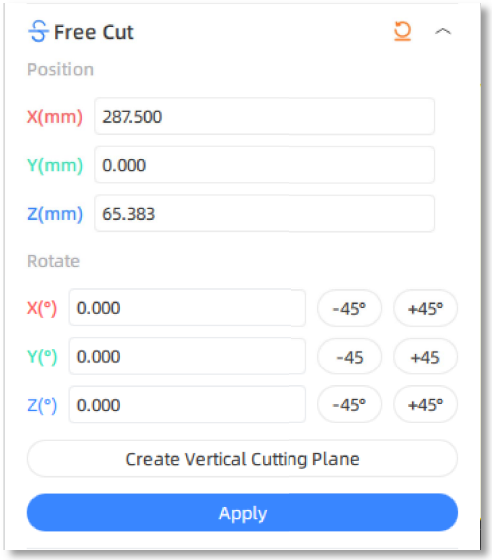


Set the number and apply

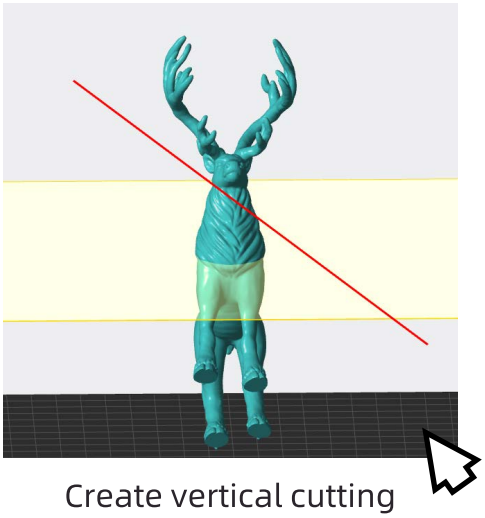


Mirror: Mirror the model in X, Y or Z direction.

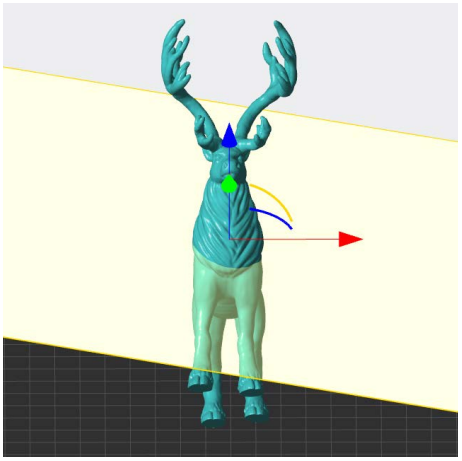
Free Cut: Adjust the cut facet by the following three methods.



Adjust the cut facet by modify the parameters

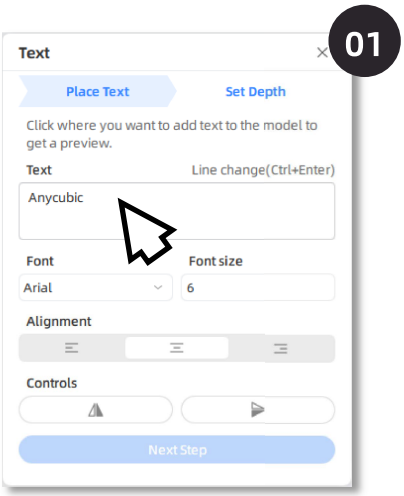


Create vertical cutting plane

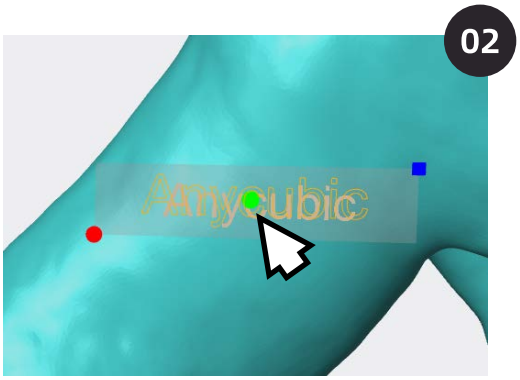


Adjust the cut facet by control

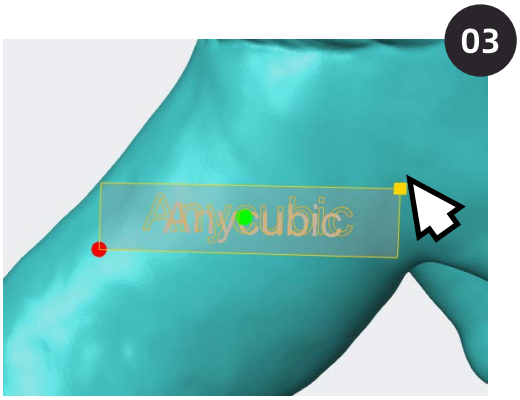
Text:



