ideaMaker Manual

RAISE3D

www.raise3d.com

1 Using ideaMaker

1.1 What is ideaMaker?

ideaMaker is a slicing software which prepares 3D models for printing and turns them into .gcode file for your Raise3D printer.

1.2 Where to download ideaMaker?

ideaMaker is available in the USB storage included in the accessory box with your printer. Or you can get the latest version on our website at:http://www.raise3d.com/pages/download

2 Install ideaMaker

1. Open the installer and choose your language preference. Then click Next to move on to the next menu. If you are using ideaMaker before your installation, you need to close all the process of ideaMaker. So that the new version can be installed without any error. Please check your task manager to make sure that there is no ideaMaker process before installation.

Figure 2.1: Choose a language for installing ideaMaker.

2. Select a path to install ideaMaker and then click ***Next*** to move on to the next menu.

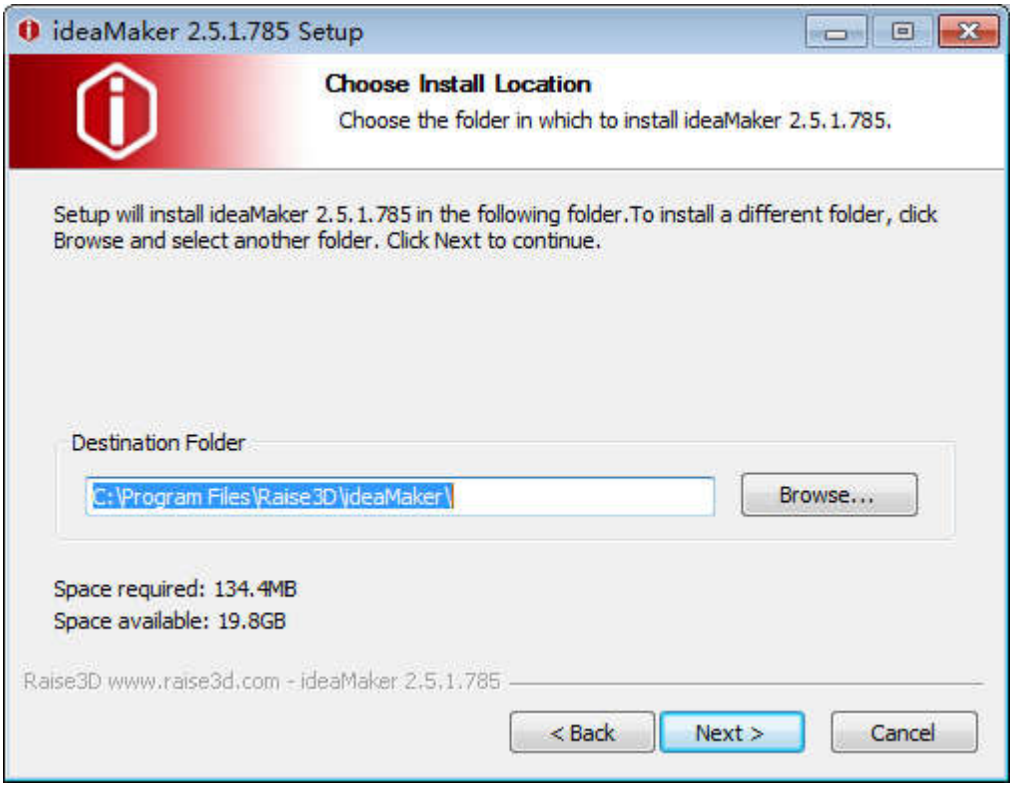


Figure 2.2: Choose a path to install your ideaMaker.

3. Follow the instruction and click ***Install***. The ***Printer Driver*** function is used for F-series printers. If you do not have one, you can uncheck the component.

The C++ is auto-checked by installation software. If you have installed that in your computer before, you don't need to check it again.

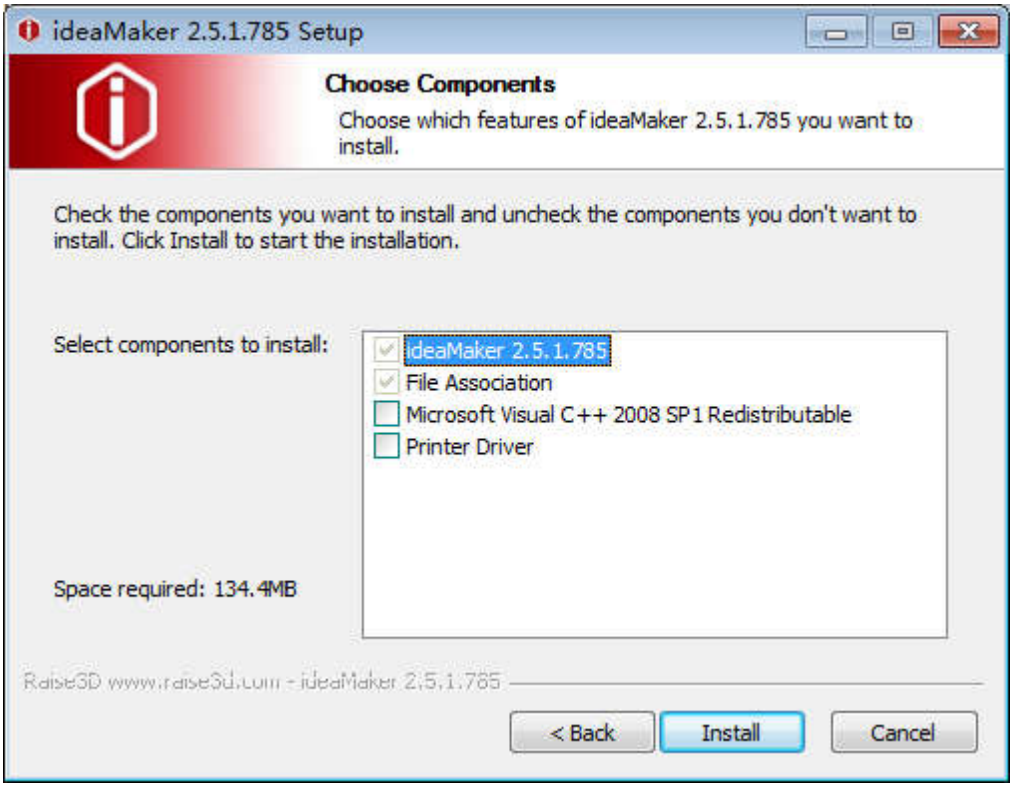


Figure 2.3: You don't need to choose Printer Driver if your printer is N-series Printer.

4. After the installation finished, click ***Next*** to move on the next step.

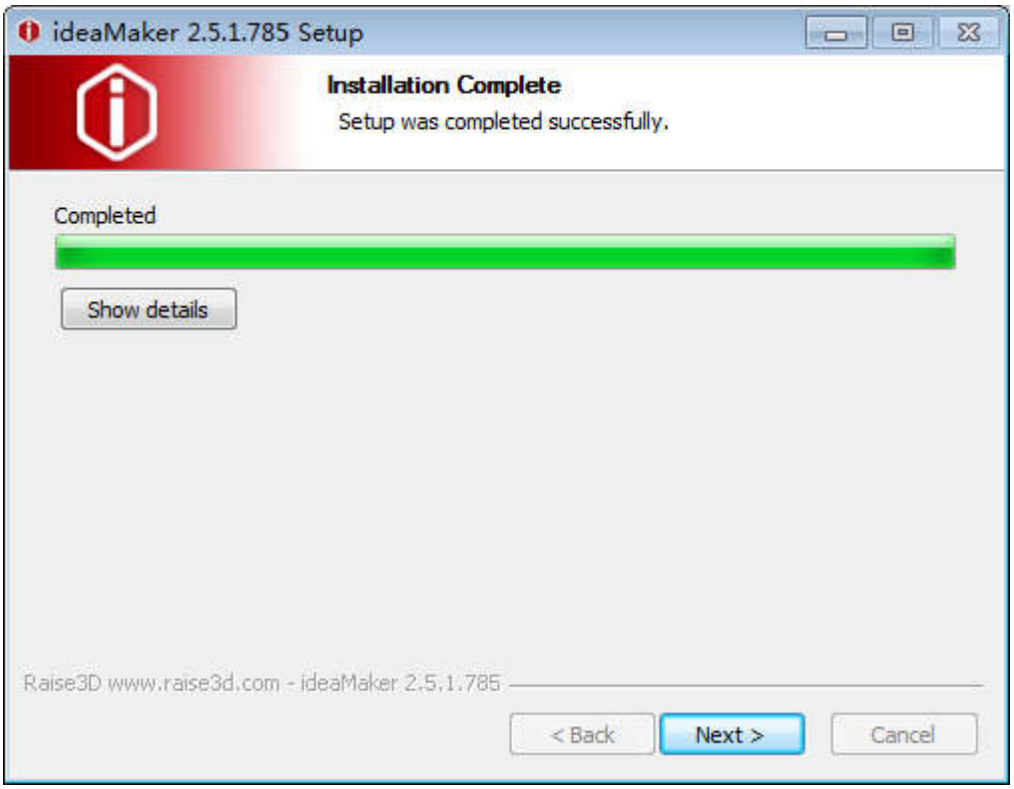


Figure 2.4: Press "Next" to complete the installation.

5. Click ***Finish*** and start your first print.

Figure 2.5: Complete the installation.

3 Let’s Print!

As you have finished the installation, now you can start your first print. Here are some basic steps.

1. Click the button**"+"**to import a .stl model. Click the ***Repair*** button to execute an auto-repair of your model if the box at bottom-right corner shows error warnings.

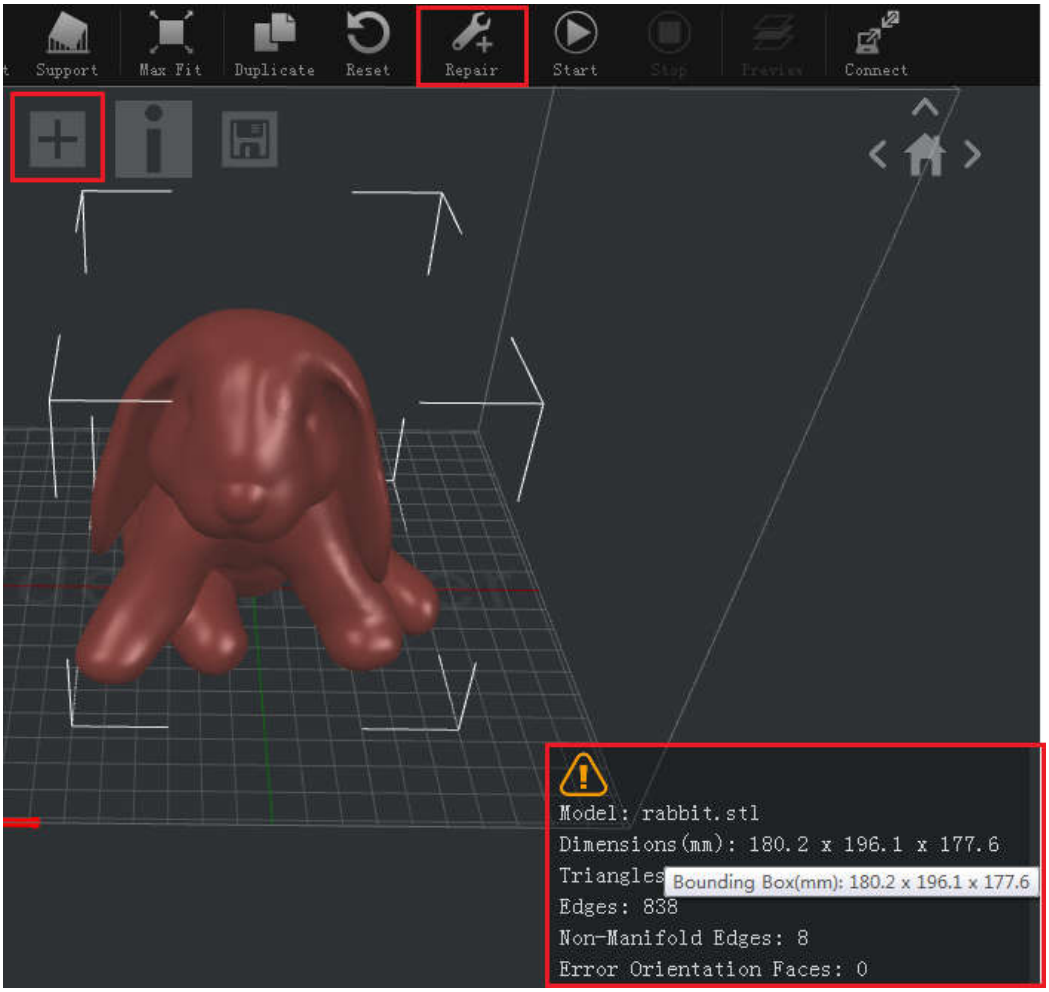


Figure 3.1: Press the "+" icon to import a .stl model. If it doesn’t display a green tick below, press the "Repair" button to execute an auto-repair.

2. Click the button ***"I"*** to start slicing the model.

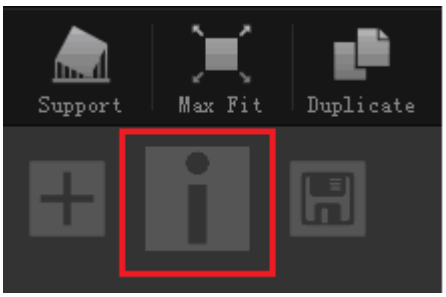


Figure 3.2: Press the "I" icon to start a slice.

3. Select your printer type and filament type. If you are using V2 hot end, please choose both printer and filament as V2. If not, both without V2.

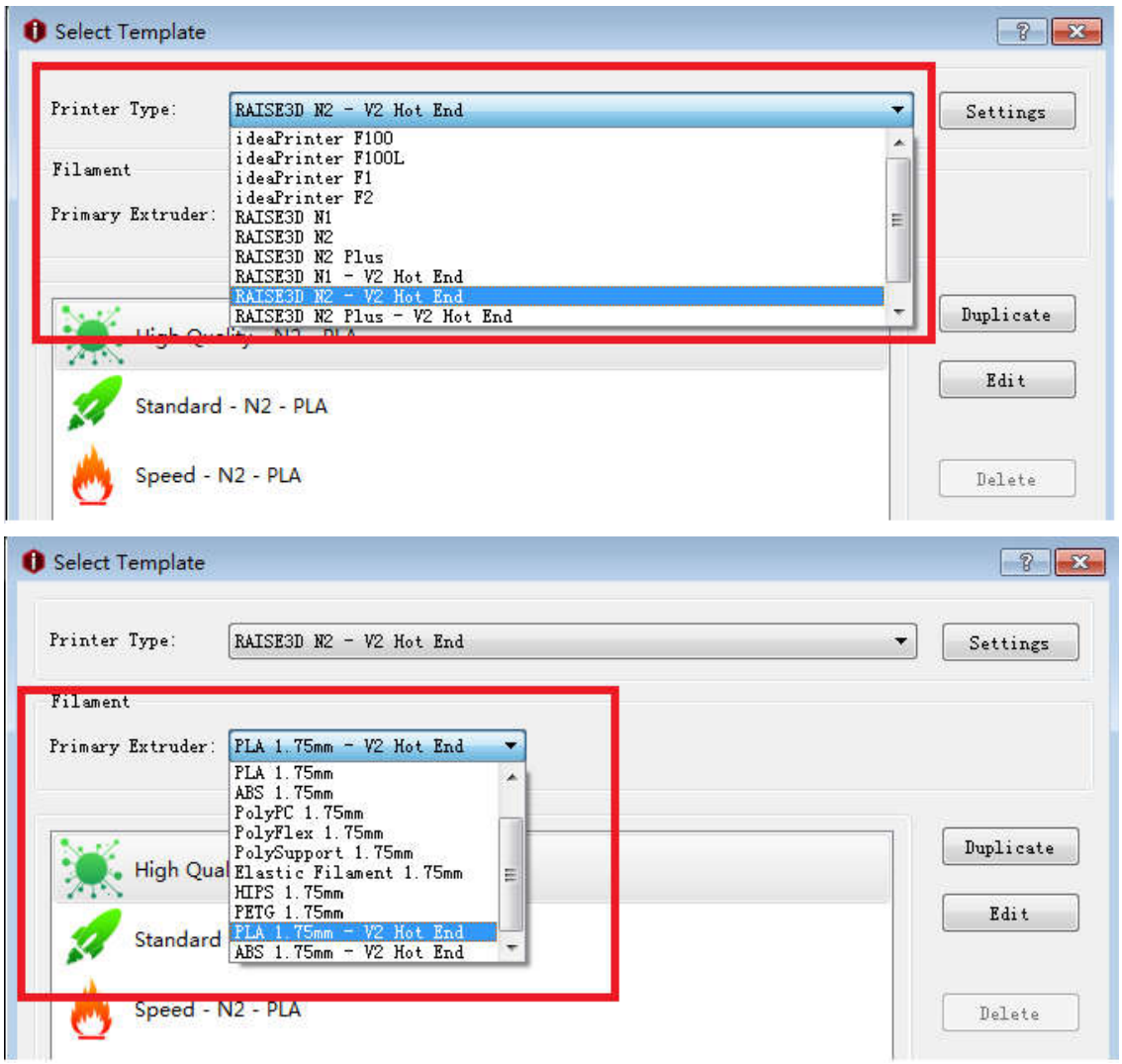


Figure 3.3: Choose the printer and filament you want to use.

4. Select a slicing template or create a new template yourself by duplicating one of the three existing templates.

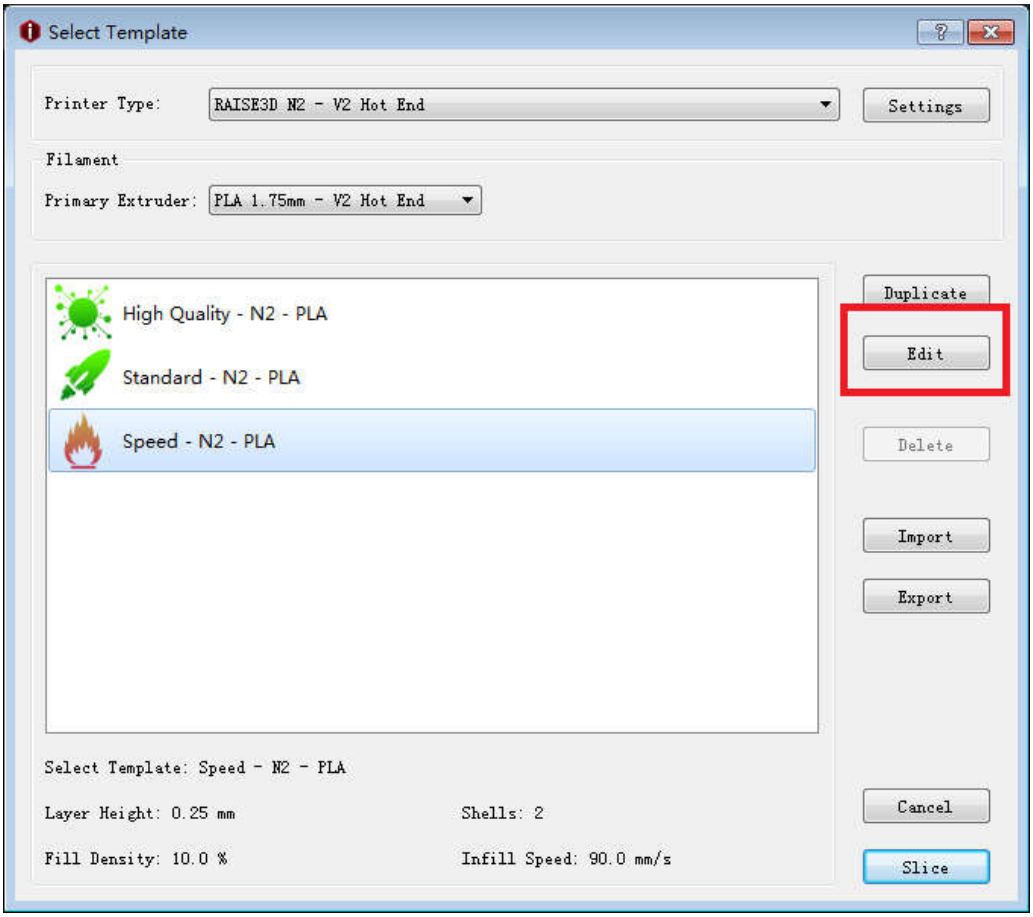


Figure 3.4: Choose a template to start with. Click "Edit" to adjust the settings.

5. Click ***Edit***button or double click the template (or not if you duplicate a new template). Select a ***Raft***type and ***Support*** type, then click***Save and Close*** button. (You can edit other parameters in***Advance*** as well if you want. We will take about that in the later chapters.)

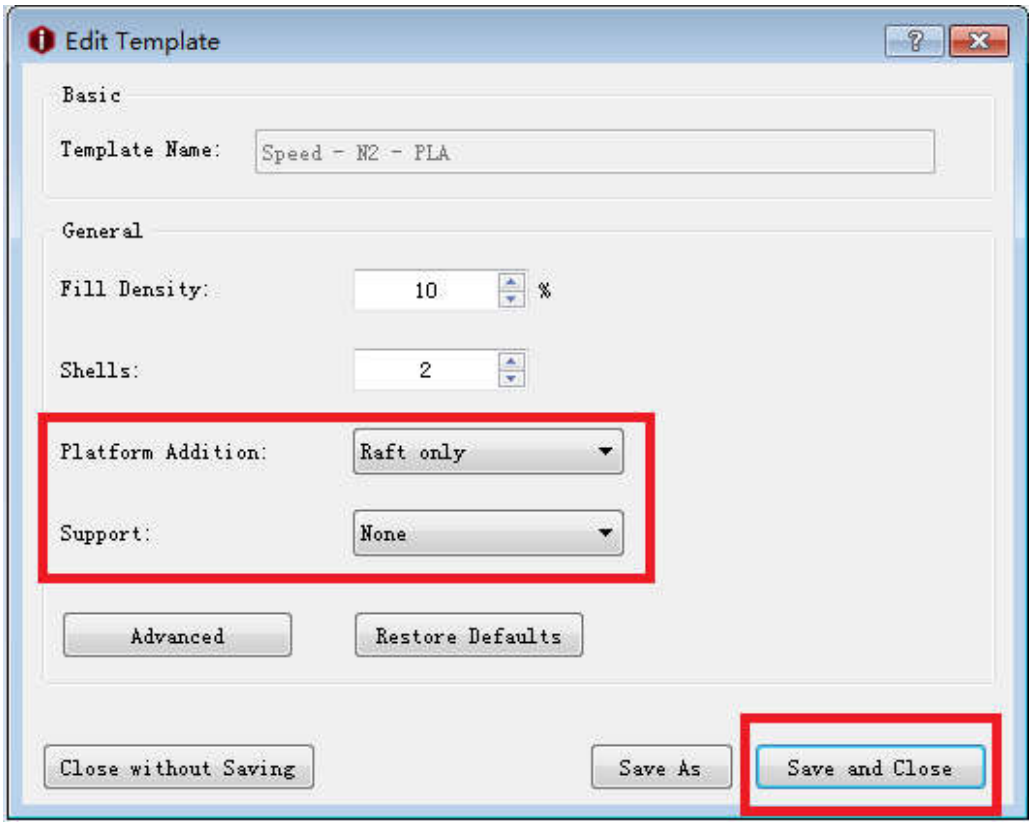


Figure 3.5: Select a suitable "Raft" and"Support" type for your model.

6. Click ***Slice*** button.

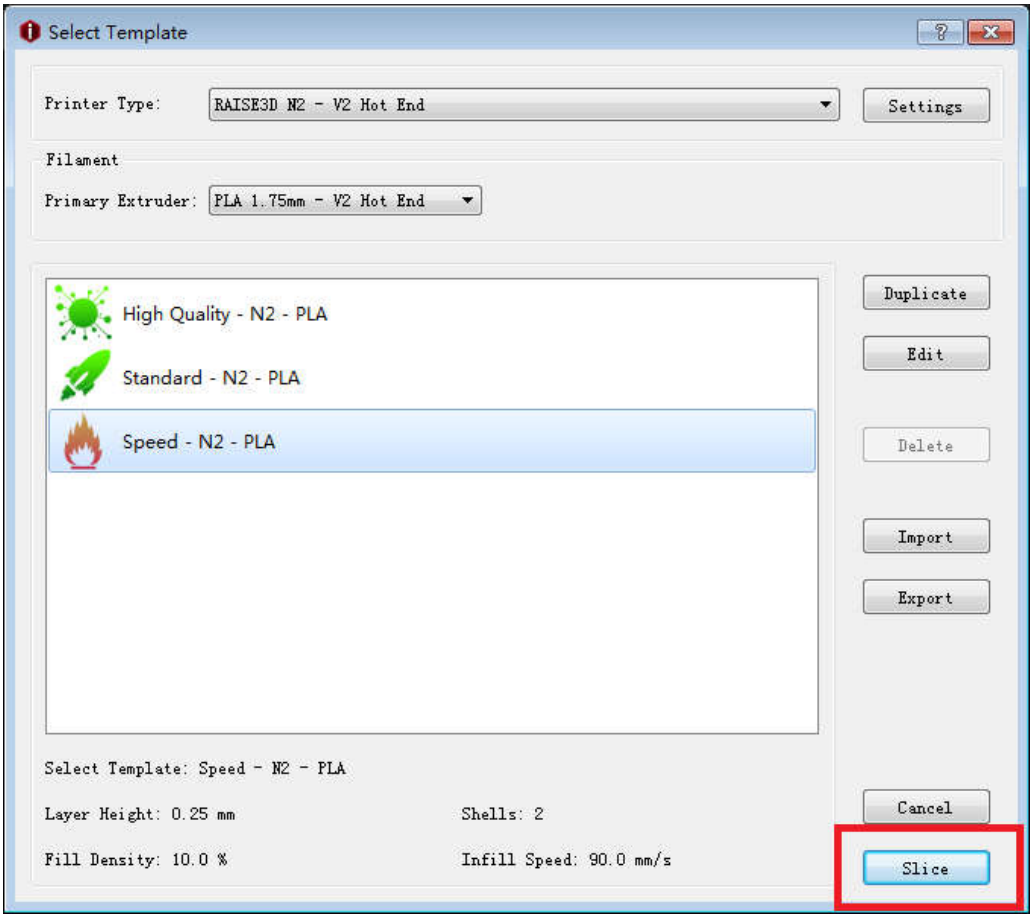


Figure 3.6: Click "Slice" to start slicing.

7. After completing the slicing, ideaMaker will advise the estimated data for your reference.

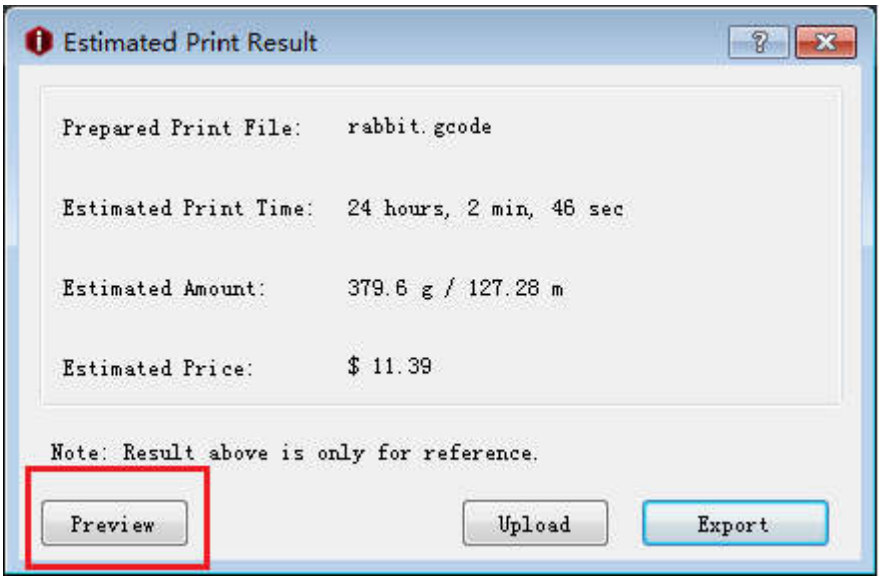


Figure 3.7: Here you can check estimated print time and amount of filament needed.

8. Click ***Preview*** button so that you can check the sliced model layer by layer.

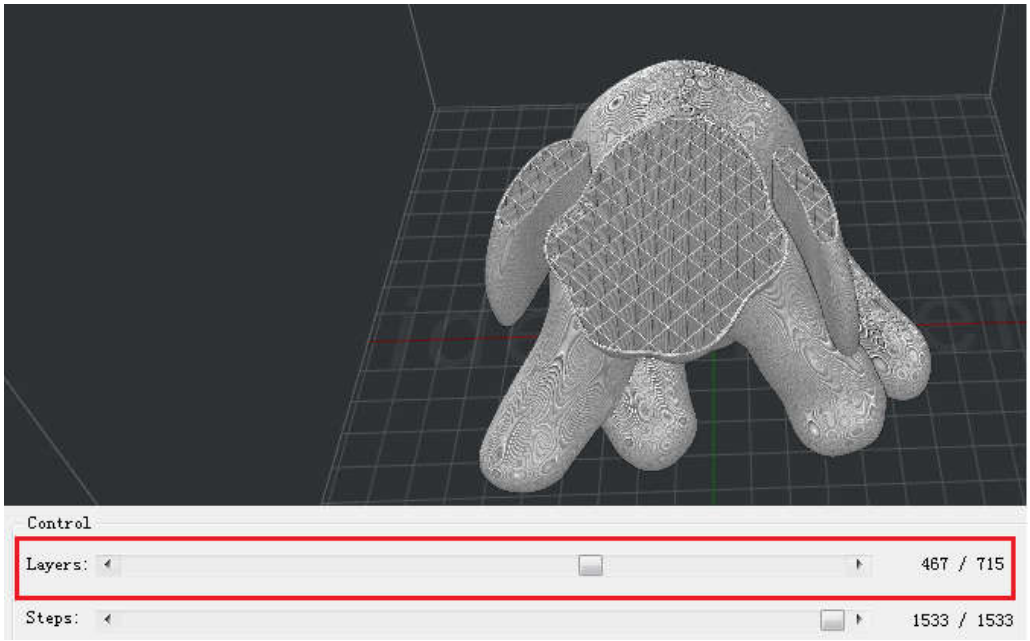


Figure 3.8: Preview how the model will be printed out.

In this page you can check the retraction and travel move of the nozzle by selecting the corresponding options.

The blue lines refer to the moving paths of nozzle. The red marks refer to the retraction points.

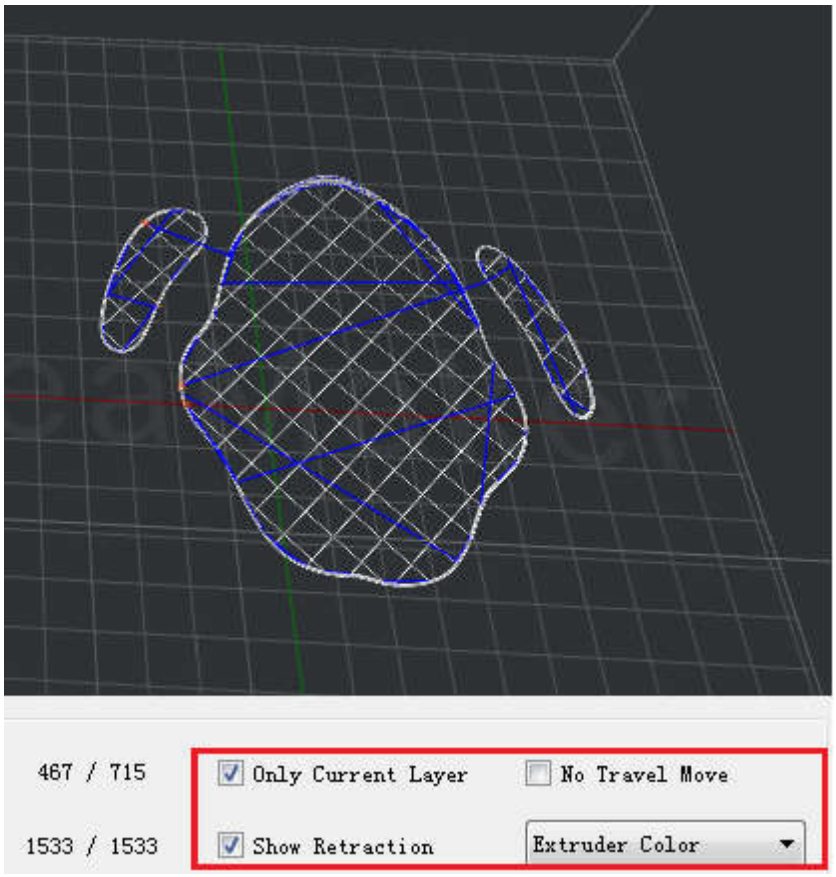


Figure 3.9: Check the retraction points with "Show Retraction". Check one layer with "Only Current Layer".

It can also show different structures in different colors by selecting ***Structure*** in option box. The cyan parts refer to support and Raft. The red part refers to outer shell. The green parts refer to the inner shells. The yellow parts refer to infill. The blue lines refer to the moving paths of nozzle.

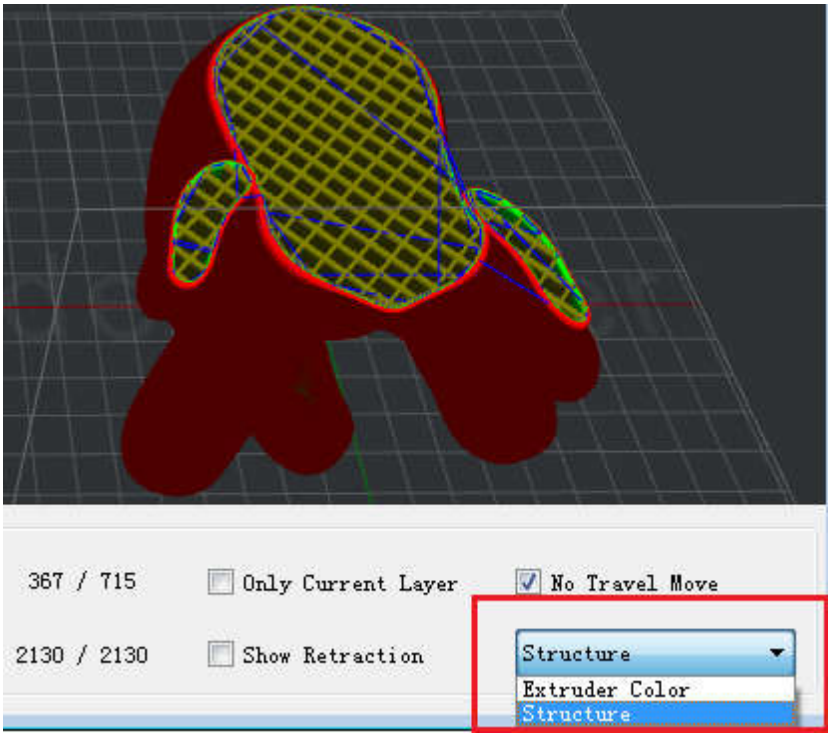


Figure 3.10: Display different structures with different colors.

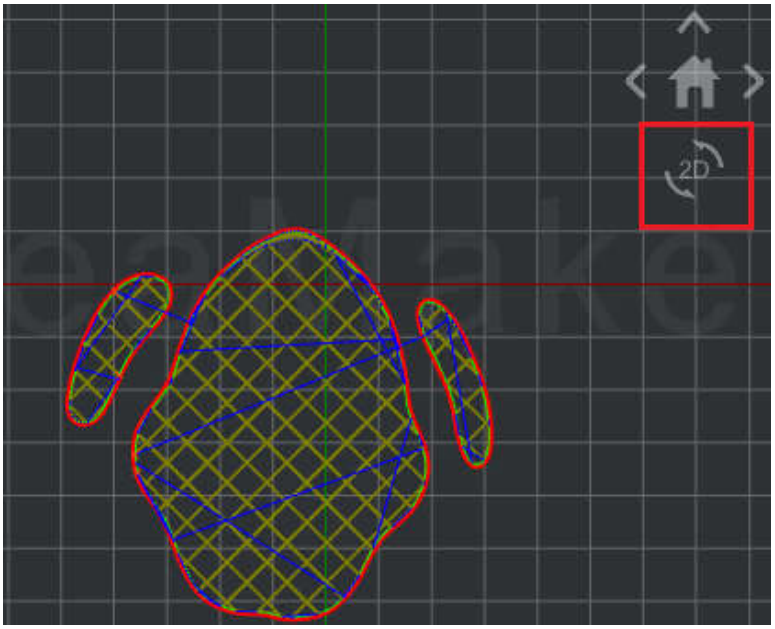


Figure 3.11: Enable "2D" to check the current layer with orthographic view.

9. Close the preview dialogue box after the confirmation.

Now you will have two options to load the files to the printer.

Option1: Export to USB storage or SD card

Export the sliced files to USB storage or SD card.

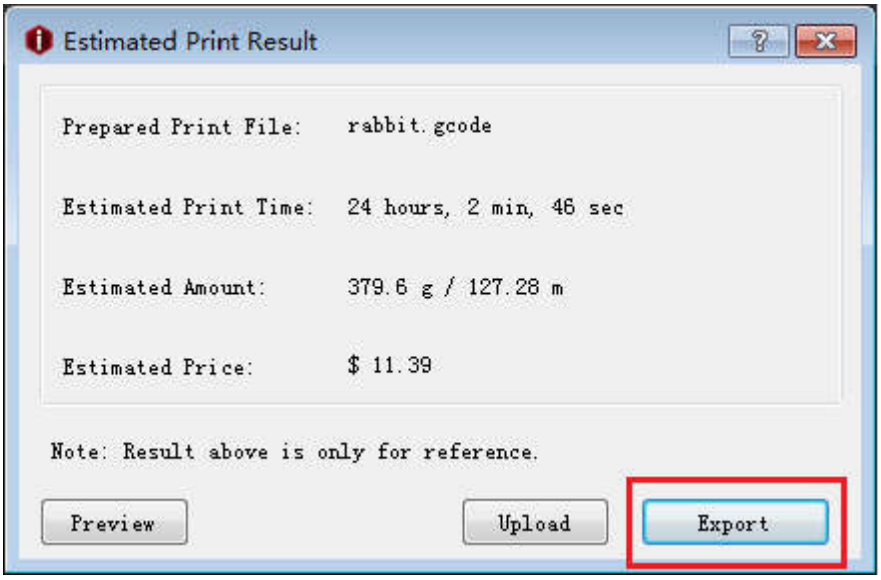


Figure 3.12: Click "Export" in "Estimated Print Result" window to save the sliced files.

1. If you want to save the sliced files, you can export the files directly to your USB storage or to folders in your computer. Then copy the sliced files (.gcode file and .data file) to USB storage. It is important to copy both files.

Figure 3.13: .gcode file and .data file are both necessary for N-series printers.

2. Insert USB storage to your printer and select the file to start your first print.

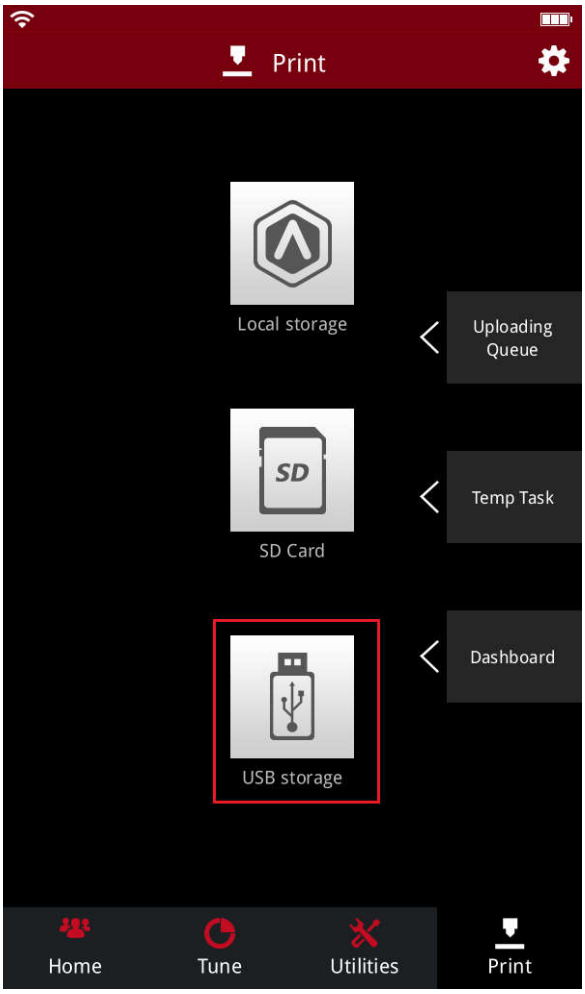


Figure 3.14: If you are using USB driver, press the "USB storage" to check the files inside. If you are using SD card, press "SD Card" to check the files inside.

Option2: Upload the sliced file via WLAN.

1. First you need to make sure the printer and your computer are connected to the same network.

For WLAN connection, there is a little gear button at the top-right corner of the screen. Press the little gear button to go to the setting page.

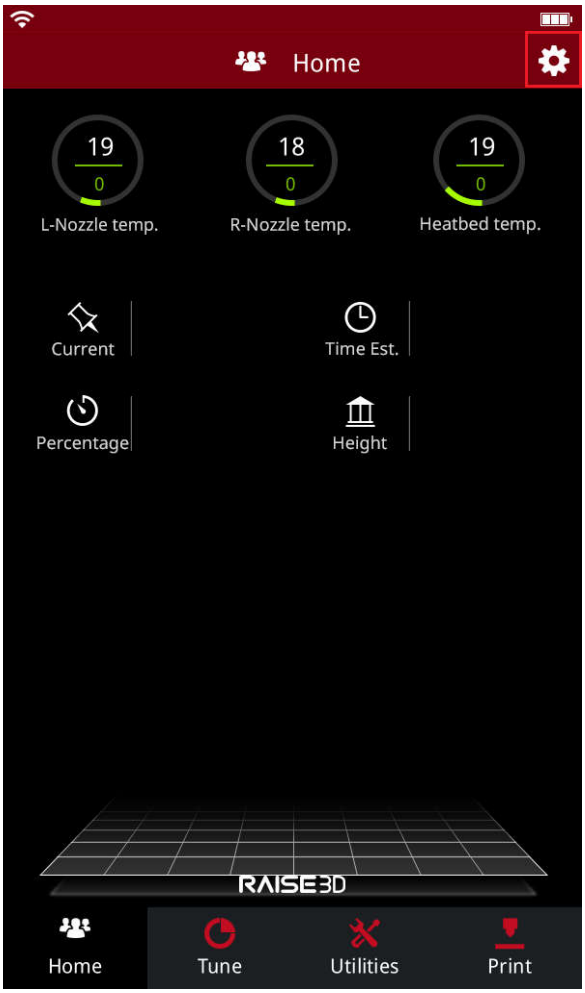


Figure 3.15: Press the gear button to enter the setting page.