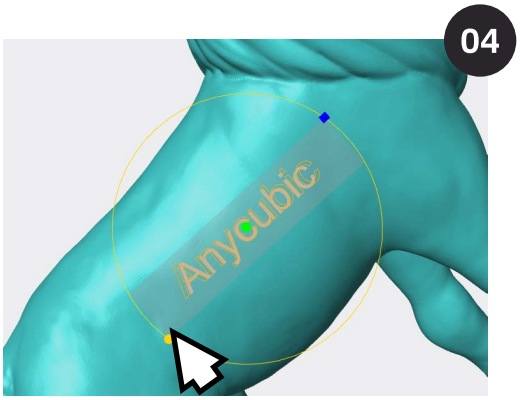
Set the text



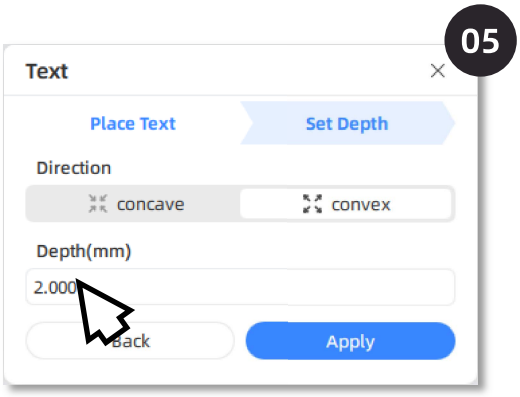
Click on the place to add text

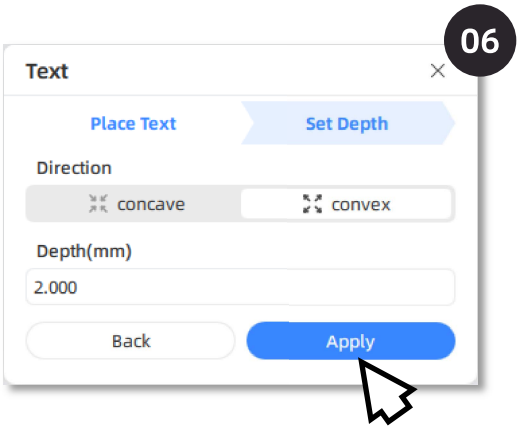


Scale



Rotate



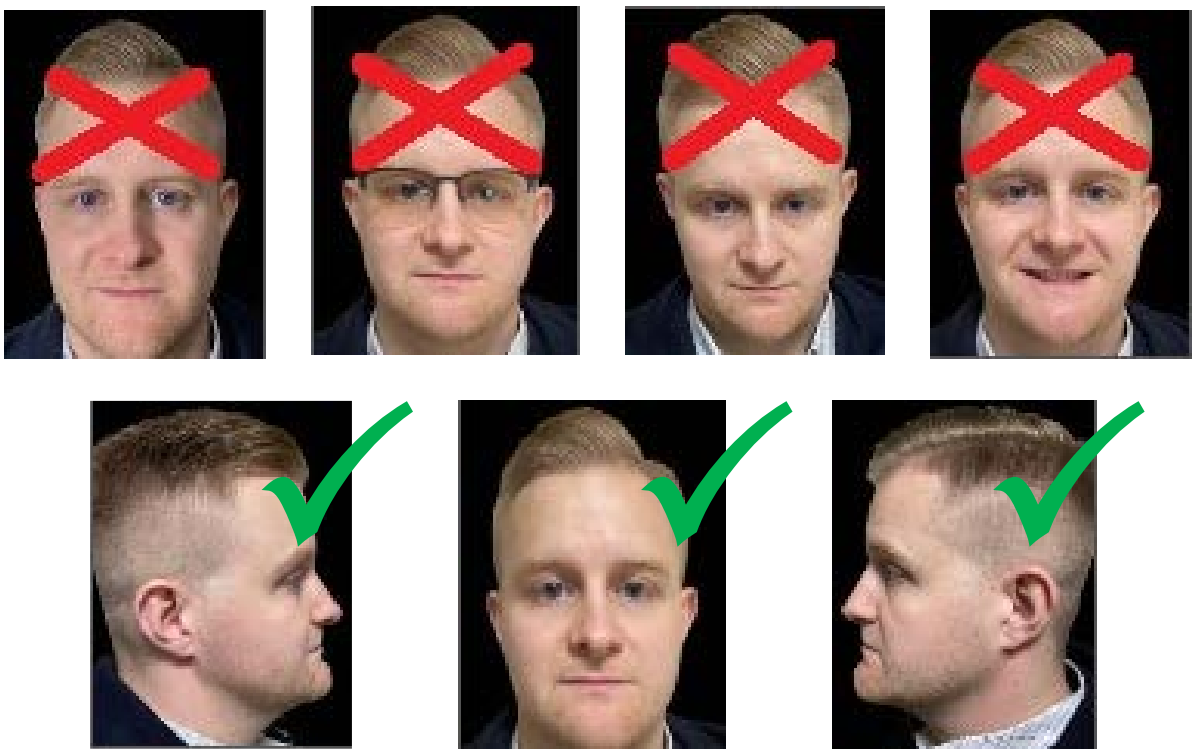


Face Model:

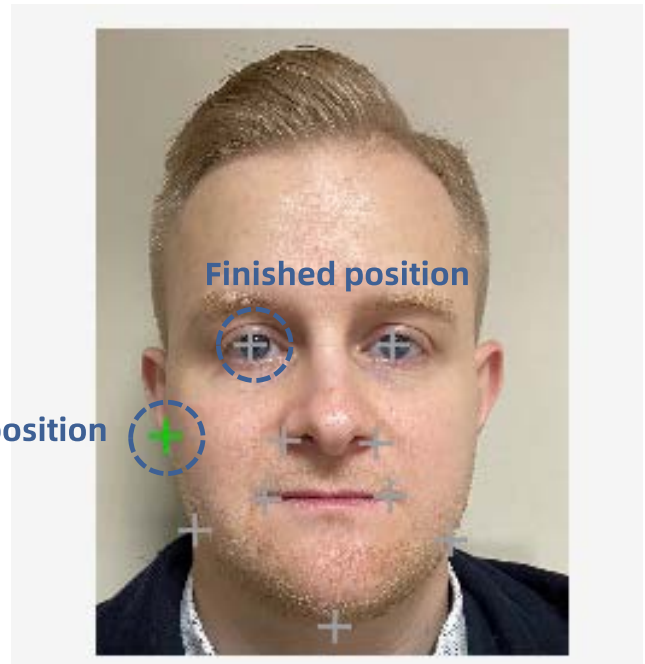
It is the function reconstructing a face from 2D pictures into a 3D form.

① Upload the proper photos according to the requirements 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.



② Click the corresponding points in accordance with the illustration and prompt to finish localization.



finished position is grey; current position is green

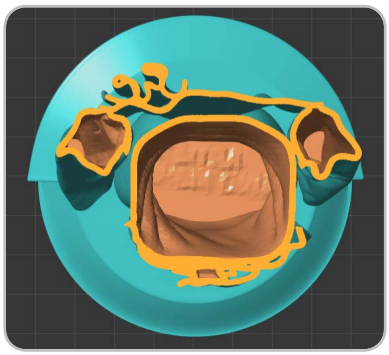
If you make a mistake, click the green cross to cancel.

③ When the steps are finished, click “Generate” to generate the face model.

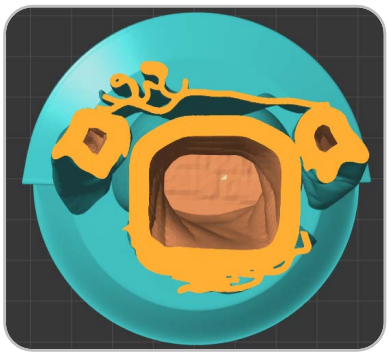
5. Hollow/Punch

Hollow:

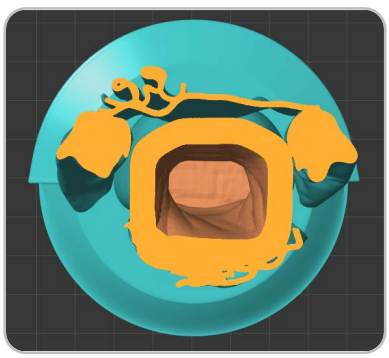
Hollow the model to reduce resin consumption. Enabling filter types can help to remove small cavities. BCC and FCC infill allow resin to drain more easily than other infill patterns.



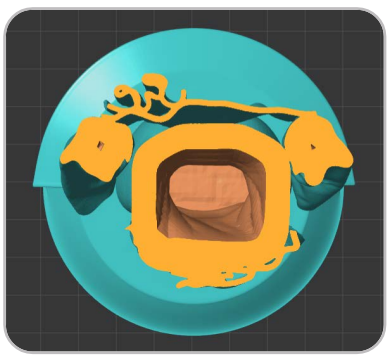
Hollow thickness: 1mm



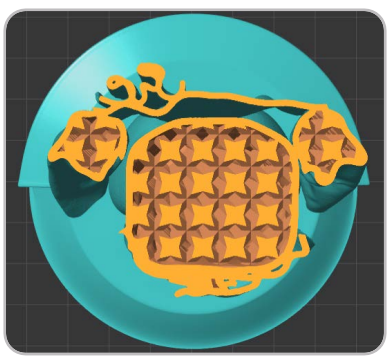
Hollow thickness: 3mm



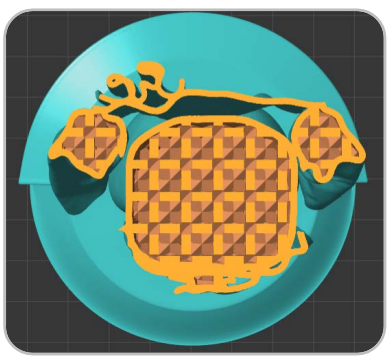
Max cavity number 1



No filter type



BCC infill

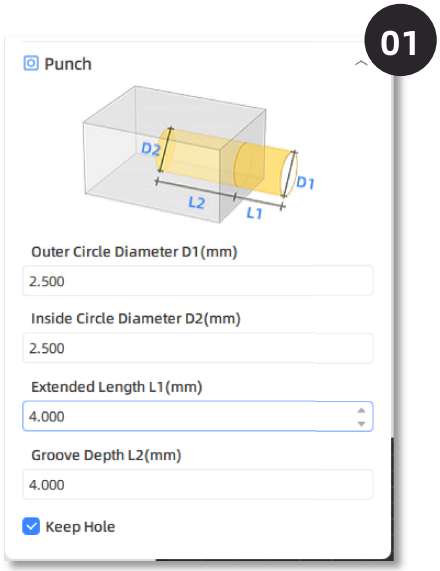


FCC infill

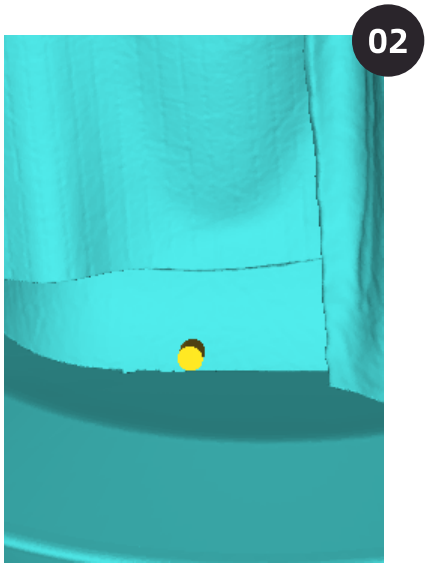
The model name: MIA
The author of the model: Fabio Nishikata

Punch:

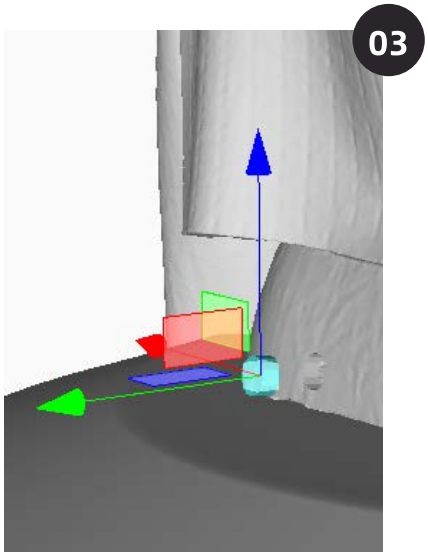
If the model is hollowed, it is suggested to pinch at the side or bottom of the model to avoid the print failure caused by vacuum seal drawing. When the print is finished, that discharge the resin inside the model can prevent model being broken after a period of time.



Set the parameters, L2 must be larger than hollow thickness



click on the model to pinch



The cylinders needs to be place properly to be printed

The author of the model: Fabio Nishikata

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

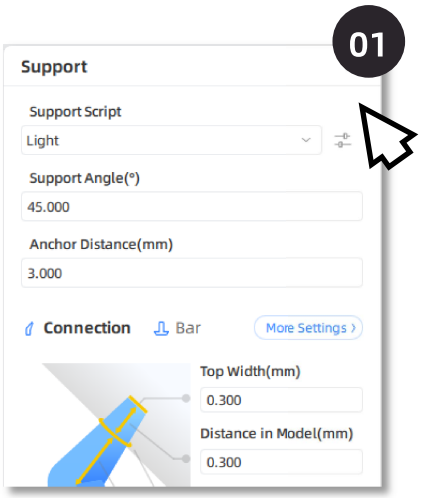
1. Support Scripts Settings

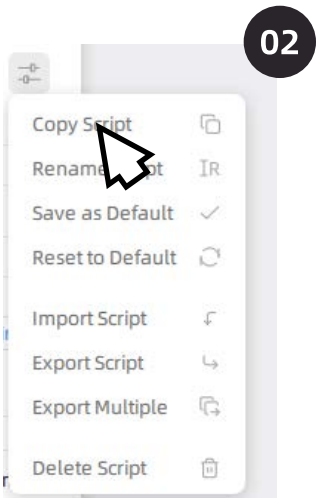
There are three support scripts: light, medium and heavy.