2. Choose ***WLAN*** tab. Enable ***WLAN*** and choose your network from the list then input the password.

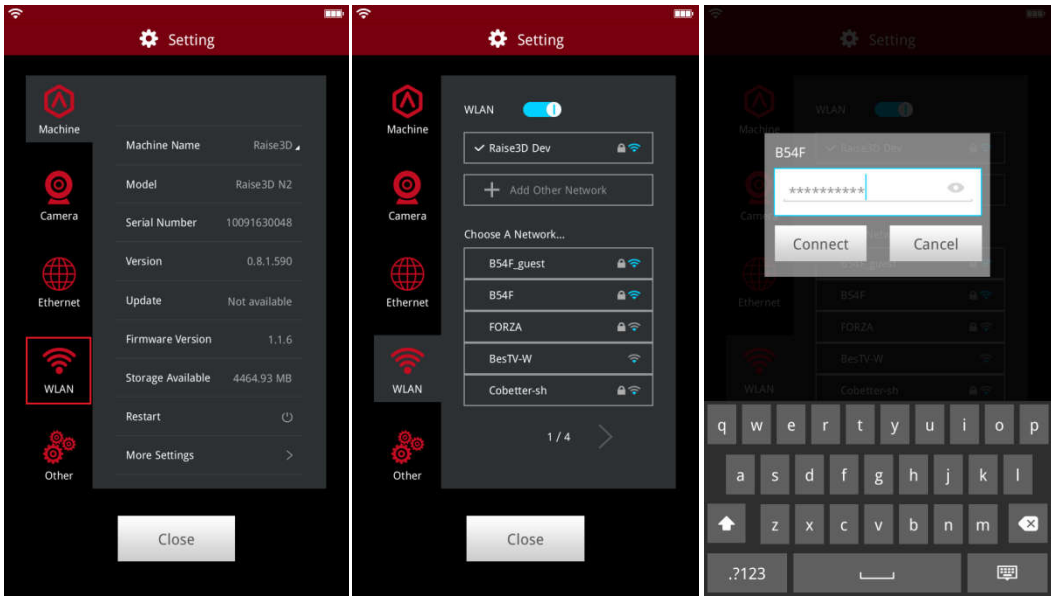
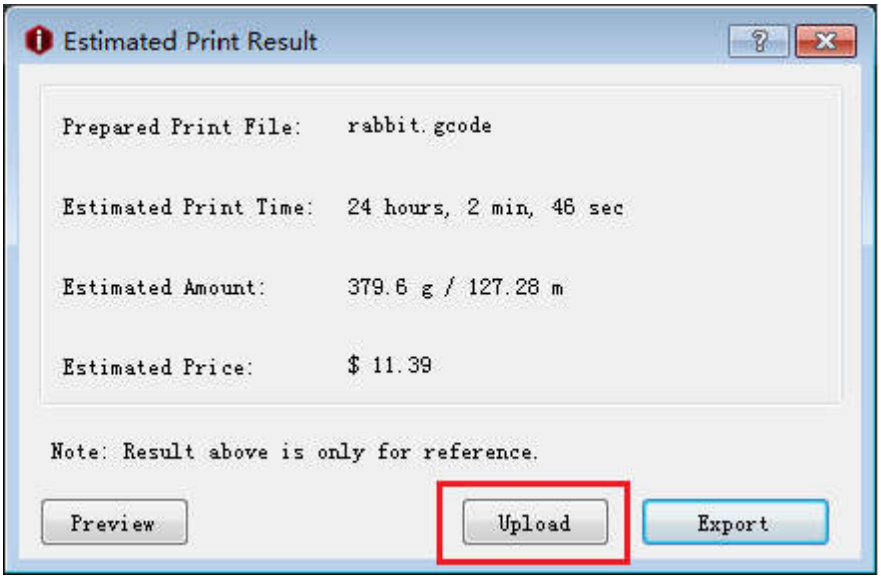


Figure 3.16: Connect the printer with your WLAN.

3. After connecting WLAN for your printer, click***Upload*** and then you will see the ***Select Printer*** page. You can choose the printer you want to print with here.



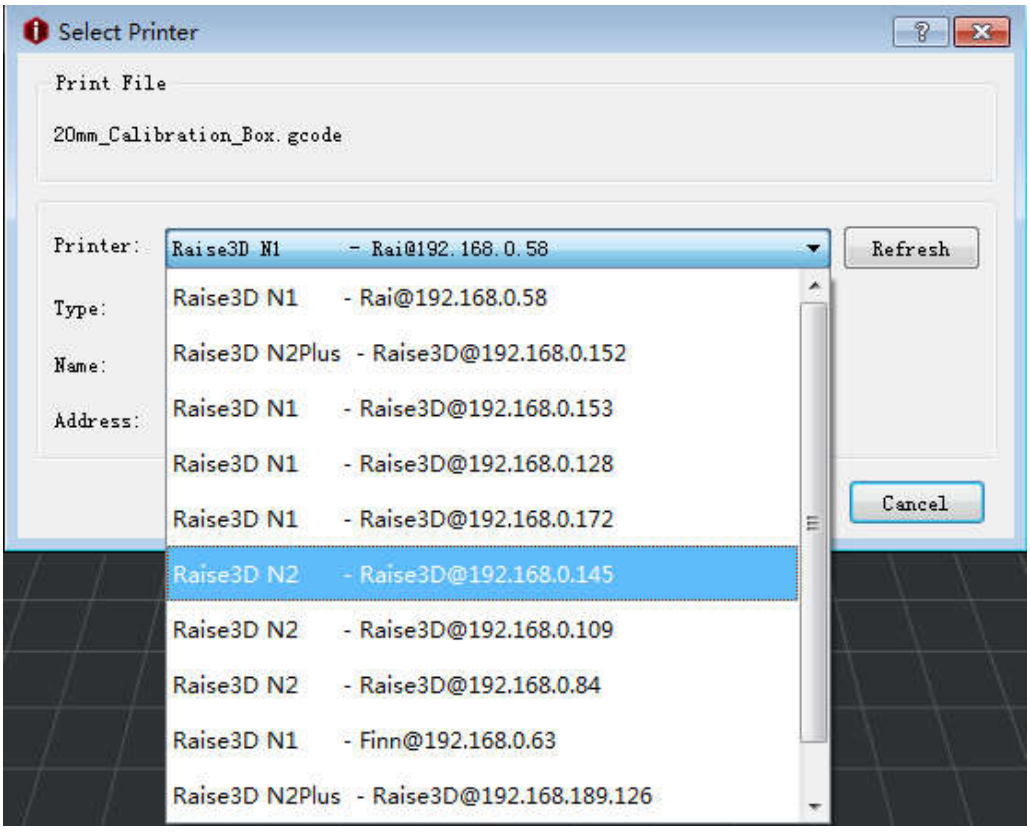


Figure 3.17: Click "Upload" in "Estimated Print Result" page and choose a printer you want to use from the drop list.

4. After you clicking ***Upload***, the ***Uploading Queue*** will appear on the left side of you screen. You can check the loading progress here.

Figure 3.18: Check the uploading progress at the left side of ideaMaker.

5. After the upload is completed, you can start a print. Choose the print from the touchscreen on the printer. The WLAN-uploaded files are in ***Local storage***.

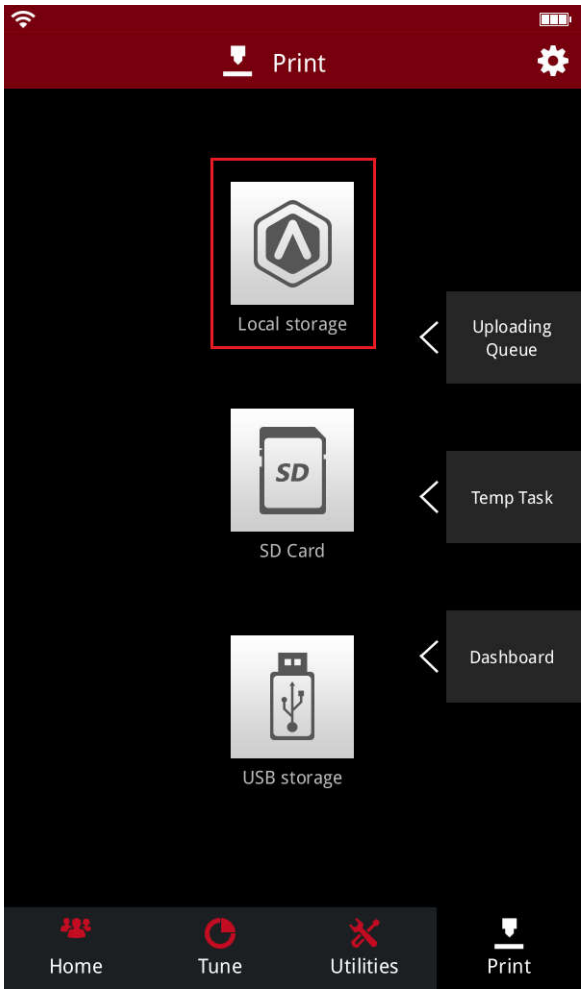


Figure 3.19: Uploaded files will be inside "Local Storage".

Or remote connect your printer through ideaMaker. Choose ***Printer*** ->***Connect To Printer(Raise3D N-series).***

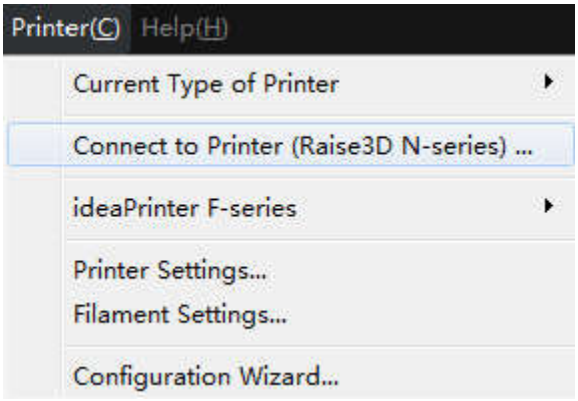


Figure 3.20: Choose Printer ->Connect To Printer(Raise3D N-series).

6. Choose the printer you want to connect with in this page.

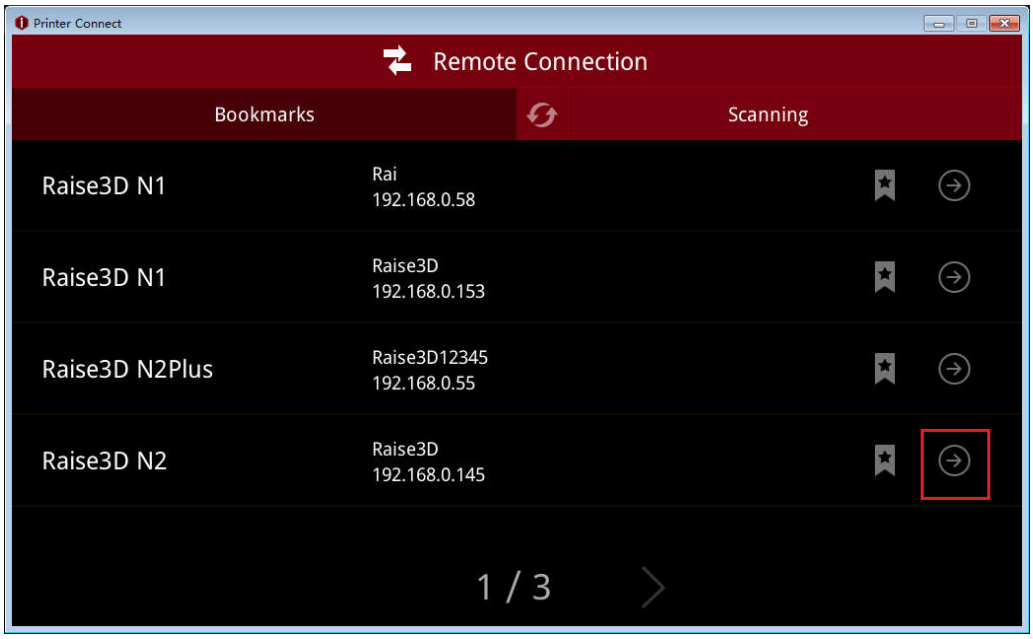


Figure 3.21: Choose the printer you want to connect in "Scanning" list. Click the arrow icon to enter.

7. The connecting page.

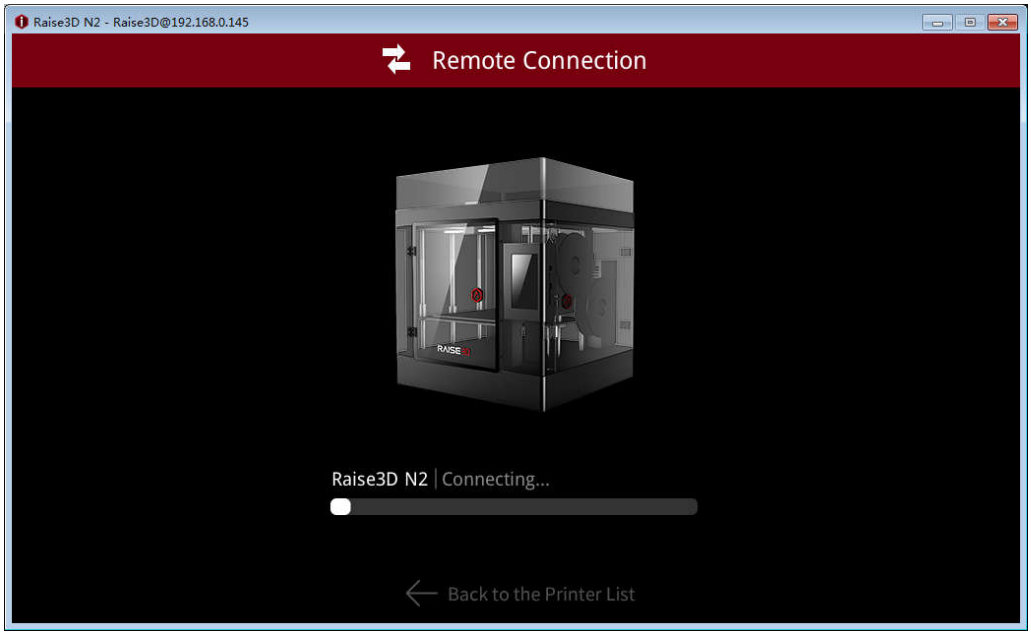


Figure 3.22: Wait for connecting.

8. Now you can control your printer remotely. ideaMaker has the same operation interface asthe touch screen on your printer. You can control the printer directly from here as well. The uploaded files are in local storage.

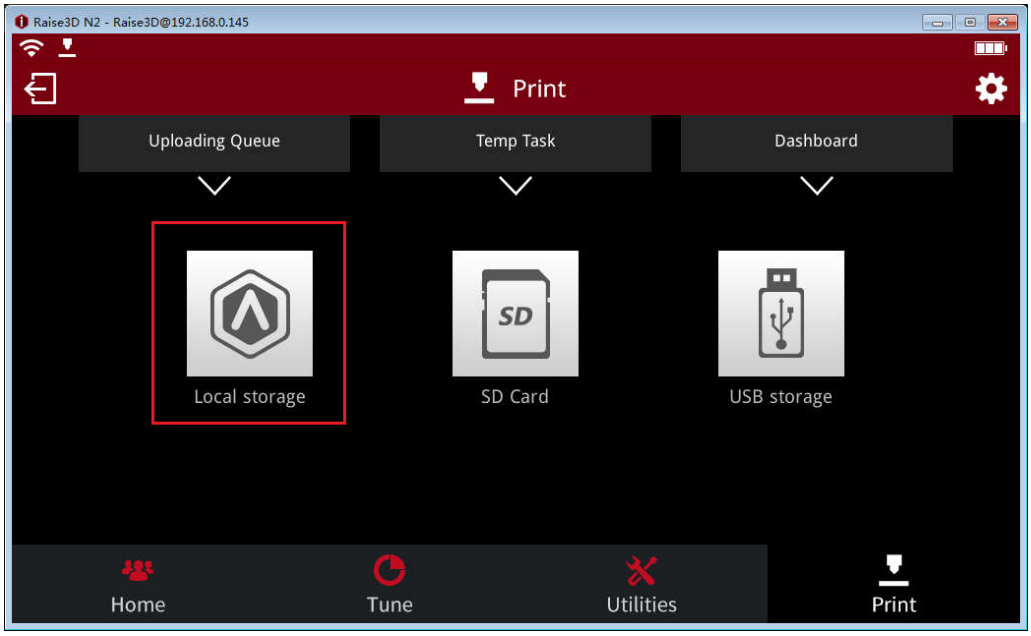
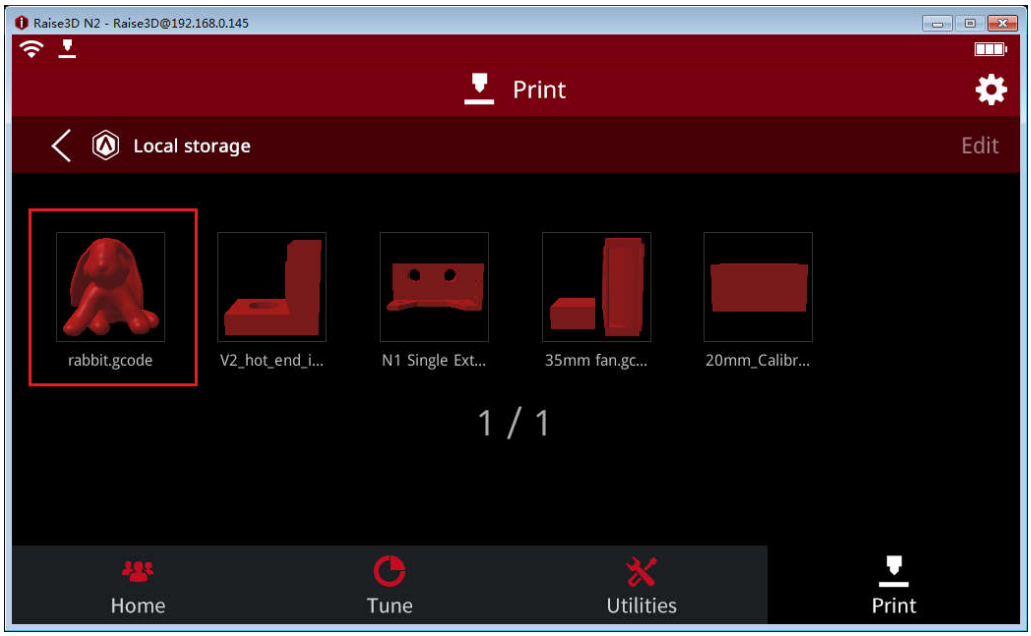


Figure 3.23: Click "Local storage" to check the uploaded files.

9. Choose the file you want to print and press ***Print*** button to start it.



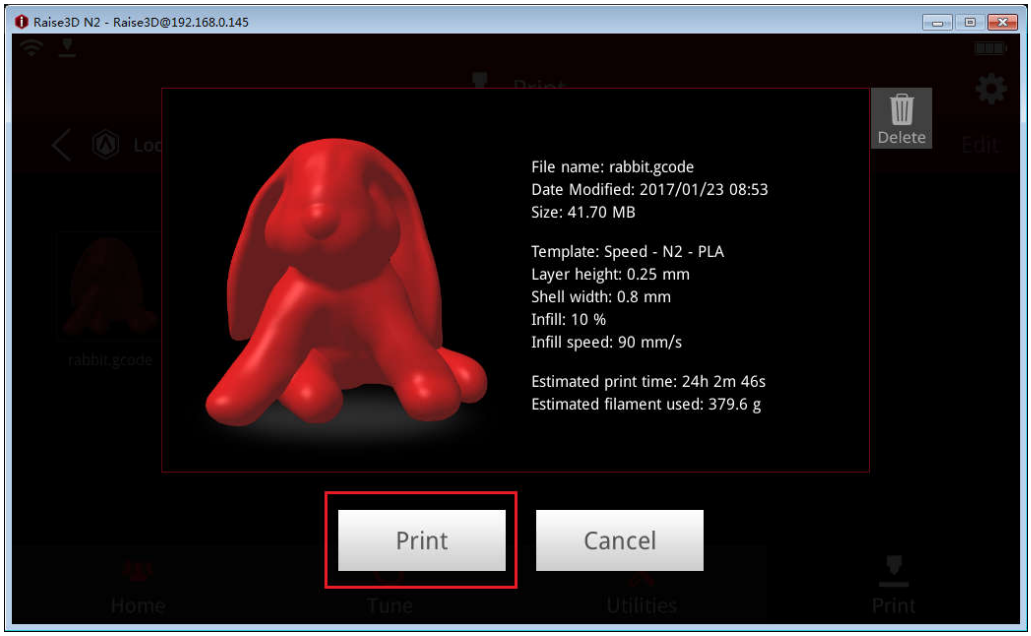


Figure 3.24: Choose one file to print and check the file information.

4 How to use ideaMaker?

4.1 Interface

When you open ideaMaker, the main screen is as shown below. We divide it into nine sections and we will go through them one by one.

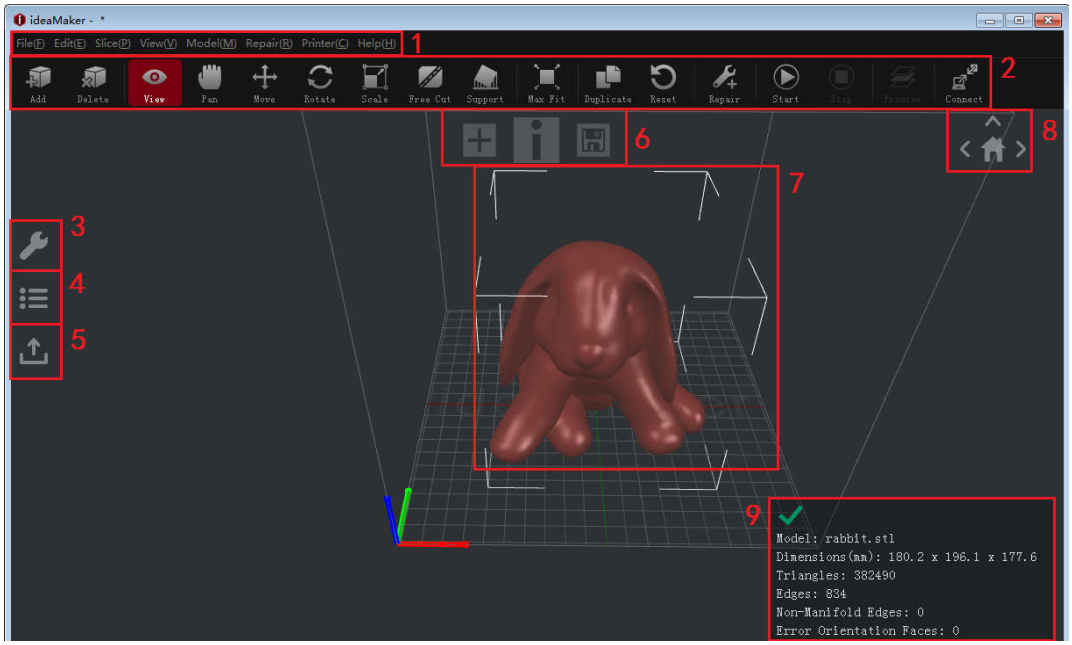
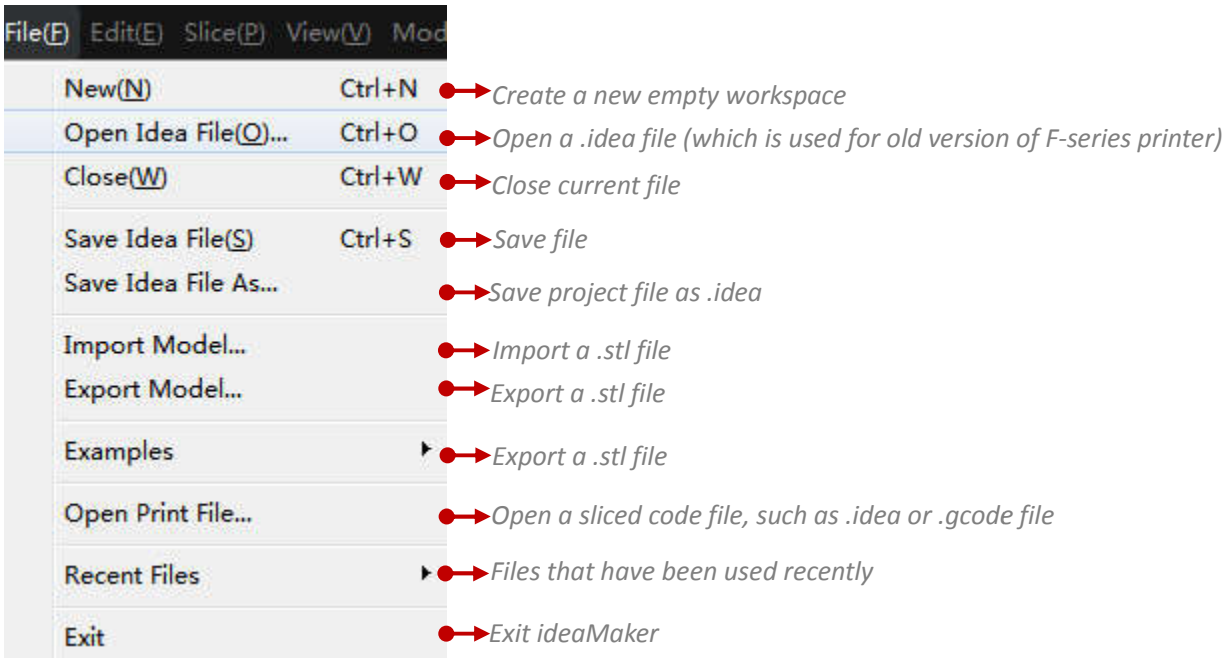


Figure 4.1: An overview over the ideaMakerinterface.

4.1.1 Menu Bar

Menu bar includes all the operation commands and advanced setting.

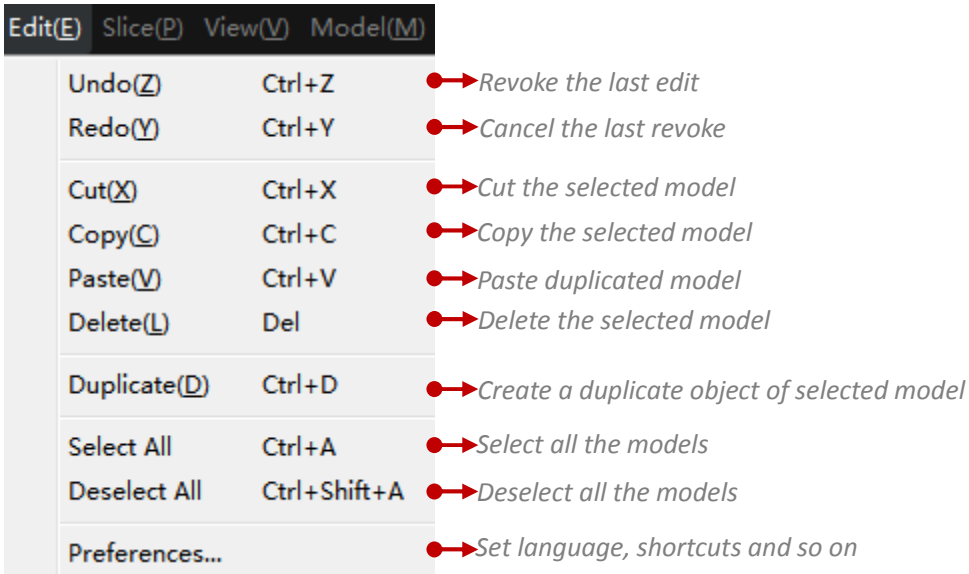
1>File



Note:

New(N): When creating a new empty workspace, ideaMaker will close the current model first.

2>Edit



Note:
There are some special options in ***preferences.***

(1) General

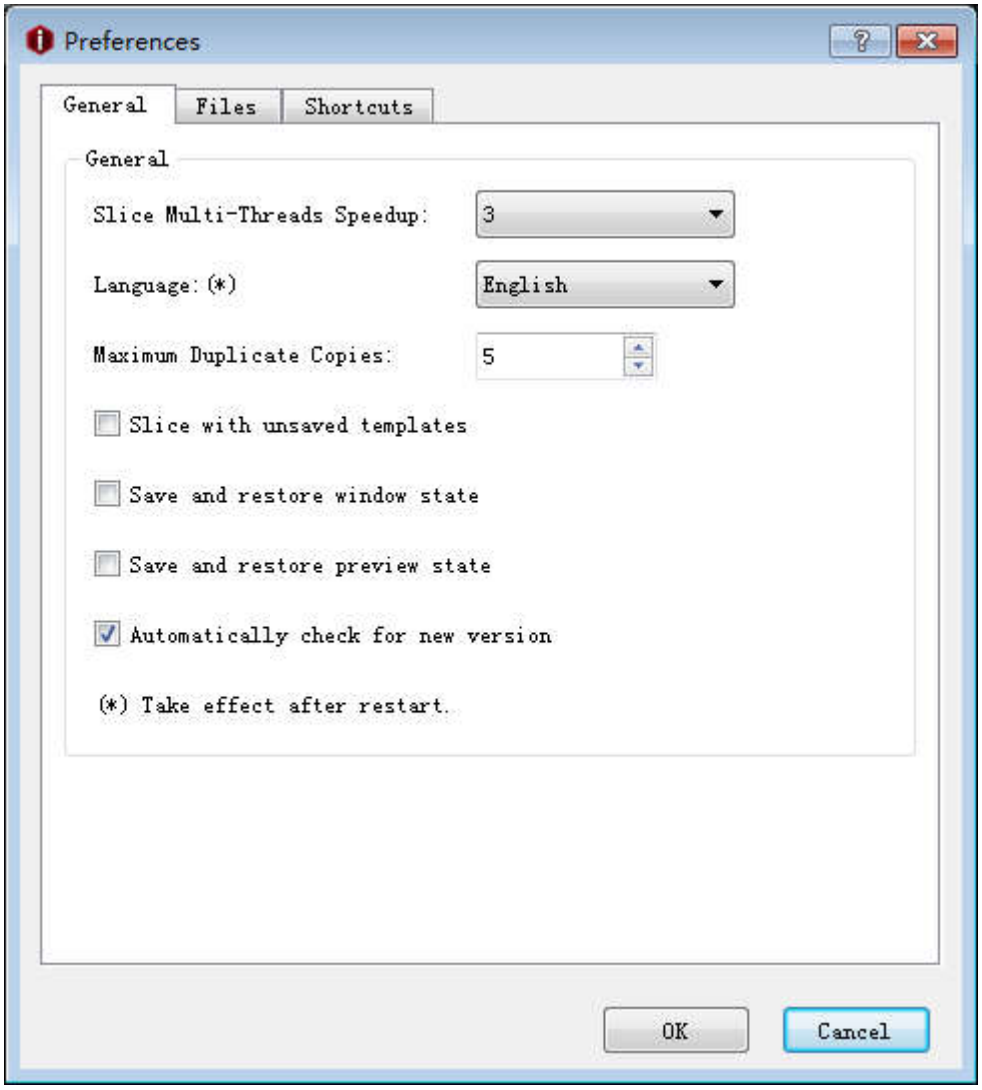


Figure 4.2: The General tab of Preference.

Slice with unsaved templates refers to the unsaved templates can be used for slicing. With enabling this function, you can select ***Restore Defaults*** or ***Restore to the Last Saved***.

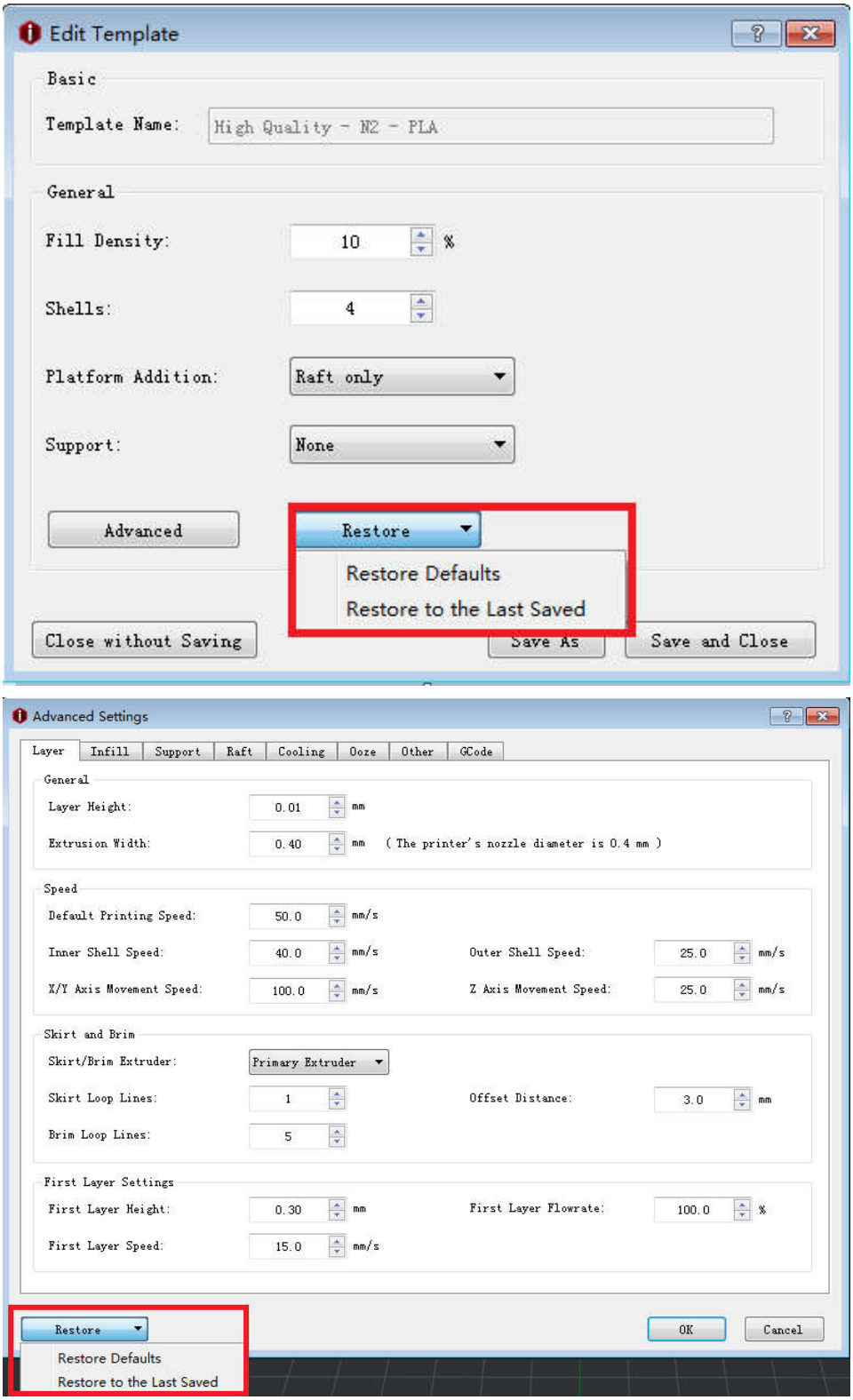


Figure 4.3: Enabled "Slice with unsaved templates".

Save and restore window staterefers towindow state can be saved and restored with enabling this function
Save and restore preview staterefers to preview state can be saved and restored with enabling this function.
Automatically check for new version refers tonew version can be checked with enabling this function.

(2) Files

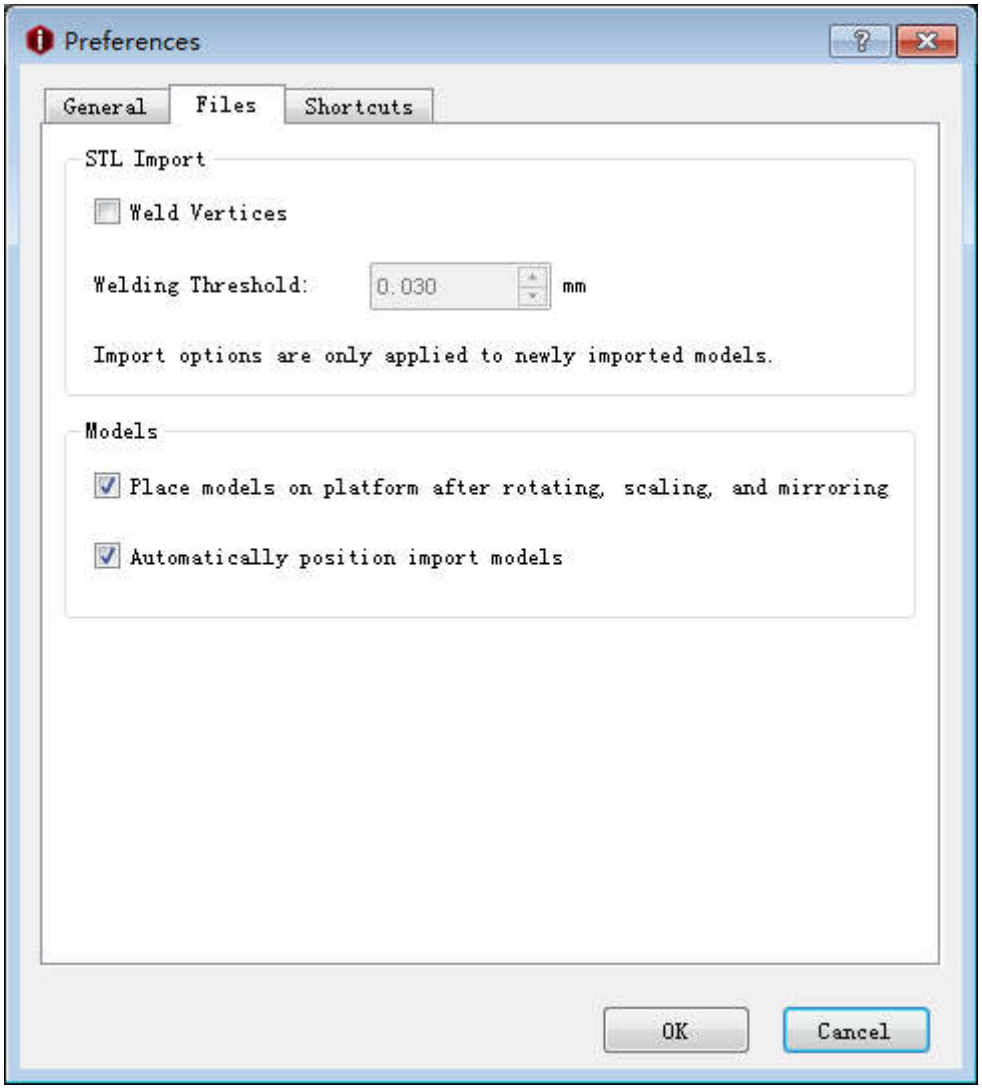


Figure 4.4: The Files tab of Preference.

Weld Verticesrefers to the nearby points will be welded with enabling this function.

Place models on platform after rotating, scaling and mirroring refers tomodel will be placed on the build platform after rotating, scaling and mirroring. As an example in rotating.

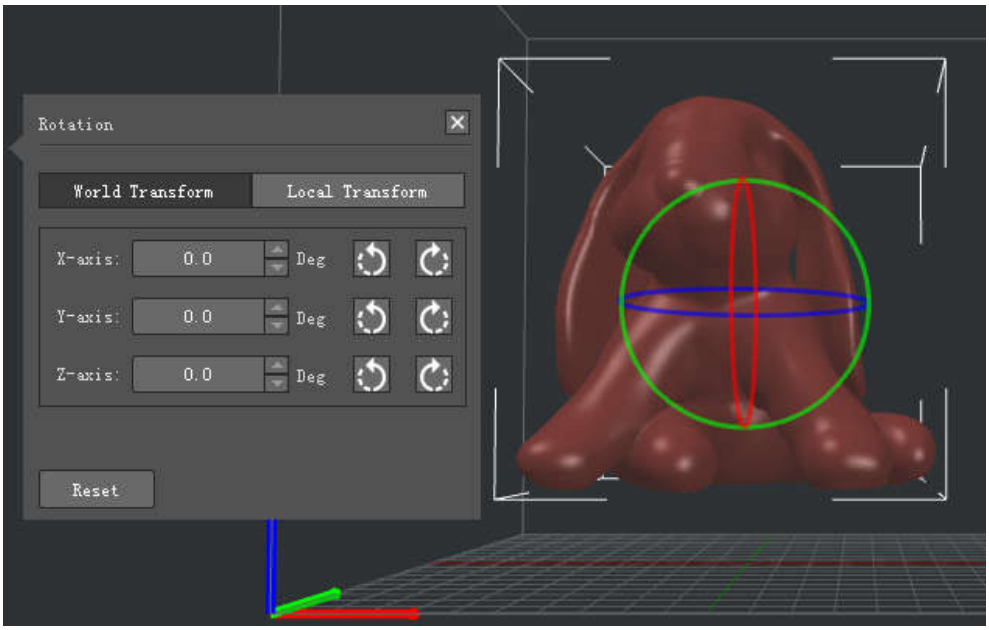


Figure 4.5: The model is floating in mid-air before rotating.

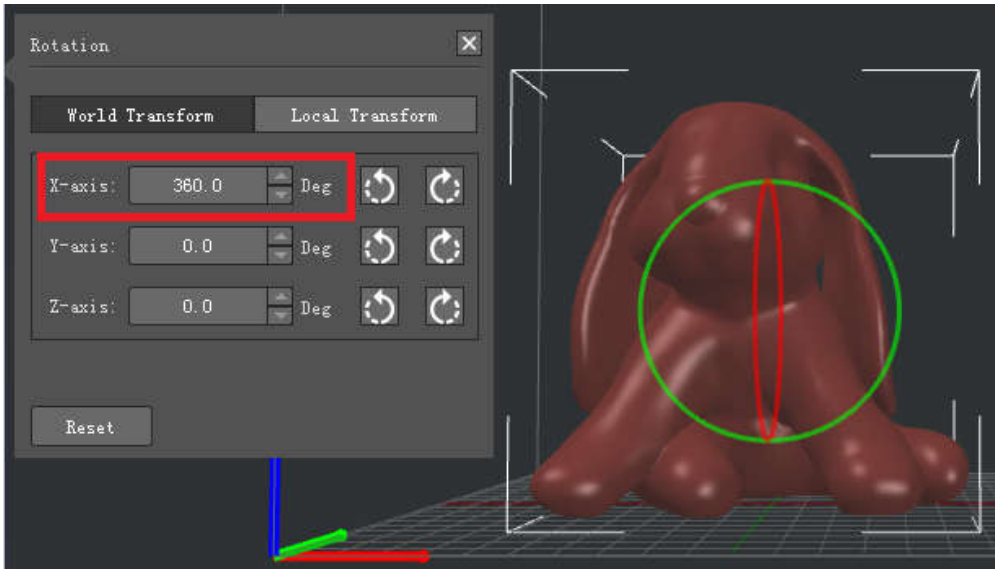


Figure 4.6: The model is placed on platform after rotating.

(3) Shortcuts

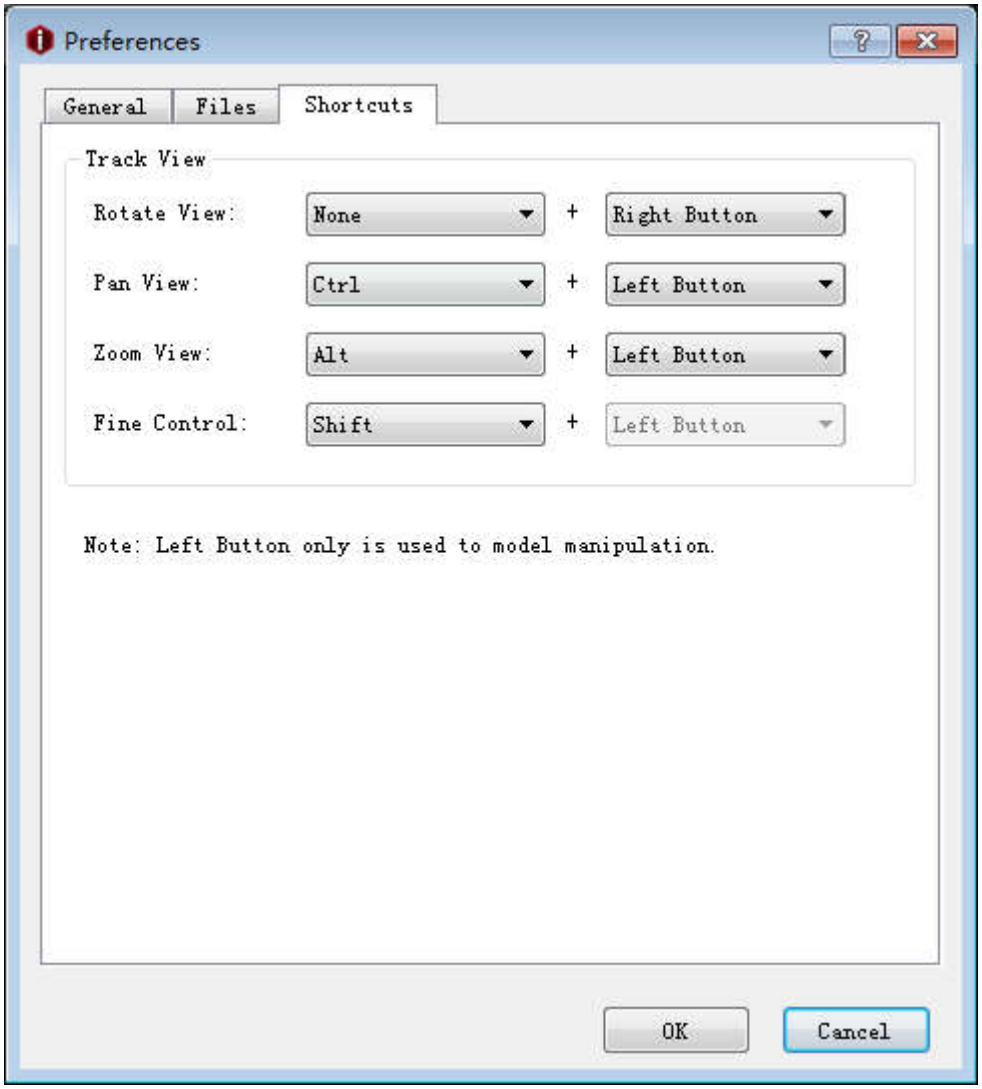
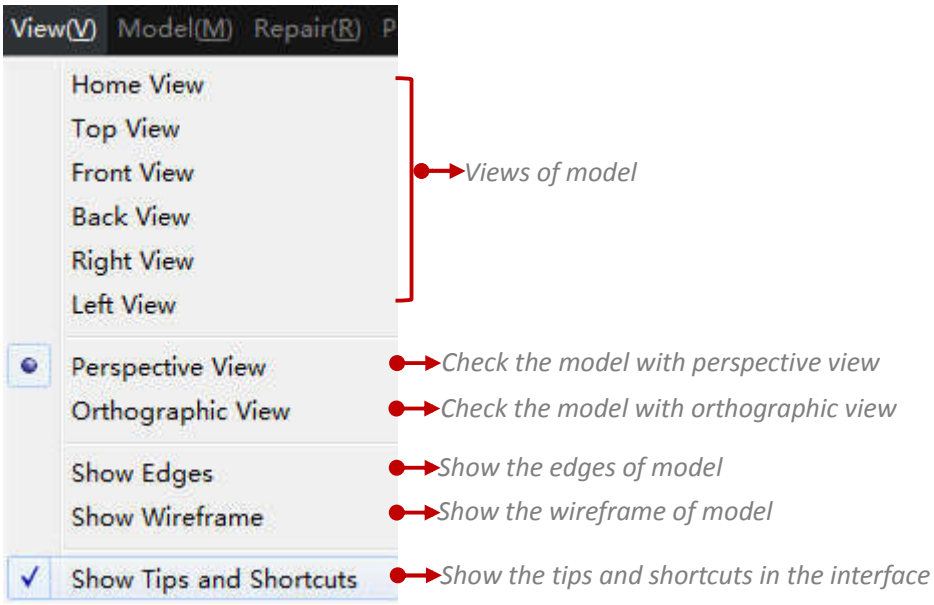


Figure 4.7: The Shortcuts tab of Preference.