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

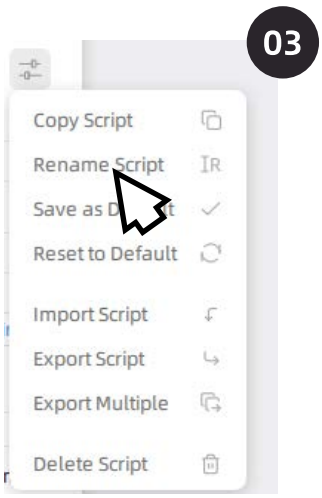
Heavy: Support contact with the model area is large and solid.

It is recommended to try the "Medium " first, and use the default settings. Also, you can add the support scripts and modify the parameters to fit your requirement.

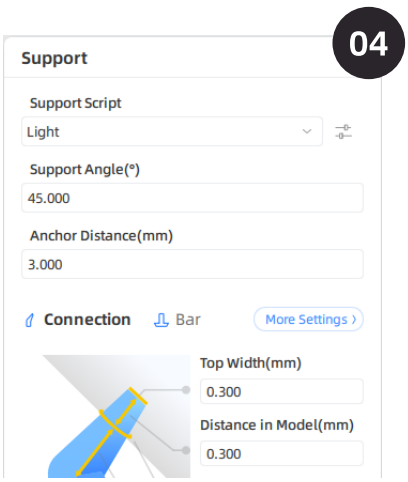




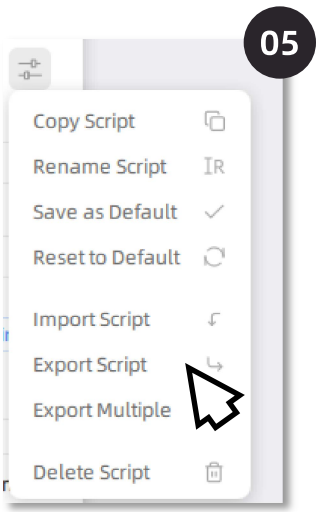
Copy a script



Rename



Set the support parameters



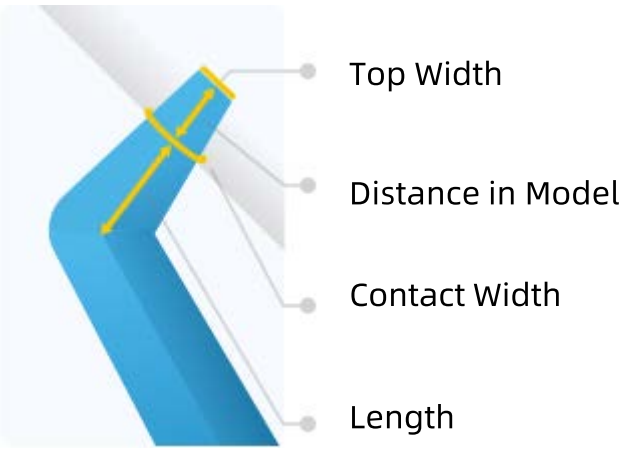
Export the script

2. Basic Settings

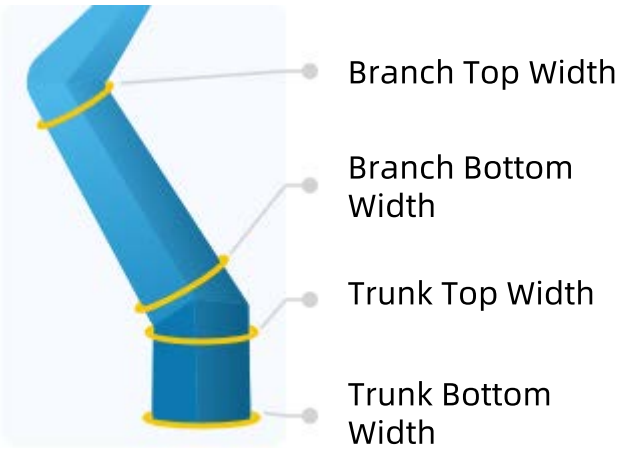
Support Angle: The larger the support angle, the more the supports.

Anchor Distance: The distance between supports. The shorter the distance, the more the supports.

Contact



Bar



Top Width: The top width of the support bar inserting into the upper surface of the model.

Distance in Model: The length of the support bar inserting into the lower surface of the model.

Contact Width: The width of the contact point that touches the lower surface of the model. The larger the width, the larger the contact area between the support and the model.

Length: The distance between the lower surface of the model and the contact point.

Branch Top Width: The width of where branch contacts the model.

Branch Bottom Width: The width of where branch contacts the trunk.

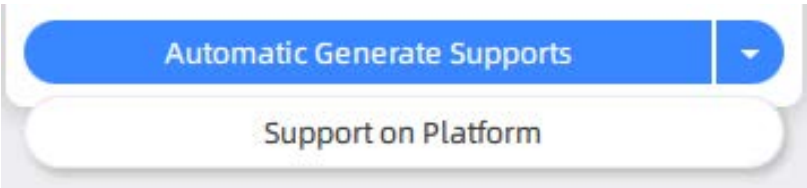
Trunk Top Width: The width of where trunk contacts all branches.

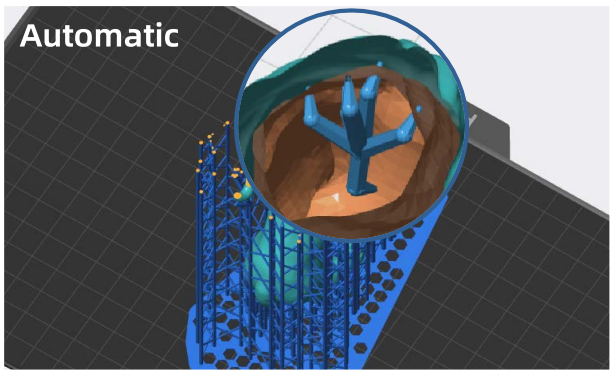
Trunk Bottom Width: The width of where trunk contacts platform or base.

Z Lift Height: Raise the models before adding supports to avoid bottom of the models being destroyed during printing.

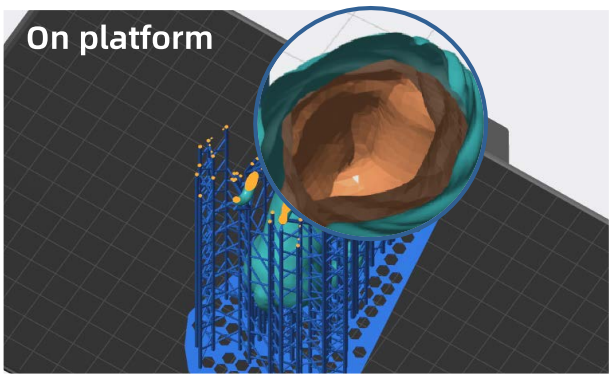
3. Generate Supports

- **Automatic Supports**



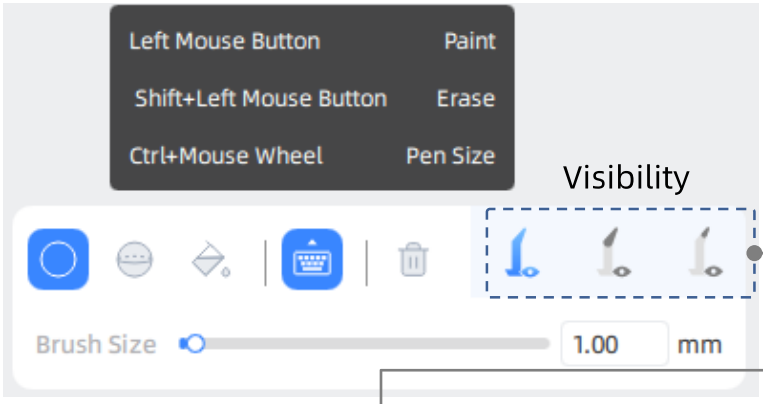


add between platform and model, between points on model

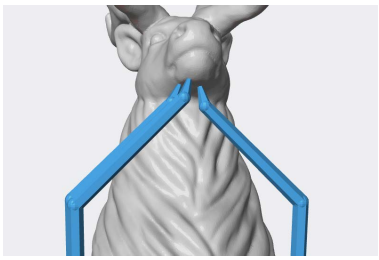


add between platform and model only

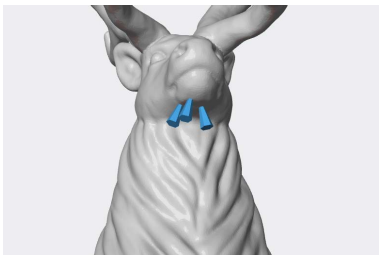
- **Supports Painting**



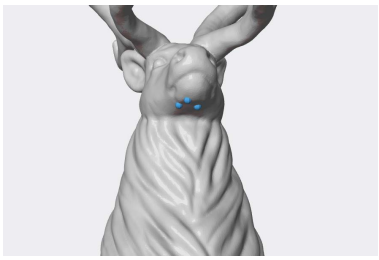
All

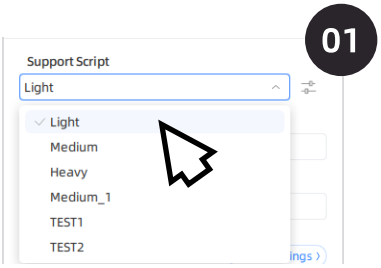


Connection

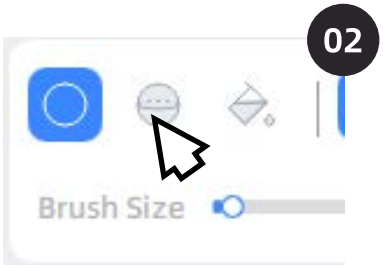


Anchor

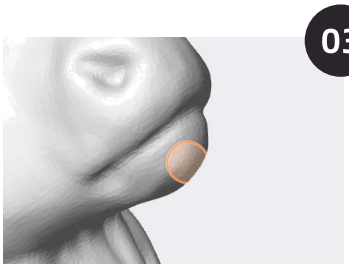




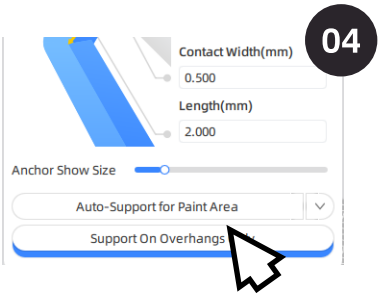
Choose the support script

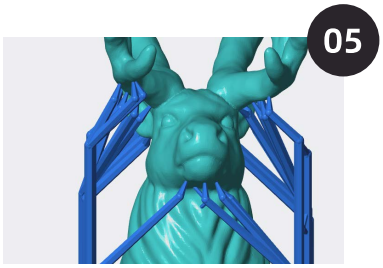


Choose brush shape



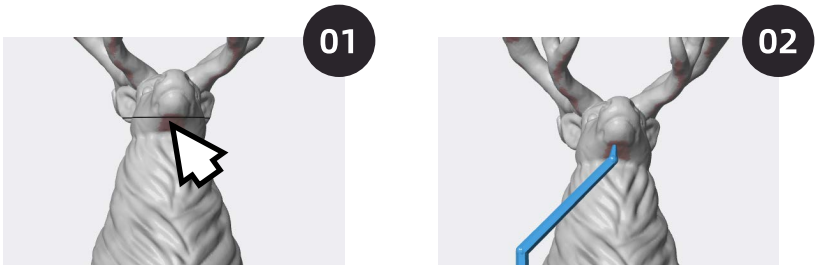
Paint the areas that need supports



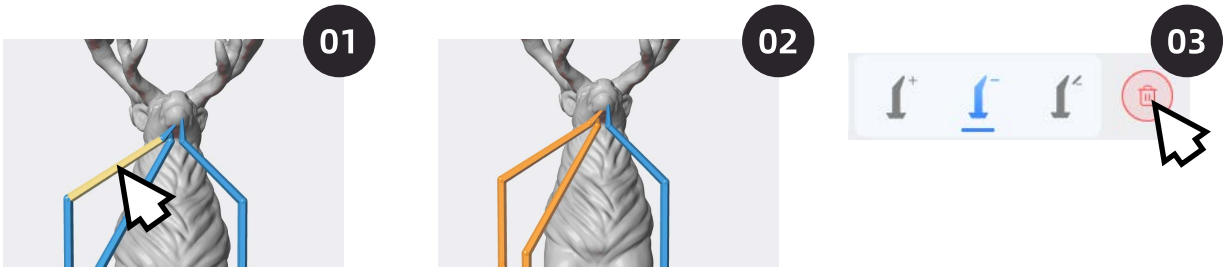


- **Manual Supports**

Add:



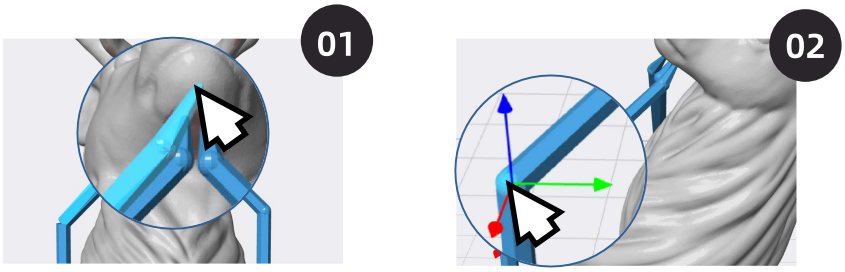
Remove:



Edit:

① Select a anchor and then drag to move it.

② Select a contact point between the top and branch or between the branch and trunk, then edit its XYZ axis position.

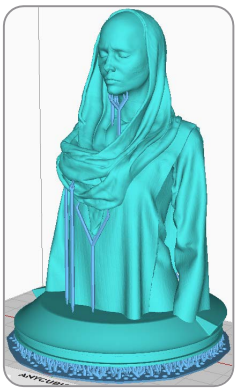


Support type:

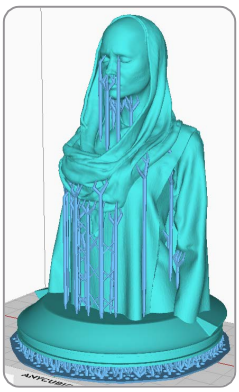
4. Support Adding Skills

- **Properly increase support angle or decrease anchor distance**

When browsing on the model, it can be found that the model still has some weak points that have not been added supports properly. If you increase support angle or decrease anchor distance, more supports can be added to some of the weak points.



Support angle 30°



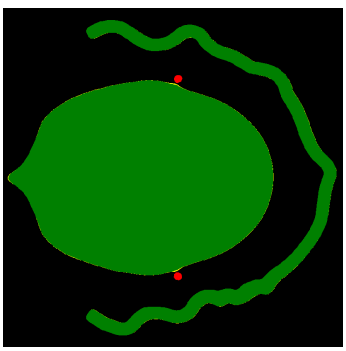
Support angle 50°

- **Add manual support after automatic support**

Manually add support to some of the weak points.

- **Check Land**

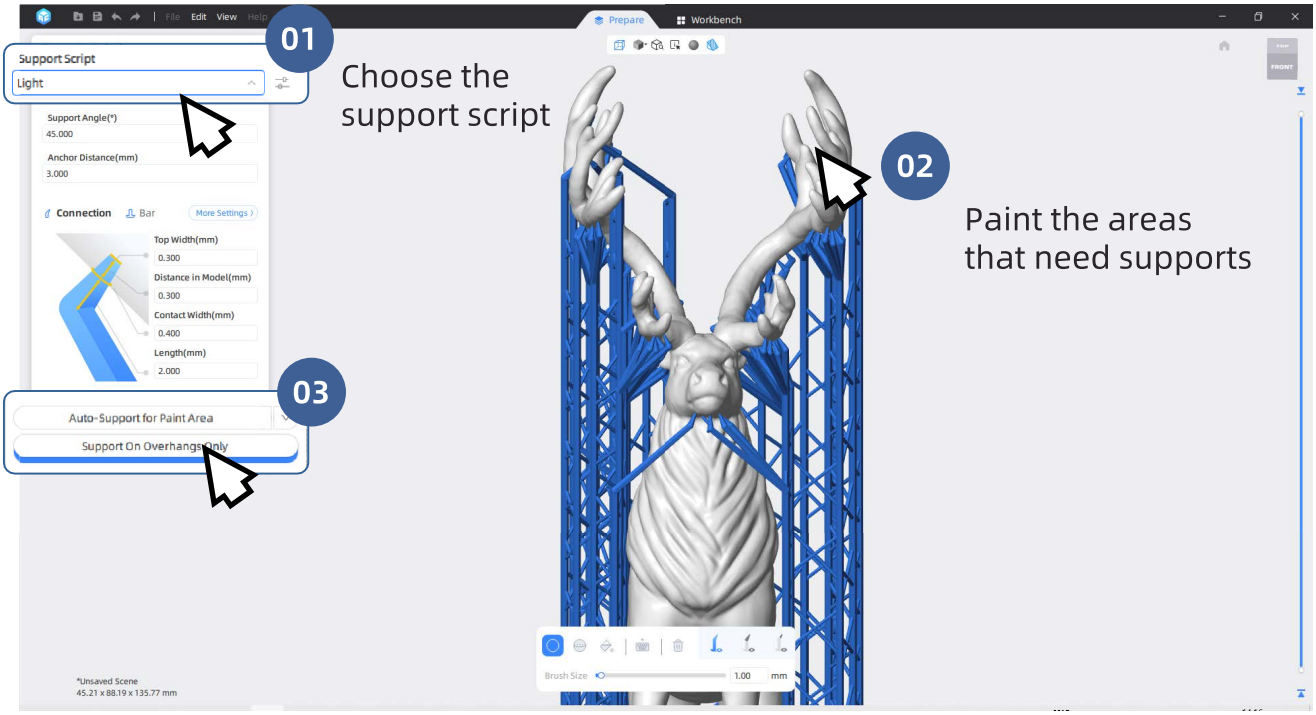
Click “Check Islands” in slice file preview interface, then drag slider to check the image of every layer. The green part means there is support below; the yellow part is connected to other parts but may be suspended and may need supports; the red part is completely suspended and must be added supports.