3>Slice

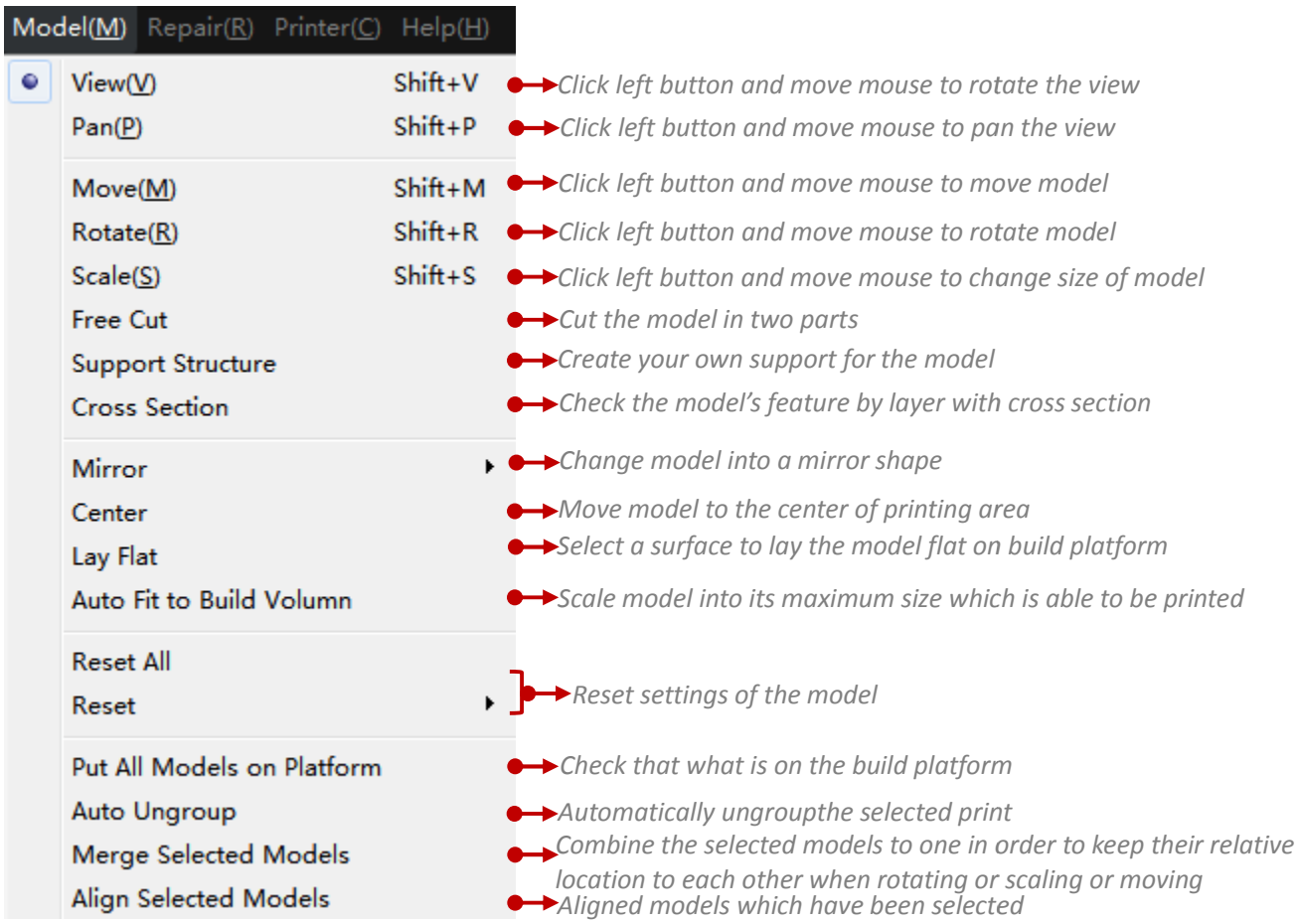
4>view



Note:

You can also change the model view freely by right click and then drag the mouse.

5>Model



Note:

Many tools have been settled in tool bar so that you can use them conveniently.

Cross Section:Check the model's feature by layer.

As an example in Z direction.***Cross Section*** will divide the model into two parts. Click on ***Reverse*** to show the upper part of the model.

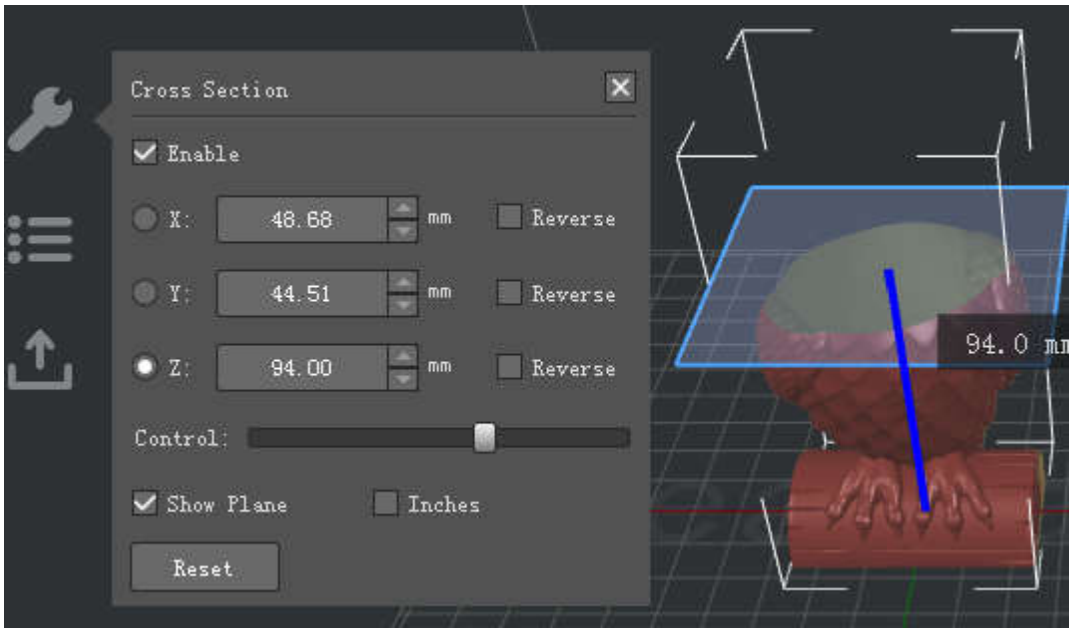


Figure 4.8: Cross Section in the Z axis direction.

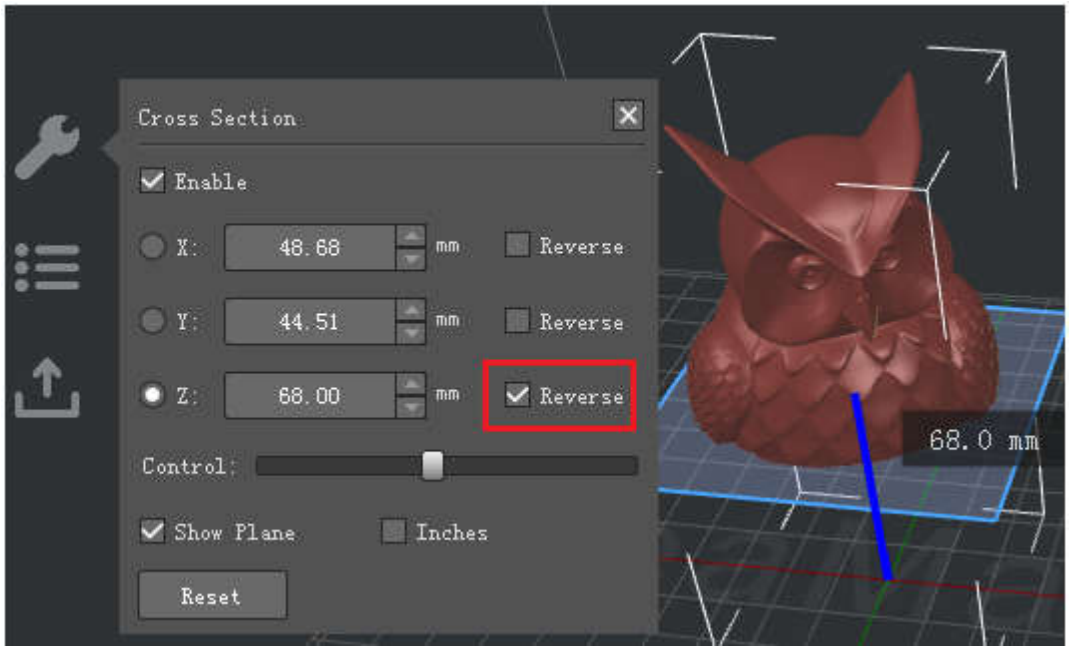


Figure 4.9: Cross Section in "Reverse" of Z axis direction.

Show Planewill toggle the visibility of the cross section plane.

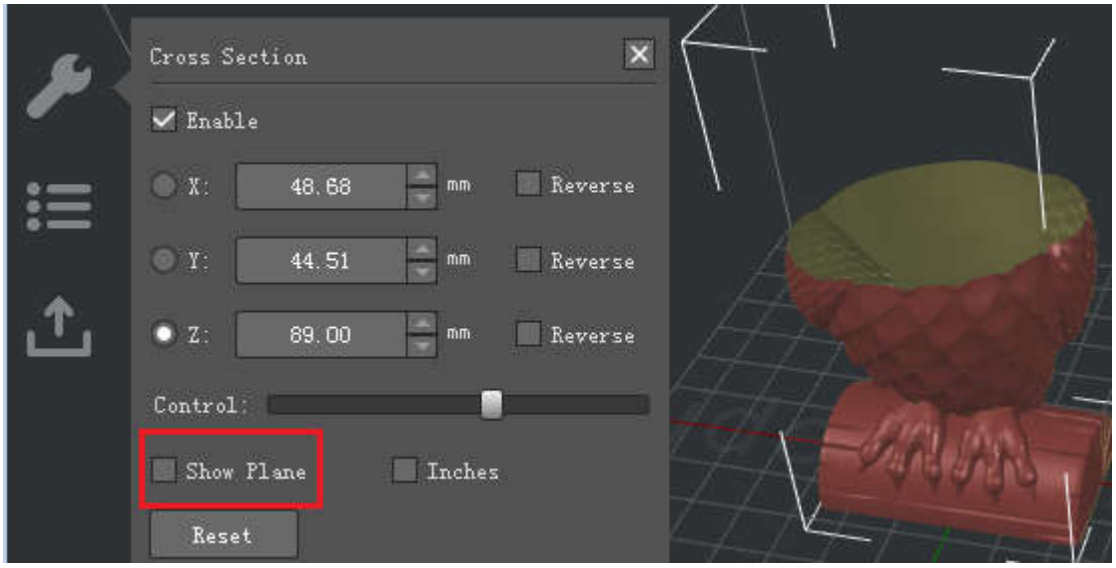


Figure 4.10: Cross Section without "Show Plane".

Incheswill choose the unit between inches or mm.

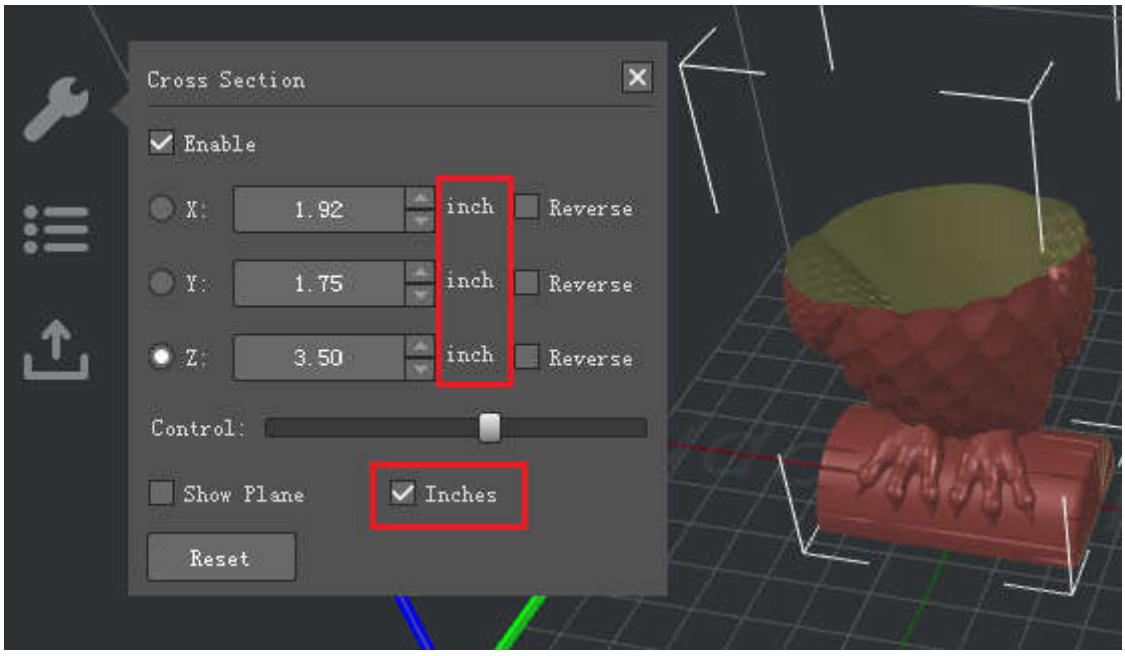


Figure 4.11: Cross Section with "Inches".

Note:

Cross Section will affect the slice result which will hide the part of model.

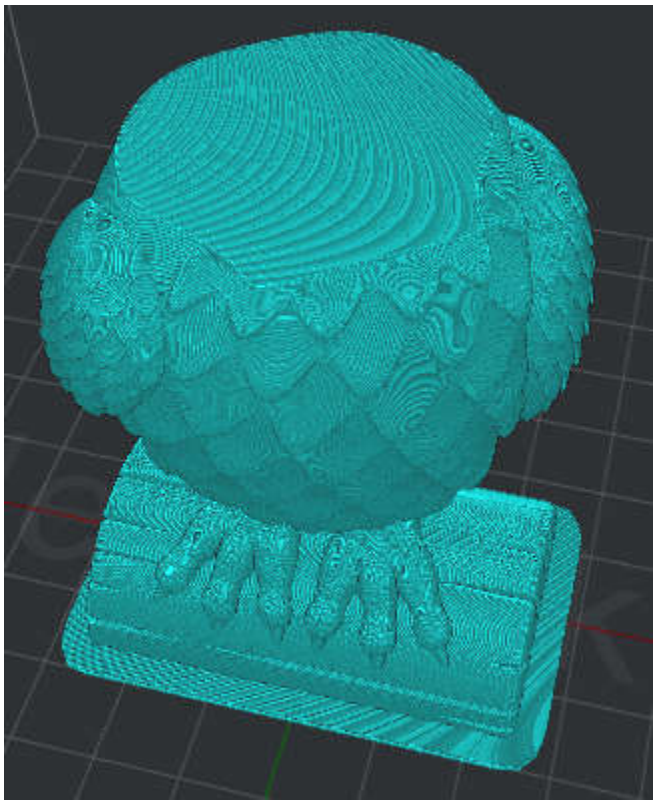


Figure 4.12: The slice preview with "Cross Section".

Lay Flat: select a surface to lay the model flat on the build platform.

Face Index refers to the face you selected.

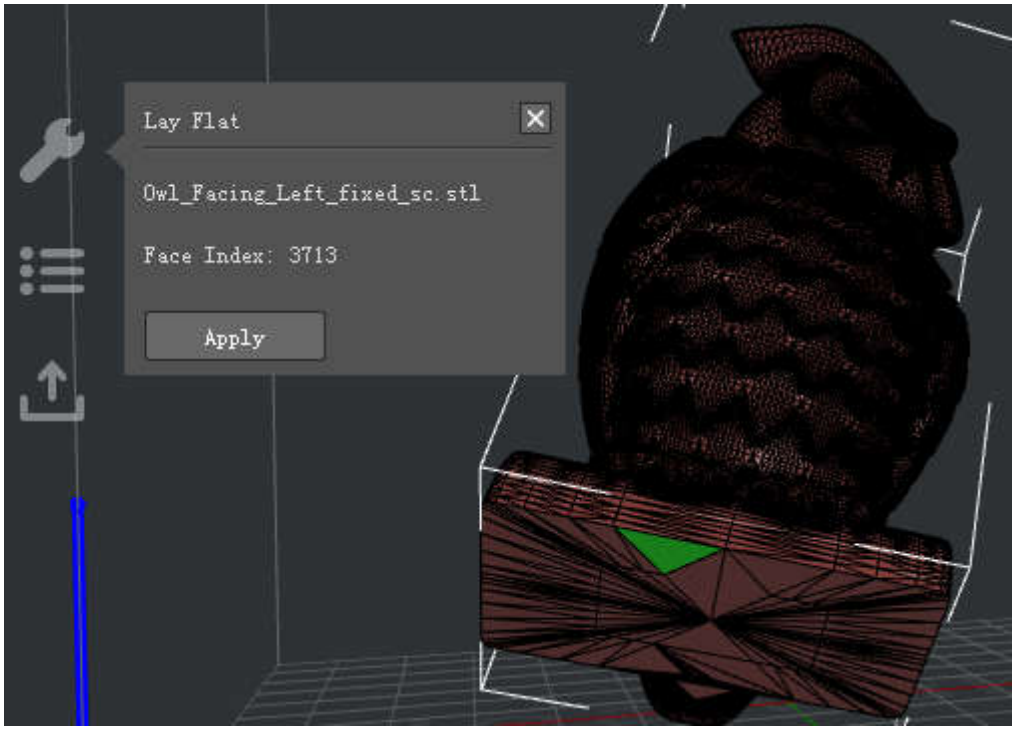


Figure 4.13: Click "Apply" to lay the model flat on the platform.

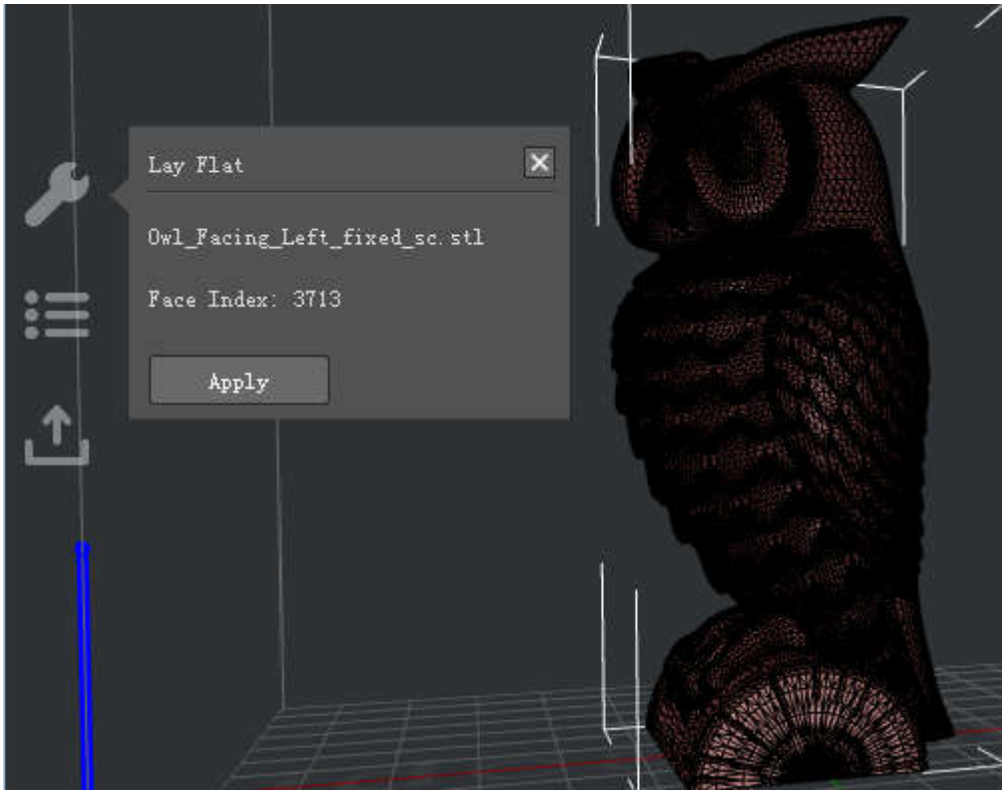
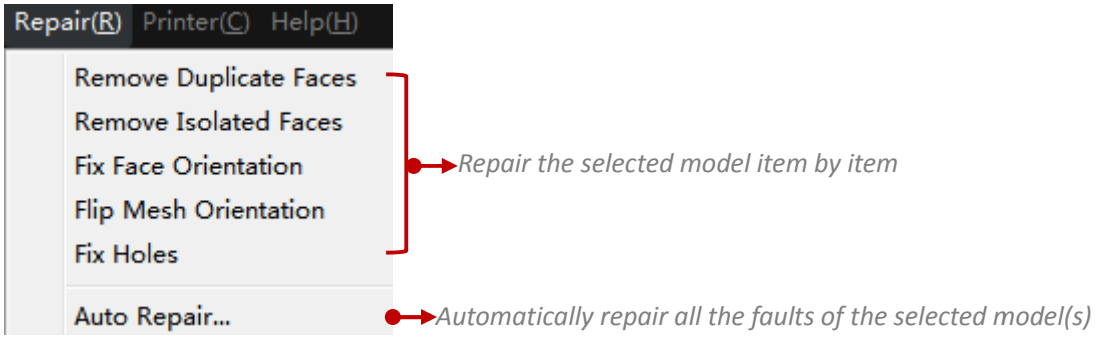


Figure 4.14: The model with "Lay Flat".

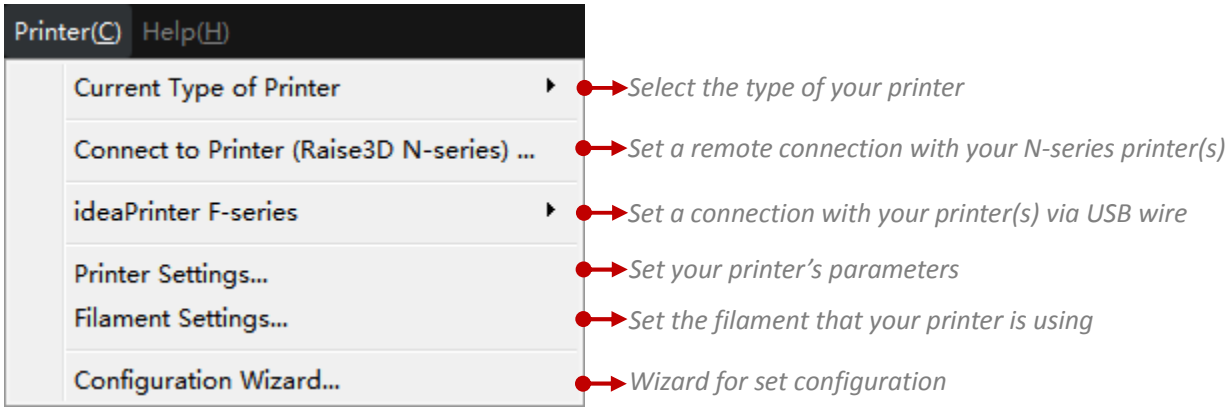
6> Repair



Note:

You can also find auto repair icon in tool bar.

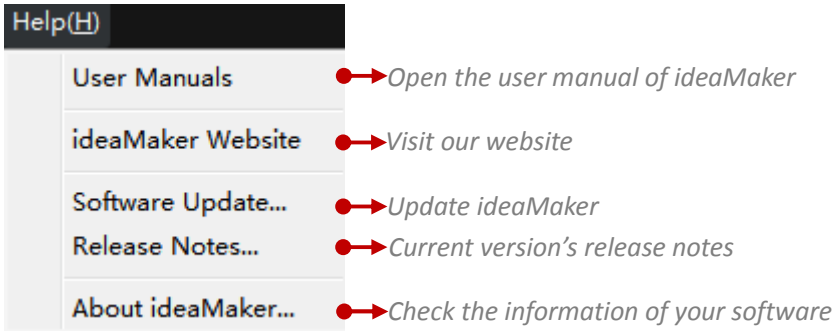
7>Printer



Note:

Filament Settings: You can set type, diameter, density, price and compensation of the filament. Also you can build a new filament template for you own filament here.

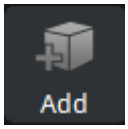
8>Help



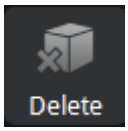
4.1.2 Tool Bar

Tool buttons for slicing models, these buttons are shortcut of menu bar.

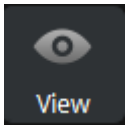
Figure 4.15: The tool bar.



: Add new .stl or .obj file.



: Delete the selected model.



: When you enable this button, you can set model's color and which nozzle to print it (if you have your printer as a dual-extruder printer).

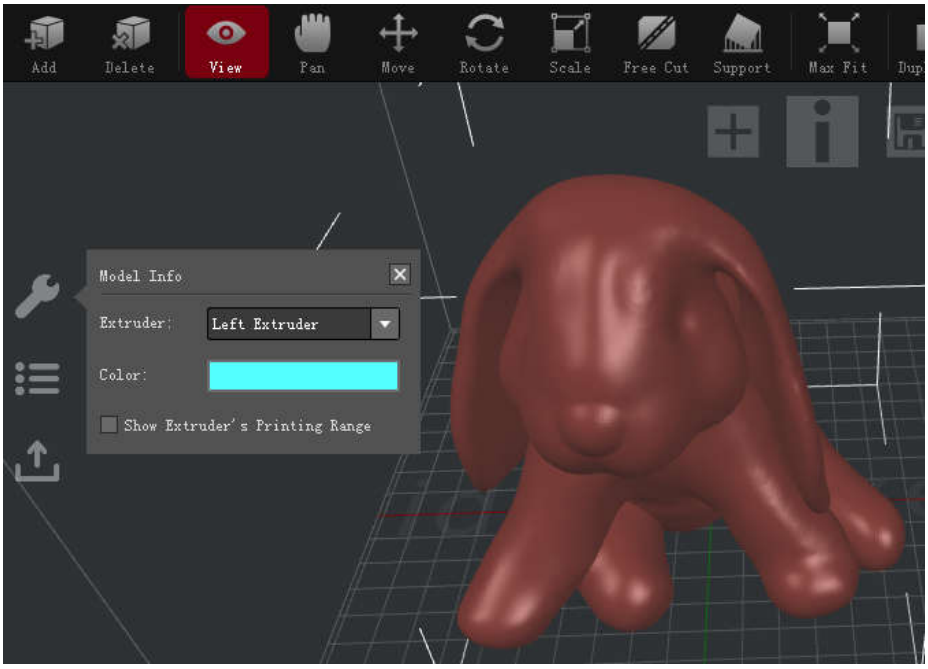


Figure 4.16: Check models with different extruders in different colors.

(Enabling this button also allows us to watch model in different angles by left-drag your mouse.)

:When you enable this button, you can pan the view by left-drag your mouse.

: When you enable this button, you can move model to another place by left-dragthe blue arrow above the model with your mouse to move the selected model in Z direction. And move the model freely in X and Y direction by left-drag the selected model. You can also set the exact X/Y/Z coordinate value to move the selected model in the operation property zone.

Make Center refers to moving the model to the center of your build plate.

On Platform refers to dropping the model on the build plate.

Align Together refers to aligning the models to the default position in 3D designing software when printing several models in one time.

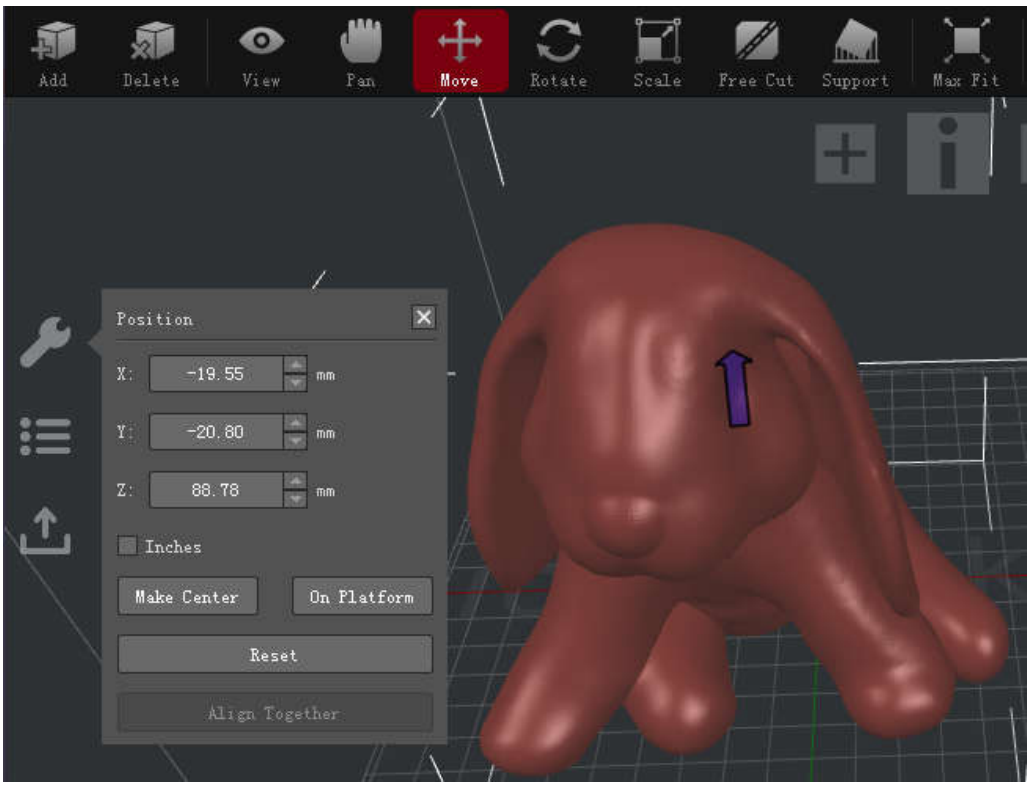
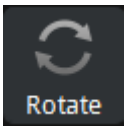
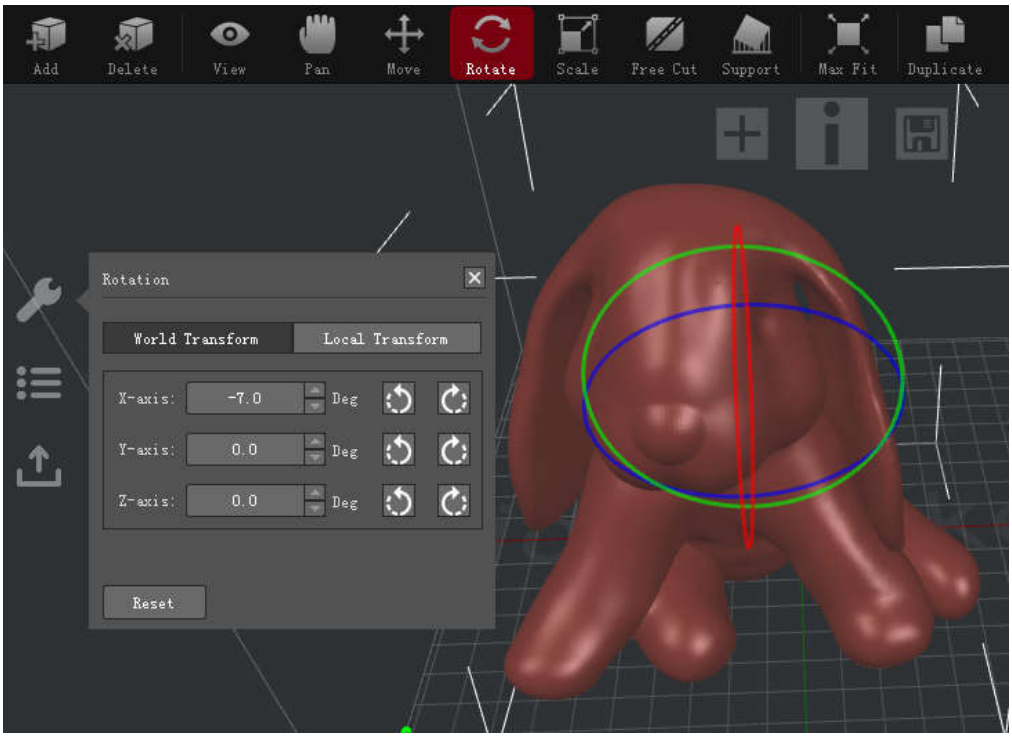


Figure 4.17: You can drag the selected model freely in X & Y directions with left-click. And click the blue arrow to drag the model in Z direction.



:When you enable this button, you can rotate model to another angle by left-drag your mouse. Or also set the exact angle value to rotate model in the operation property zone which based on the aircraft principal axes. The ***Roll*** refers to longitudinal axis. ***Pitch*** refers to lateral axis. ***Yaw*** refers to vertical axis.



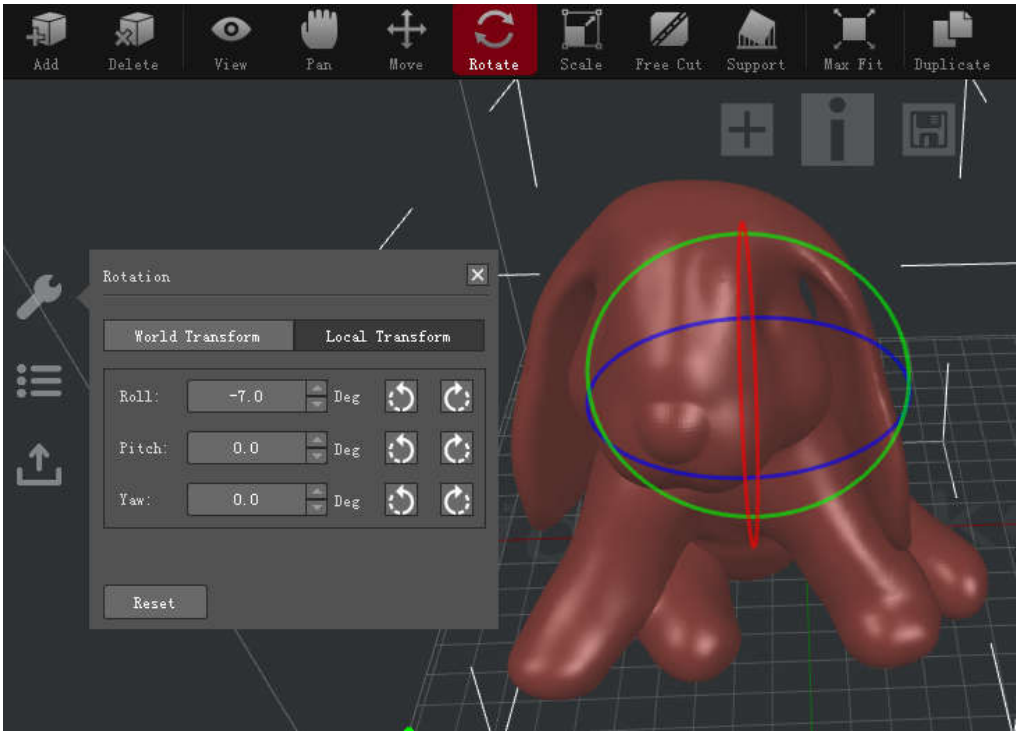


Figure 4.18: You can use both the color rings and left values to rotate the model.

: When you enable this button, you can amplify model's size by left-drag your mouse.

You can also set the exact amplify rate value to change model in the operation property zone.

Inch refers to changing size display from mm to inch.

Uniform Scaling refers to scale the width, depth and height in the same scale.

The percentages at beside the size show the relative scale of the original size.

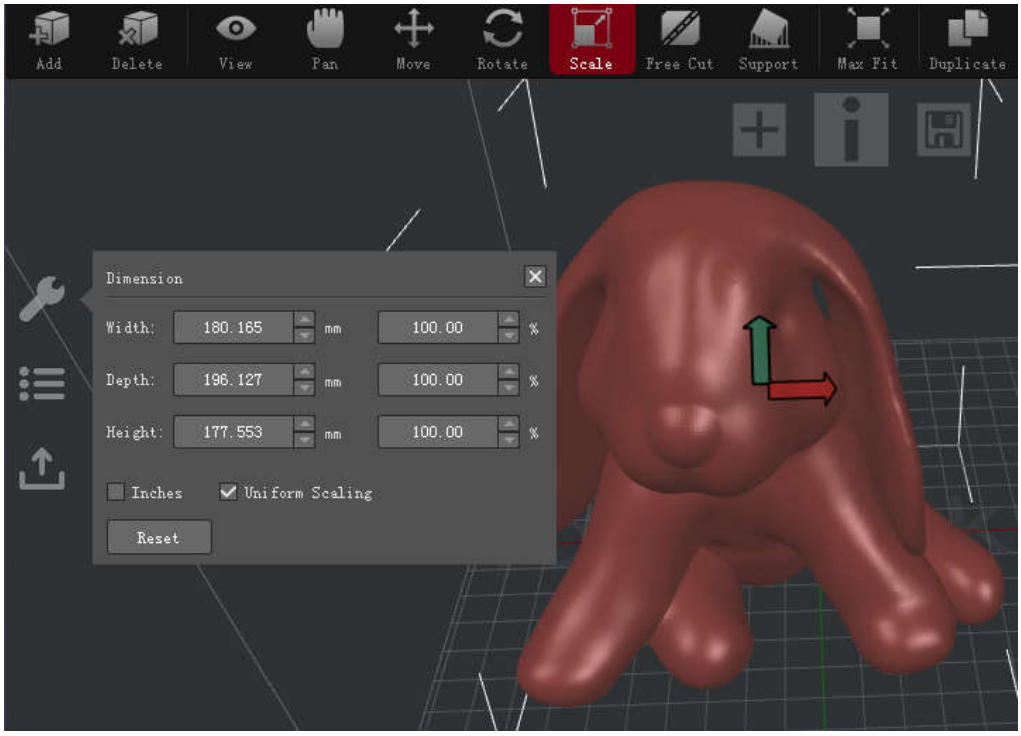


Figure 4.19: Scale the model by You can use both the color arrows and left values to scale the model.

: When you enable this button, there will be a plane which you can move to cut the model apart.

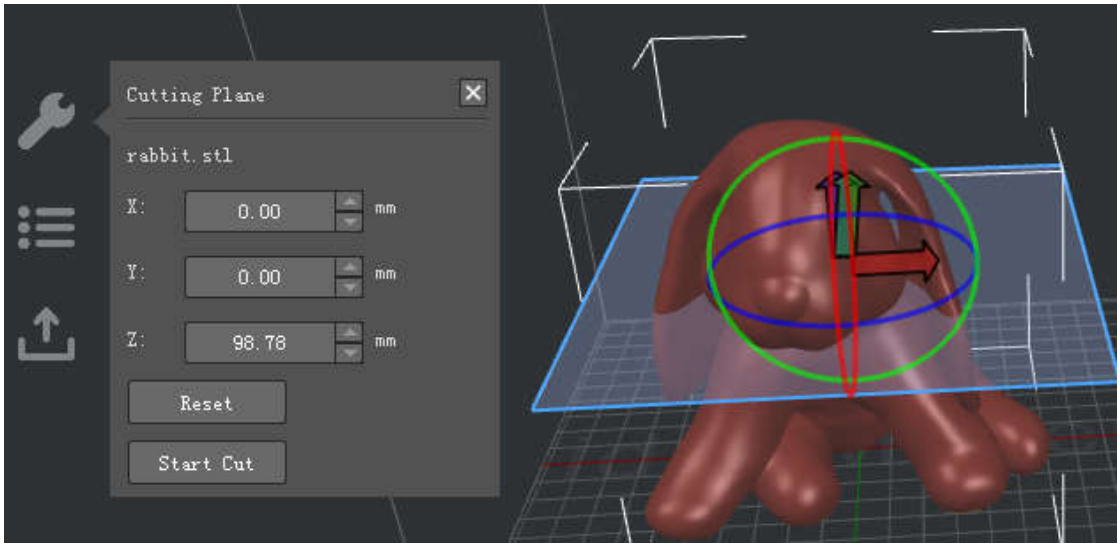


Figure 4.20:Click"Start Cut" to cut the model apart.

Now you will have three options to cut the model you selected.

Option 1:Move the red, blue, green arrow. As an example in red arrow.

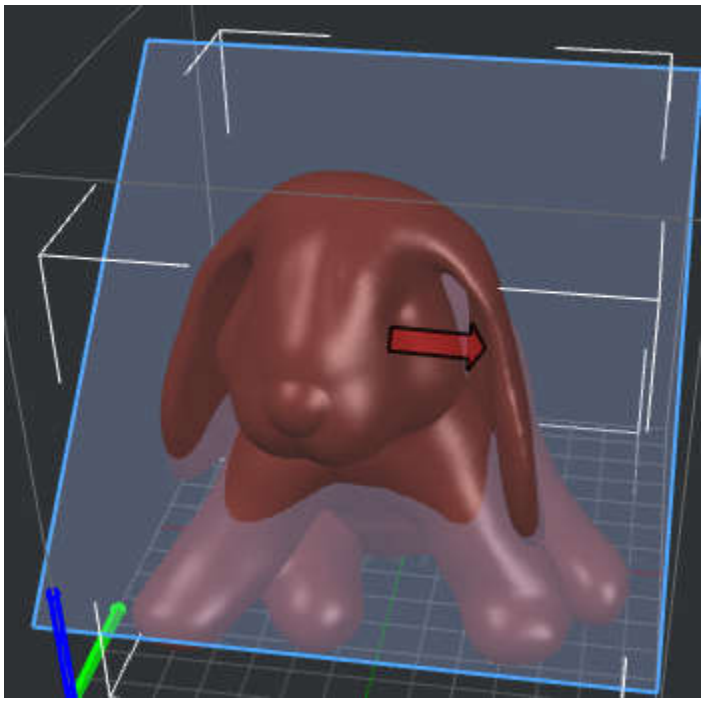


Figure 4.21: Click red arrow to move the "Cutting Plane".

Option 2: Rotate the red, blue, green circle. As an example in red circle.

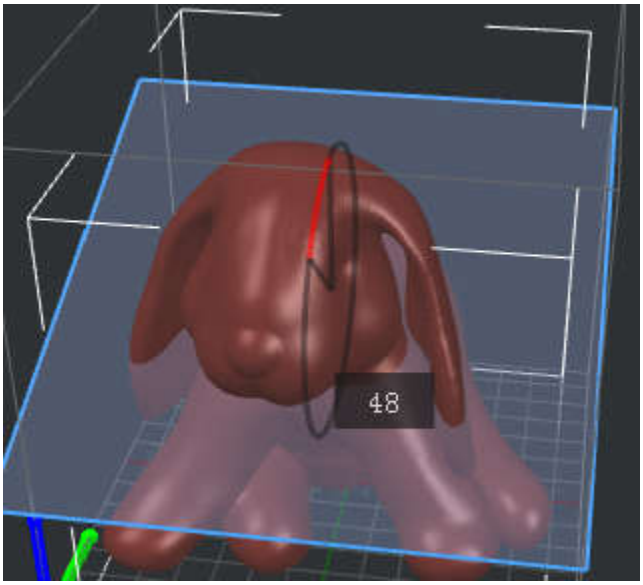


Figure 4.22: Click red circle to rotate the "Cutting Plane".

Option 3:Input the exact value in. As an example in Z direction.