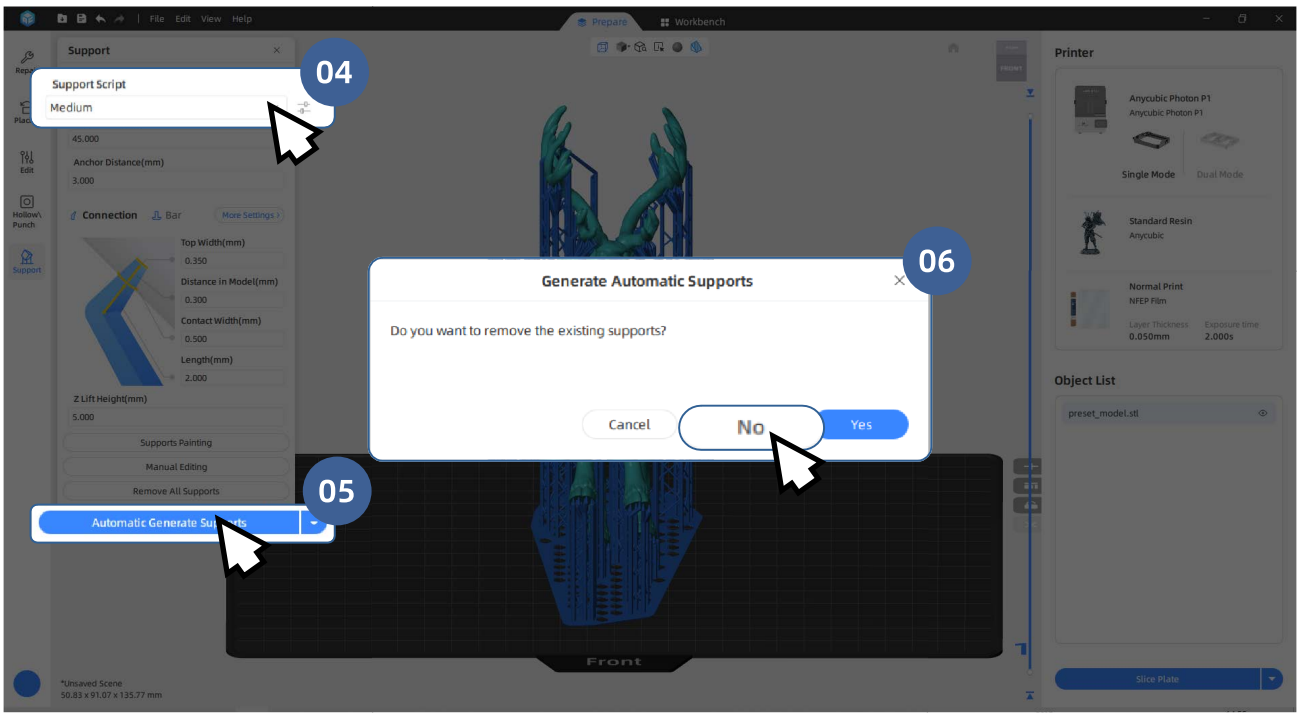


The author of the model: Fabio Nishikata

- **Apply different support script in certain areas**

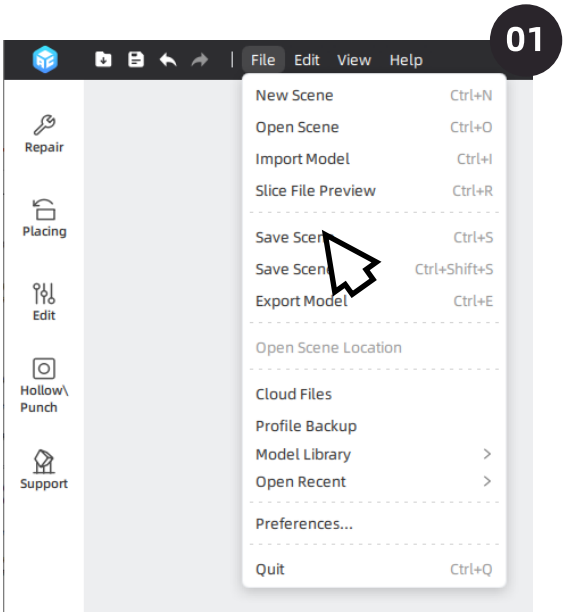
Use supports painting to add finer supports to detailed areas to reduce the risk of damaging surface details during post-processing. Apply middle supports to the other areas to ensure a high print success rate.

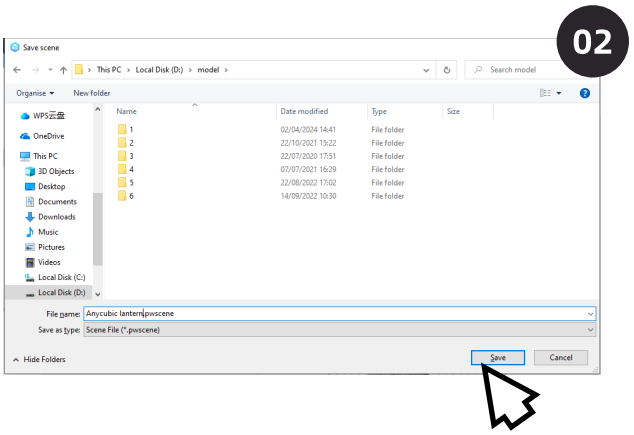




5. Save Scene File (Optional)

Save the current scene including model, settings, supports and so on for easy reuse and editing.





6. More support settings

- **Number of Anchors**

Support Angle: The angle between the support surface and the horizontal plane. The larger the critical angle, the greater the area that needs to be supported.

Anchor Distance: The distance between the support anchor points.

Border Offset: The distance from the support to the support area boundary.

Reinforce Lowest Points: When active, the anchor distance near the lowest point can be defined.

Lowest Anchor Distance: The minimum distance between the anchor points on the overhang.

Reinforce Radius: The height of the lowest part which is reinforced.

Enable Support in Shell: Add support to the hollowed part.

- **Connection**

Distance in Model: The length of the support bar inserting into the lower surface of the model.

Top Width: The top width of the support bar inserting into the upper surface of the model.

Contact Width: The width of the contact point that touches the lower surface of the model. The larger the width, the larger the contact area between the support and the model.

Ball Contact: To set the contact shape as ball.

Ball Diameter: The larger the diameter, the larger the contact area.

Length: The distance between the lower surface of the model and the contact point.

Tip Type: Set the tip being perpendicular to contact area/horizontal plane.

Min Angle to Surface: The minimum angle of the bottom of support to the surface of model.

Max Angle to Vertical: The maximum angle of the bottom of support to the vertical direction.