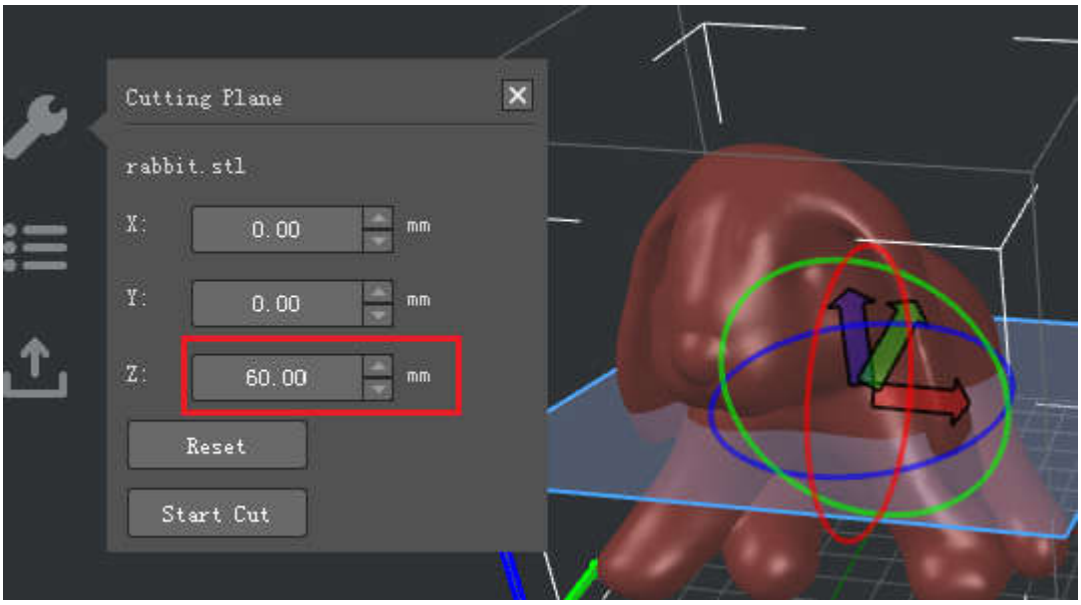


Figure 4.23: Input values to move the "Cutting Plane".

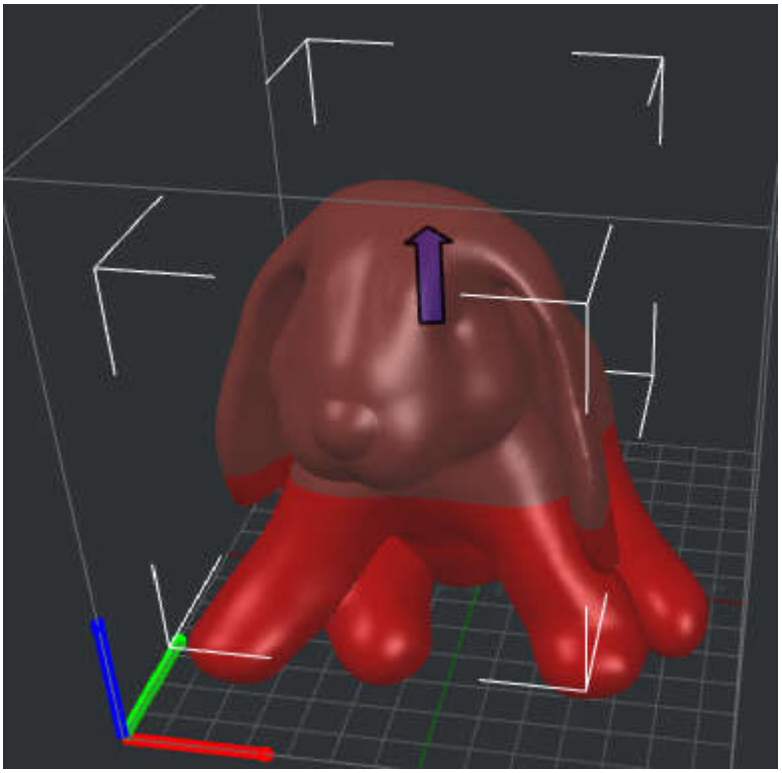


Figure 4.24: The selected model have been cut into two parts.

: Click on the ***Support*** icon in the toolbar, you can create supports in the operation property zone.

Auto Supports:

Pillar Size refers to the size of each pillar in support.

Overhang Anglerefers to the minimal overhanging angle which needs support to print.

Touch Platform Only refers to only creating the supports which will touch the build platform.

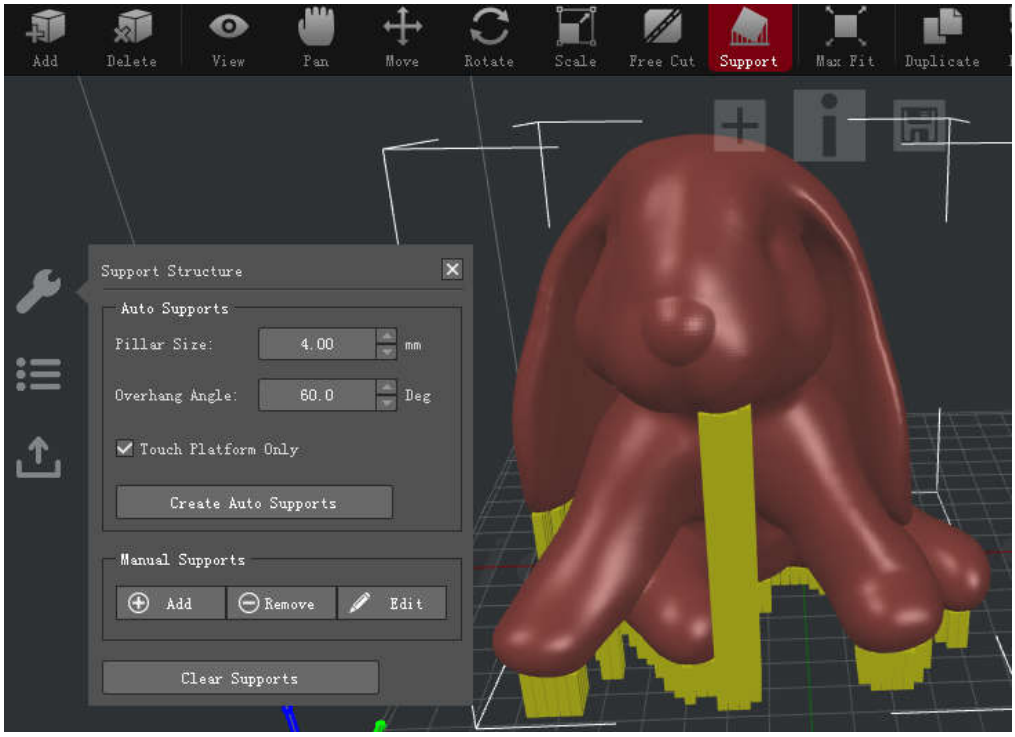


Figure 4.25: You can set the "Pillar Size" and "Overhang Angle" first. And try "Create Auto Supports". Then adjust the support with "Manual Supports".

Manual Supports:

Add one support pillar, ***Remove*** one support pillar and ***Edit*** the support size.

Pillar Z-Pos refers to the start height of the selected support pillar.

Pillar Height refers to the full length of the selected support pillar.

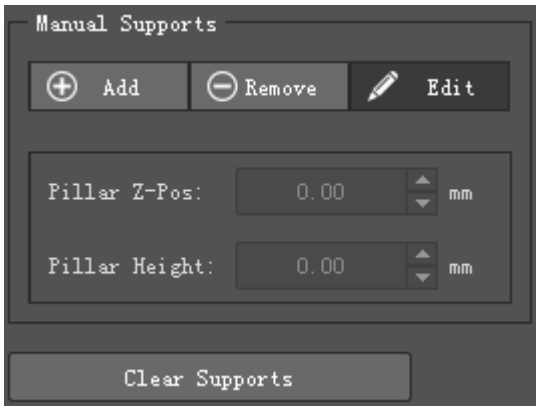
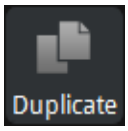


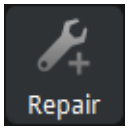
Figure 4.26: These two options will only appear after you click the "Edit" button.

:When you click this button, the selected model will be amplified to max size that your printer can print.



:When you click this button, the selected model will be copied.

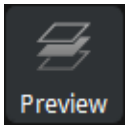
:Revoke all the settings of the selected model.



:Automatically repair model’s defects.

: Start to slice the model.

: Abort slicing.



: Show the sliced file in layers.

: Connect to the printer with your computer via WLAN.

4.1.3 Operation property

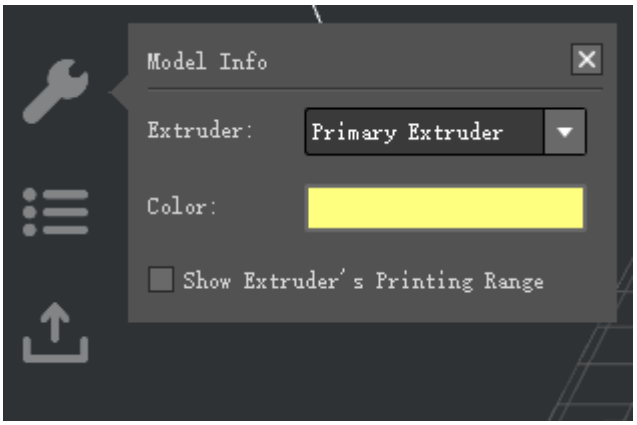


Figure 4.27: The information shown here depends on which function you are using.

This section shows the information and settings of the selected model.

4.1.4 Model list

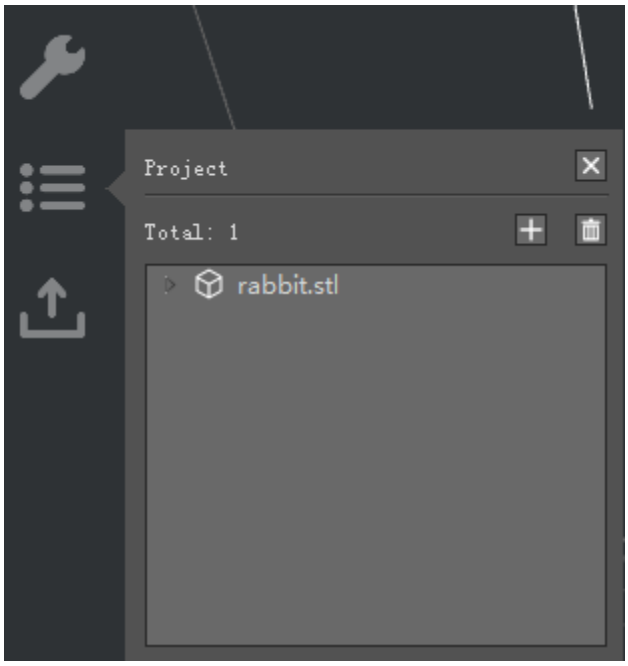


Figure 4.28: You can also add or delete models here.

This section shows the basic information of models such as the quantity of the faces or the size of the models.

 : Click **"+"**to add other models.

 : Delete the model you have selected.

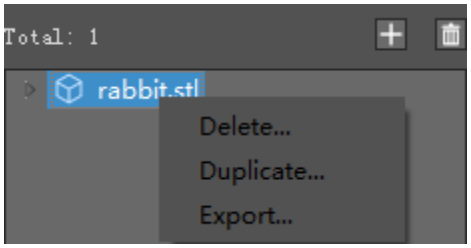


Figure 4.29: Right-click the blue font section will display Delete, duplicate, Export.

Delete refers to the model will be deleted via this button.
Duplicate refers to the model will be copied via this button.
Export refers to the model you have selected will be export via this button.

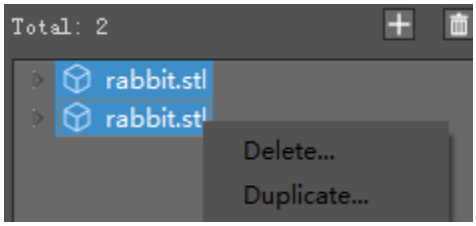


Figure 4.30: Please export model one by one.

4.1.5 Uploading queue

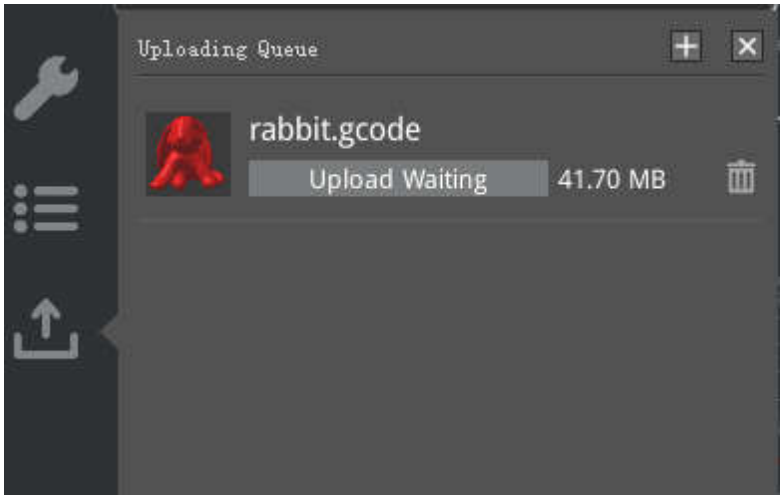


Figure 4.31: ideaMaker also can upload .gcode files sliced by other slicing software. Press "+" button to find the files you want to upload to your N-series printer from your computer.

This section shows the uploading list to your printer.

4.1.6 Instant operation bar

 : Add models

 : Start slicing directly

 : Save as another printing file

4.1.7 Model preview

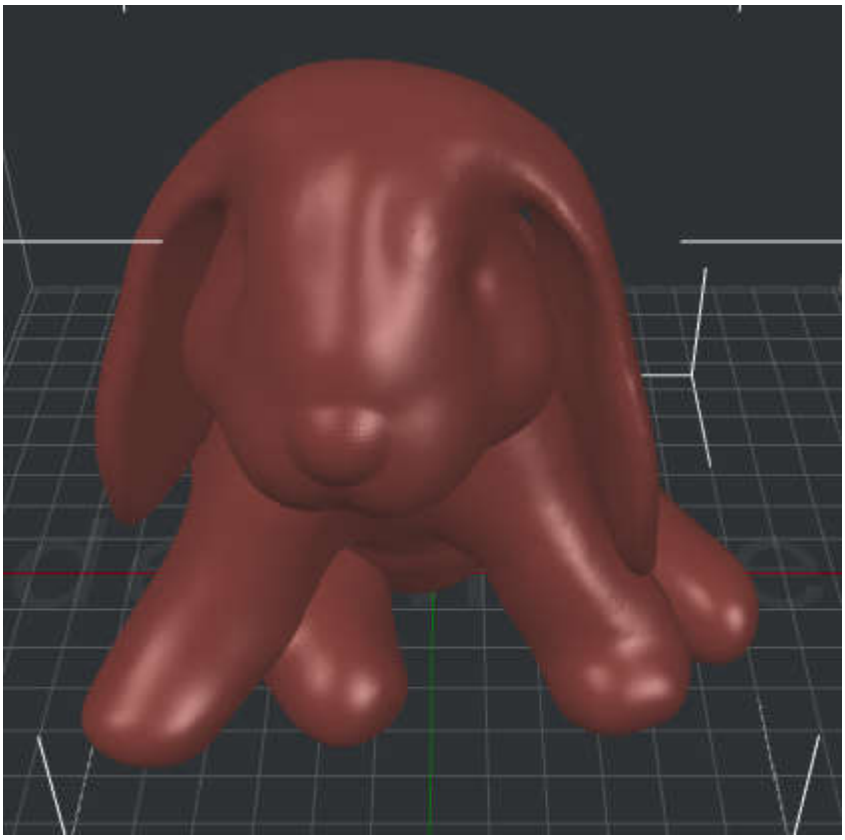


Figure 4.32: Preview your model.

This section is designed for previewing the current model.

4.1.8 Perspective transformation

Figure 4.33: It will be set as perspective view by default. If you want to check your model with orthographic view, click View -> Orthographic View.

Check the modelwith the default views.

4.1.9 Detecting Information

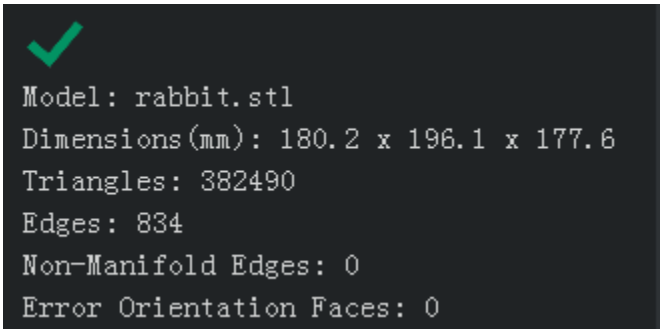


Figure 4.34: If here appears an error or a warning, try Auto-Repair.

Check the correct or warning information of the model's auto-detection.

Dimensions refers to the size of the model.

Triangles refers to the number of triangles in the model.

Edges refers to the number of edges in the model.

Non-ManifoldEdges refers to the number of non-manifold edges in the model.

Error Orientation Faces refers to the number of error orientation faces in the model.

4.2 Slicing settings

There are many settings you can change to optimize the print results in ***Edit*** when slicing.

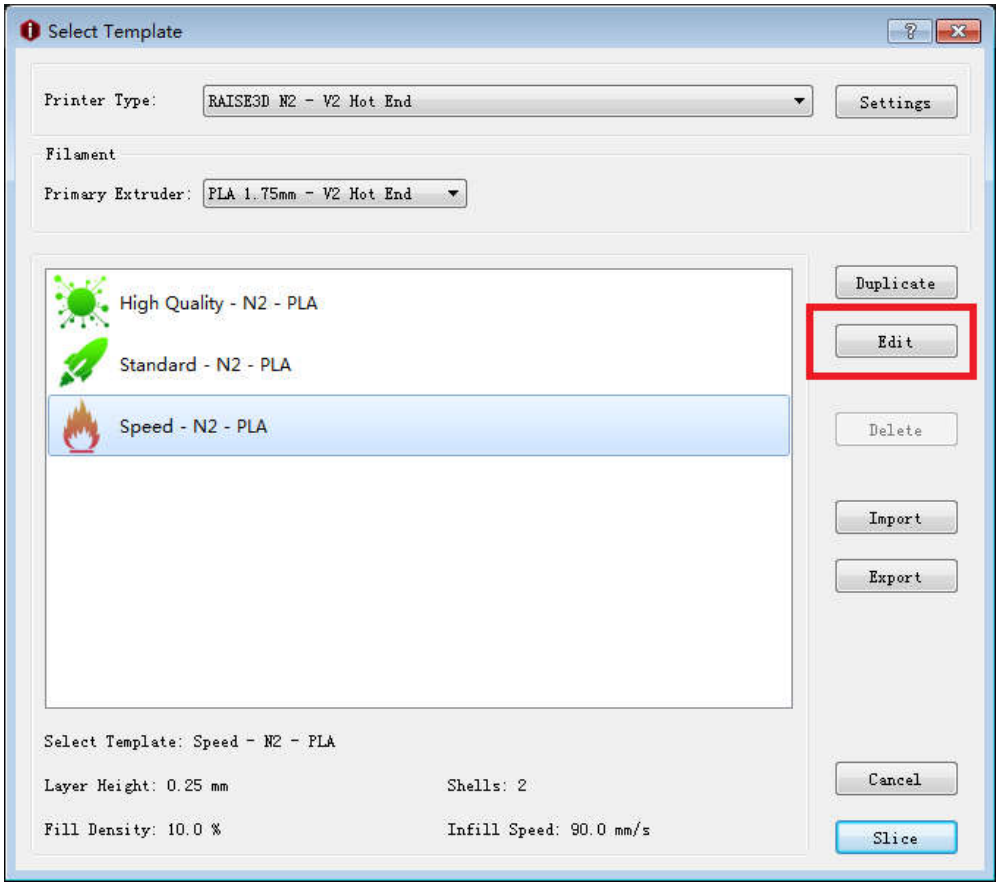


Figure 4.35: Choose a template and press "Edit" to enter the setting page.

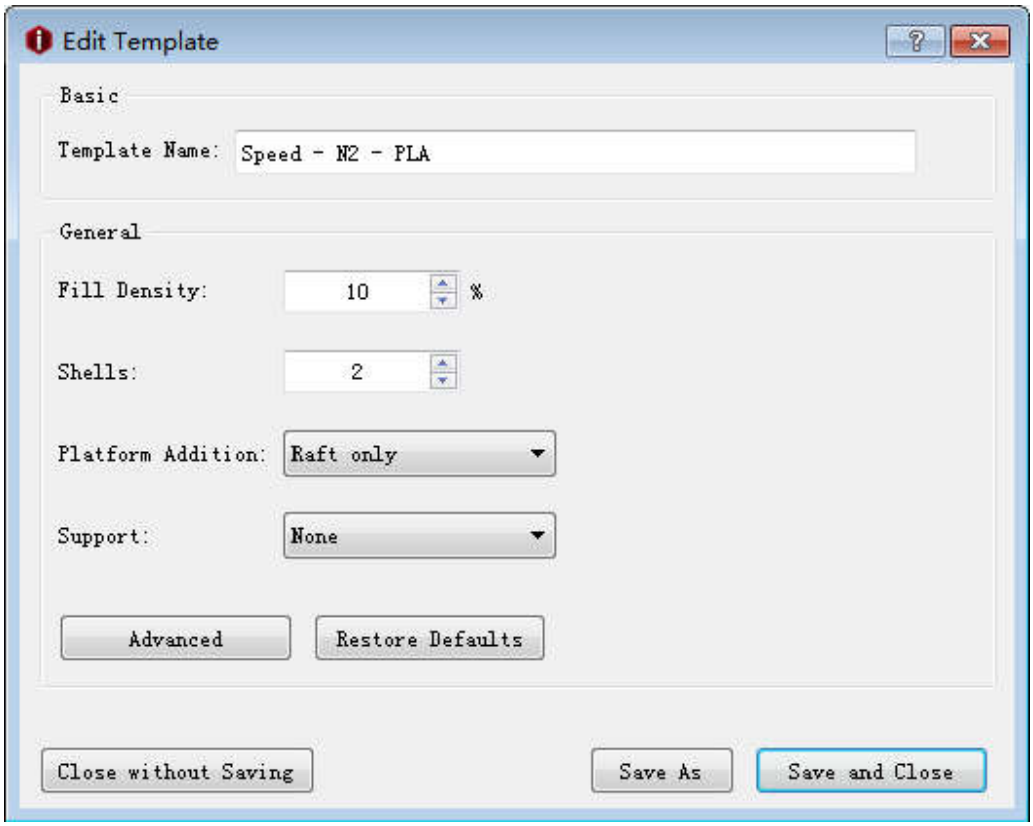


Figure 4.36: Common parameters.