Filter Supports

Max Small Pillar Height: Support length less than this value will be changed to small pillar.

Small Pillar Width: The width of the small pillar.

Remove Short Supports: When it is enabled, the supports shorter than a set length cannot be generated on the model.

Short Support Length: The support that length less than this value and bottom connect to model will be filtered out.

Platform Connection

Height: The height of the support platform.

Radius: The larger the radius, the larger the support platform.

Slope Angle: The slope angle of the edge of the support platform.

- **Bar**

Bar Shape

Polygon Edge Number: The number of the sides for the Polygonal prism.

Max Branch Number: The maximum number of the branches of the bar.

Branch Top Width: The width of where branch contacts the model.

Branch Bottom Width: The width of where branch contacts the trunk.

Trunk Top Width: The width of where trunk contacts all branches.

Trunk Bottom Width: The width of where trunk contacts platform or base.

Branch Min Angle: The minimum angle between the branch and the horizontal direction.

Branch Max Start Height: The maximum distance between the starting point of the branch and the top of the trunk.

Bar Cross Connection

Start Height: Cross is generated from a certain height.

Cross Bar Width: The width of the cross.

Cross Bar Angle: The angle between the cross and horizontal plane.

Interval Height: The distance between the cross.

- **Base**

Add base to increase the adhesion between model and the print platform to reduce the risk of the print failure.

Plate Offset: The minimum distance between the support and edge of the base. The larger the offset, the larger the base.

Plate Height: The thickness of the base.

Slot Angle: The slope angle of the edge of base.

Plate Type:


Rectangle


Convex Hull


Minimum Area


Project Area

Perforation

Hole Radius: The larger the holes, the less the resin needed, the smaller the contact area with the print platform.

Hole Interval: The larger the interval, the less the holes.

Hole edge number: The larger the edge number, the more similar to circle the hole is.

- **Advance**

Support Avoidance Setting

Min Spacing from Model: The minimum spacing allowed between the main column of a support to any point on the model surface.

Max Offset from Contact: The maximum lateral offset allowed between a support's model contact point and its main column's centerline. Supports rooted on the print platform would only generate when the lateral offset is within limit set by this parameter.

Auto-reduction Scale: The width of the support will be reduced automatically according to this ratio when it is detected that the width of the original supports is too large and causes it interferes with the model.
Min Contact Width: The minimum contact width, when the support width is reduced, the contact width will not be less than this value.

Thin-wall Detection

Scale of Distance in Thin-wall: The scale of the depth of support inserted into the thin wall to the wall thickness.
Min Distance in Thin-wall: The minimum depth of support inserted into the thin wall.

Simplified Support

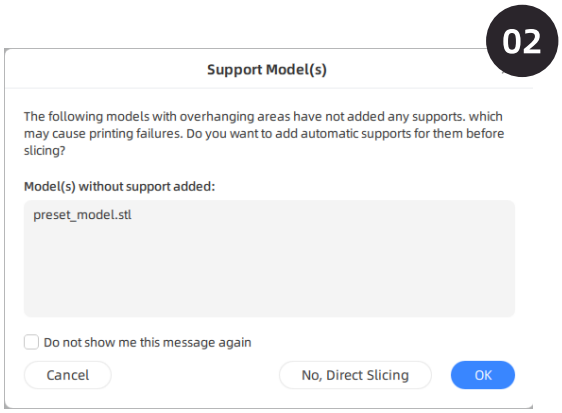
Overhang Self-support Offset: The distance at which the overhang will be offset inward when the model itself can provide support for its overhang.
Min Support Line Length: The minimum support line length, support line length less than this value will be filtered out.
Min Overhang Area: The minimum overhang area, overhang Area less than this value will be filtered out.
Min Horizontal Surface: The minimum horizontal surface, horizontal surface area less than this value will be filtered out.

Reinforce Single Support: When active, the support that without cross bar connection will be automatically reinforced.
Reinforce Support Height: Only support that without cross bar connection and height more than this value will be reinforced.
Spacing from Reinforce Support: The distance between the reinforced support and the original support.

1. Slice

When the setting is finished, click Slice. Anycubic 3D printer can only read its corresponding sliced file formats, please choose the machine type you use at Machine Settings to avoid print failure.

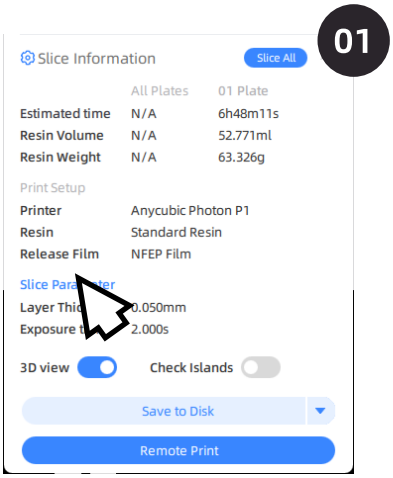
It is suggested to add supports if it prompts that the model with some overhanging areas is needed supports. To cancel the automatic check for supports, enter into file-preferences-slicing and uncheck “show support model prompt dialog when slicing” .



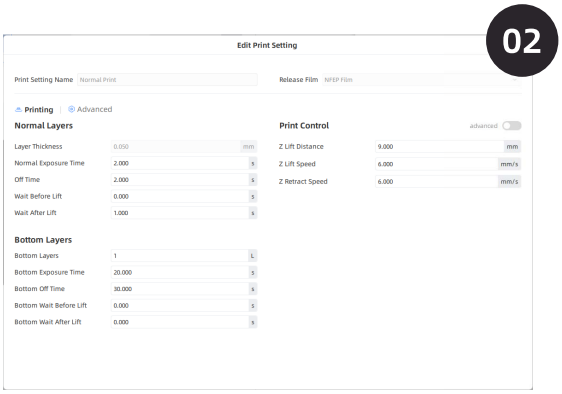
Check the supports

2. Preview

In the slice file view interface, you can preview slice settings and adjust the slice settings.



Click to enter in the slice setting interface



Adjust the parameters then click OK to save

3. Export

- **Remote Print:** Send the sliced file to the printer and start printing. The print job can be remotely controlled by Anycubic Photon Workshop or App.
- **Save Sliced File**

  Save to Disk: Save the sliced file to PC.

  Save to Cloud: Save the sliced file to Anycubic Cloud.