Fill Density refers to the density of infill inside the model, the more infill the model will be the more solid.

Shells refer to the thickness of model's wall.

Platform Addition refers to the type of bottom layer.

The **Skirt** layer means the circle outside of model or raft.

The ***Raft*** layer will print couple of thick layers as model's ground.

The ***Brim*** layer will only print a single layer of shell around the model to enlarge its touch surface to build plate.

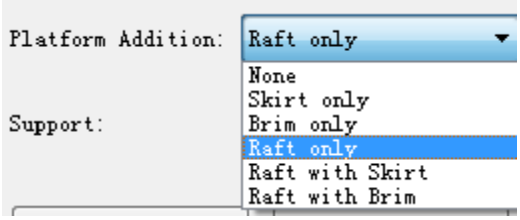


Figure 4.37: There are 3 types of Raft.

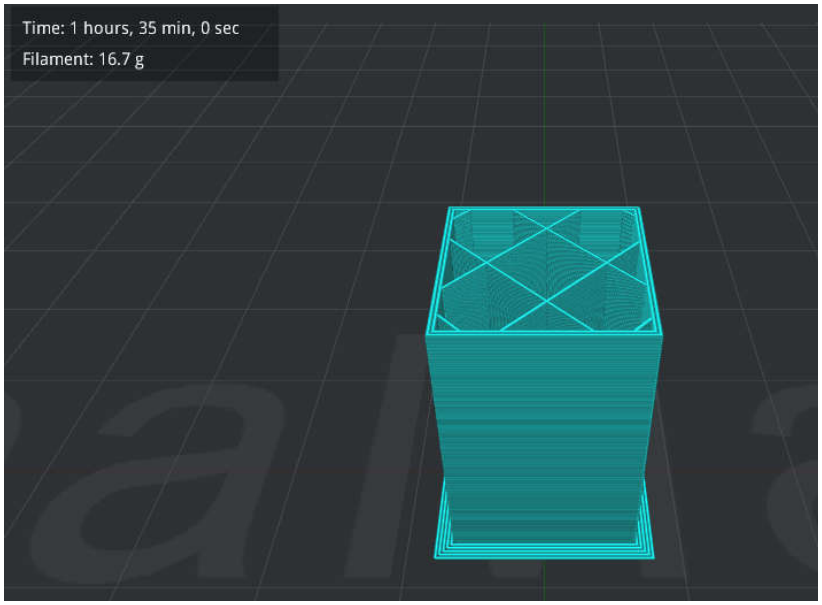


Figure 4.38: Printed with 2 shells, 10% fill density and Brim.

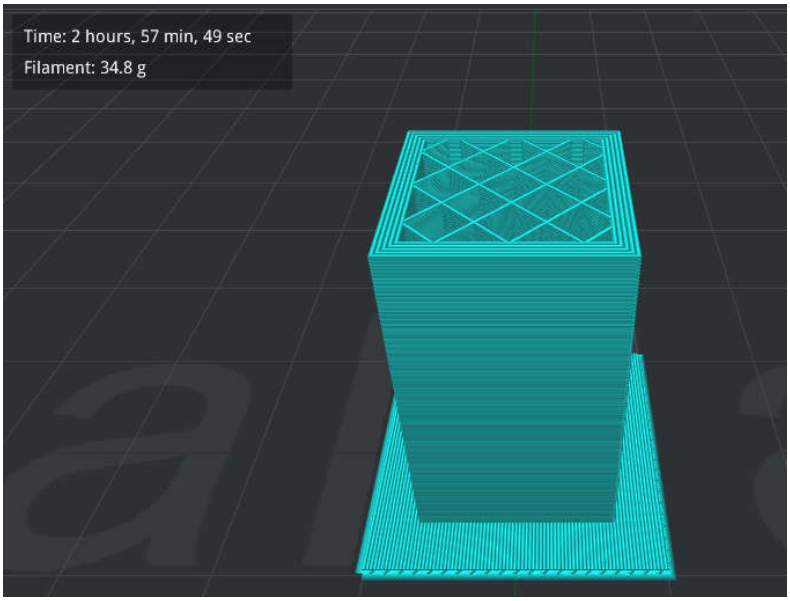


Figure 4.39: Printed with 5 shells, 20% fill density and Raft.

Support means that the printer will print support structure for model's overhang part.

The ***None*** setting refers to no support structure for the model.

The ***Touch Platform Only*** setting refers to only adding support structure which can touch the build platform. Those located from one surface of the model to another surface of the model will not be created.

The ***All*** setting refers to adding support structure to all the overhang part of the model.

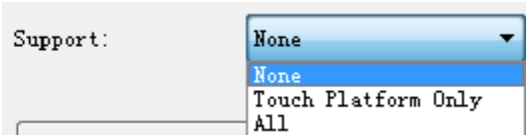


Figure 4.40: There are 3 types of supports.

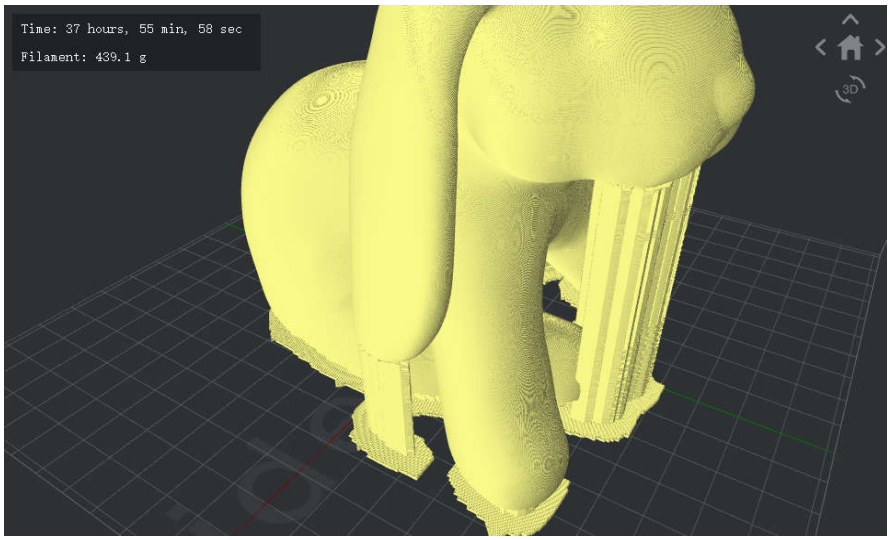


Figure 4.41: Printed with supports of "Touch Platform Only".

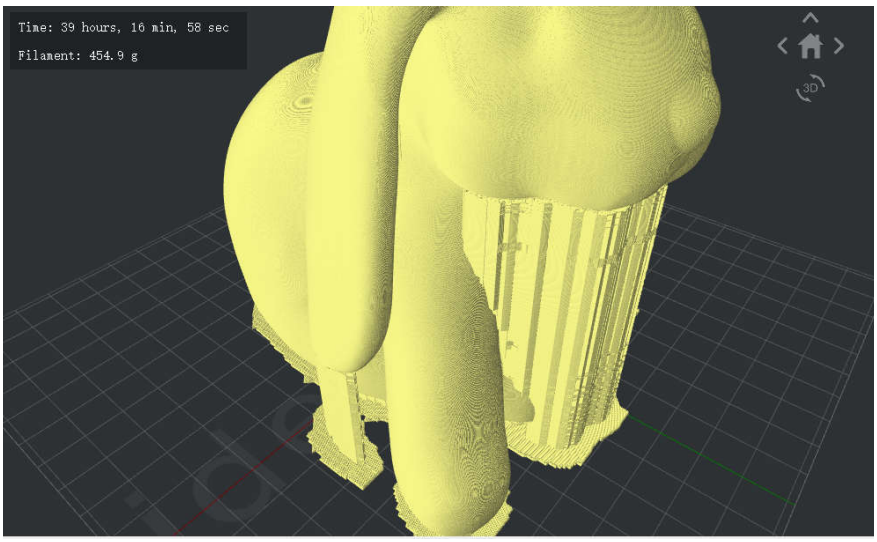


Figure 4.42: Printed with supports of "All".

Note: Advance Setting.

In ***Edit Template***, click ***Advanced*** to go to ***Advance Settings*** interface which include ***Layer***, ***Infill***, ***Support***, ***Raft***, ***Cooling***, ***Ooze***, ***Other***, ***Gcode*** and we will go through them one by one.

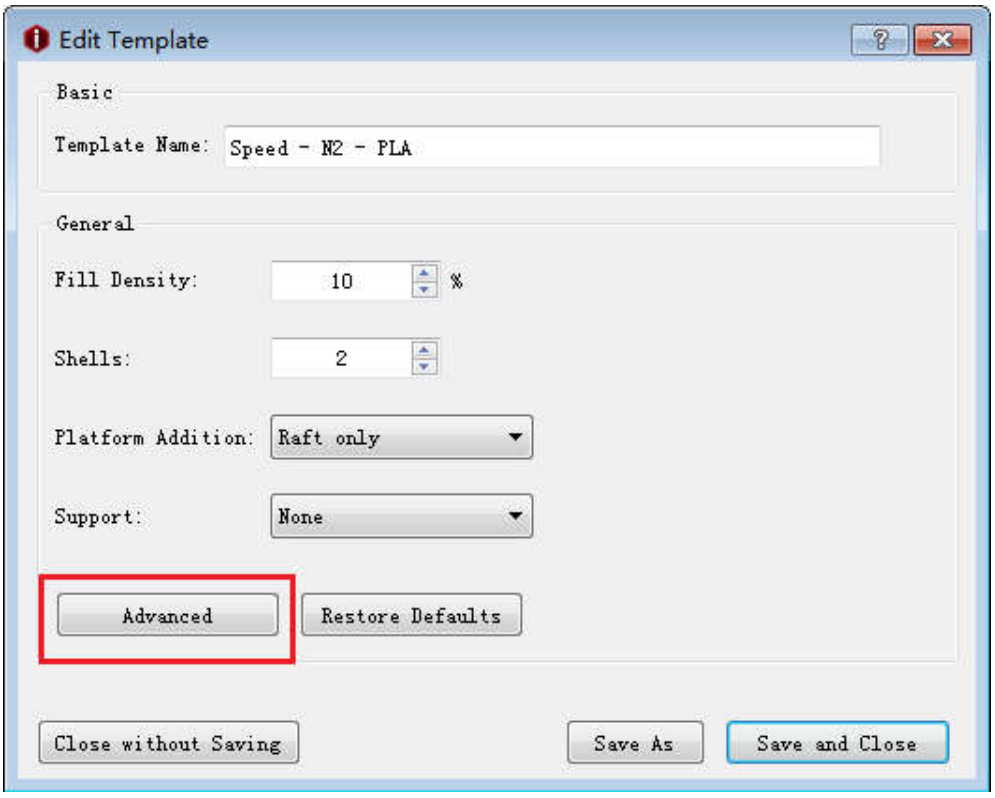


Figure 4.43: There are more settings in Advanced.

4.2.1 Layer

General:

Layer Height refers to the thickness of every single layer. Someone call it resolution as well.

Extrusion width refers to width of extruded line. The default extrusion width will be the same as origin 0.4mm nozzle's diameter. If you have changed your nozzle to be other size in ***Printer Settings***, please remember also to edit here.

Speed:

Default Printing Speed refers to the speed of printing non-specified area.

Inner Shell Speed refers to the speed of printing the model's inner shell.

Outer Shell Speed refers to the speed of printing the model's outer shell.

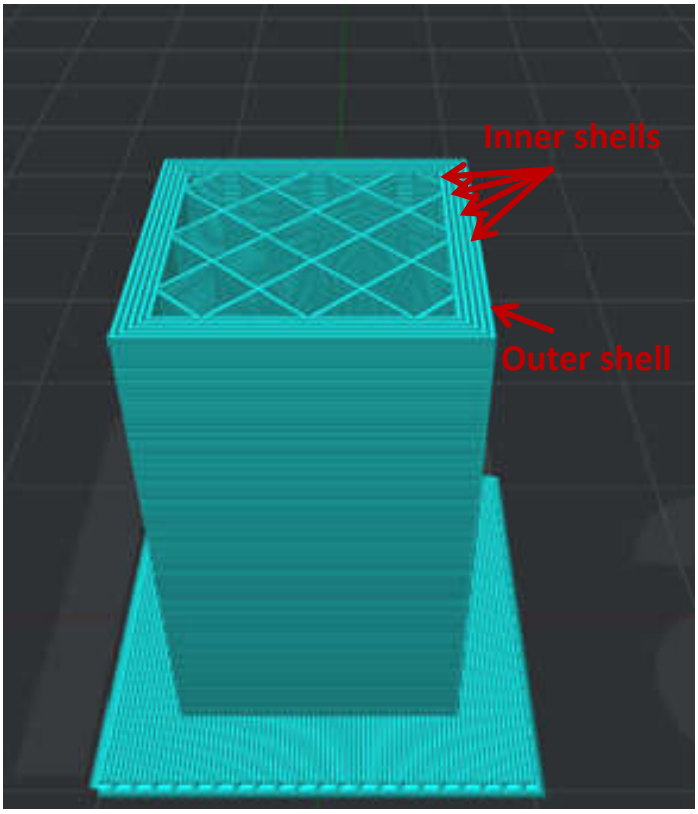


Figure 4.44: Only the wall at outer edge will be the outer shell. No matter how many shells have you set, expect the one outer, the remaining will be inner shells.

X/Y Axis Movement Speed refers to the speed that the nozzle moving at to another place without printing in X and Y directions.

Z Axis Movement Speed refers to the speed that the build plate moving at without printing in Z direction.

Skirt and Brim:

Skirt/Brim extruderrefers to choosing which extruder to print skirt or brim. (if you set number of extruder to 1,then it will only show ***Primary Extruder***; if you set it to 2，then it will show ***Left Extruder*** *and you can switch to* ***Right Extruder***)

Force Addition of Skirt refers to enabling a skirt.

Skirt Loop Lines refers to the amount of loops which are drawn around the object at the first layer which helps to prime your extruder.

Offset Distance refers to the distance between the loop line and the first layer.

Brim Loop Lines refers to a line drawn close to the object at the first layer which helps to prime your extruder.

First Layer Settings:

First Layer Height refers to the thickness of the model's first layer.

First Layer Flowrate refers to a dimensionless value that controls the amount of extrusion.

First Layer Speed refers to the speed of printing the model's first layer.

Restore Defaults refers to going all the settings back to default.

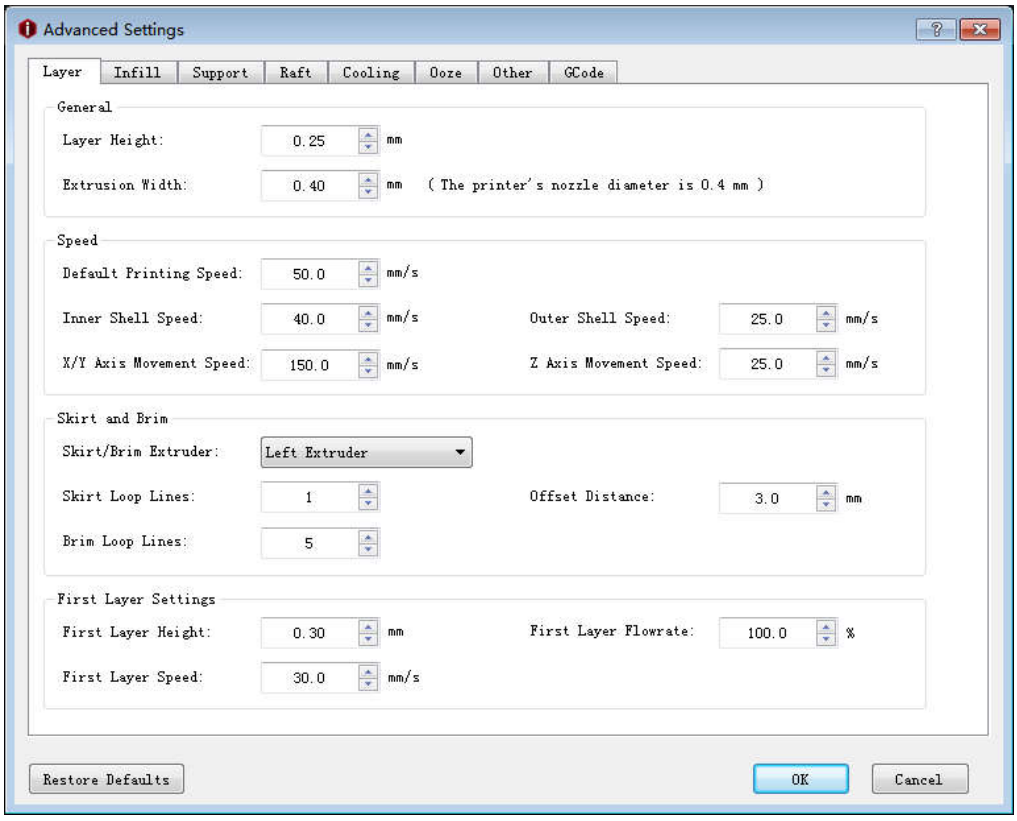


Figure 4.45: The Layer tab.

4.2.2 Infill

Infill:

Infill Extruder refers to choosing which extruder to print infill. (If you set number of extruder to 1, then it will only show ***Primary Extruder***; if you set it to 2, then it will show three functions :***AllExtruder***, ***Left Extruder and Right Extruder.*** The ***All Extruder*** refers to the extruder is same asthe model's).

Infill Speed refers to the speed of printing the model's infill structure.

Infill Overlap refers to the amount of overlap between the infill and the shell.

Infill Flowrate refers to the amount of material extruded which will be multiplied by this value while printing infill. ***Flowrate*** refers to a dimensionless value that controls the amount of extrusion. 100% equals to default amount.

Infill Pattern Type refers to infill structure.

Grid refers to the mesh structure.

Rectilinear refers to the infill structure with a continuous moving path. No moving direction changes during printing means that there will be no risk of nozzle clogs from retraction.

Figure 4.46: Choose an infill pattern depending on the kind of the model.

Top and Down Solid Part:

Bottom Solid Fill Layers refers to the amount of solid bottom layers.

Top Solid Fill Layers refers to the amount of solid top layers.

Bottom Solid Fill Speed refers to the speed of printing solid bottom layers.

Top Solid Fill Speed refers to the speed of printing solid top layers.

Bottom Solid Fill Flowraterefers to the infill filament flowrate of the bottom layer.

Top Solid Fill Flowrate refers to the infill filament flowrate of the top layer.

Solid Fill Pattern Type refers to the fill structure of solid layers.

Lines refer to the segments at the same direction.

Rectilinear refers to the fill structure with a continuous moving path. No moving direction changes during printing means that there will be no risk of nozzle clogs from retraction.

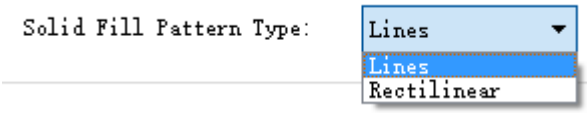


Figure 4.47: Choose a solid fill pattern depending on the kind of the model.

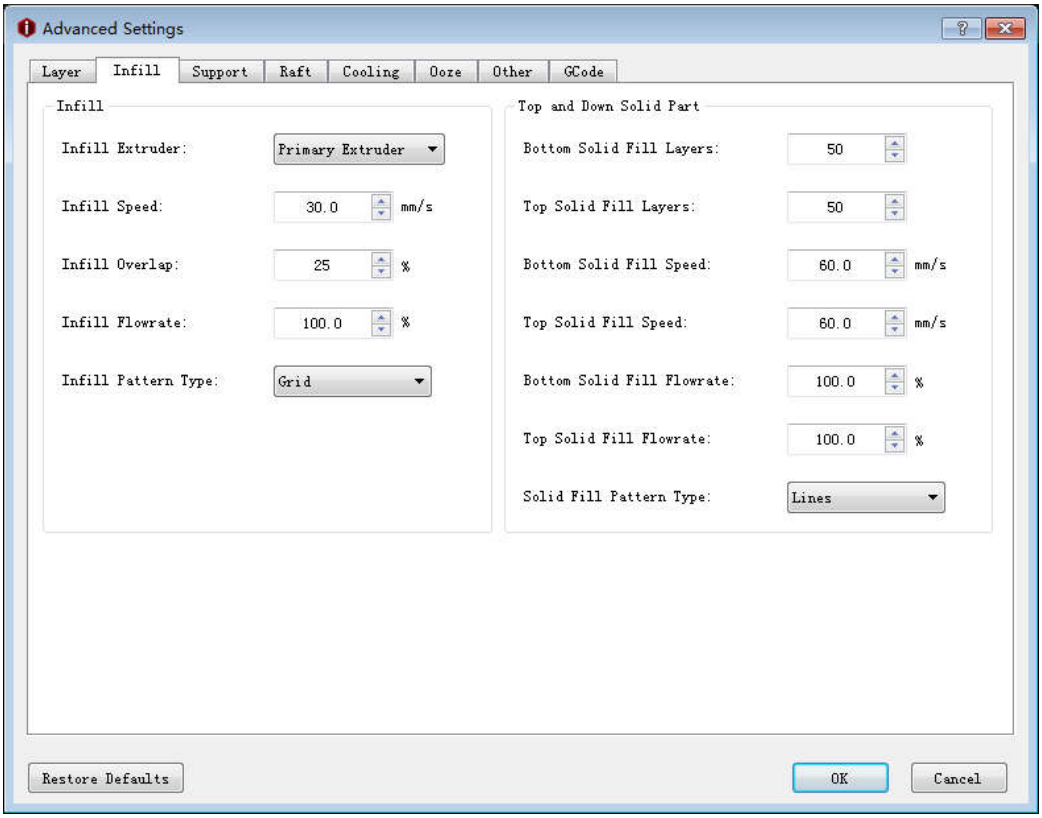


Figure 4.48: The Infill tab.

4.2.3 Support

Support Extruder refers to choosing which extruder to print support. (If you set number of extruder to 1, then it will only show ***Primary Extruder***; if you set it to 2, then it will show ***Left Extruder*** *and you can switch to* ***Right Extruder***).

Support:

Support Type refers to the structure of support.

Normal refers to the support which is computed based on locality which may cause random hang.

Pillar refers to the support which is computed by pillars based on entirety.

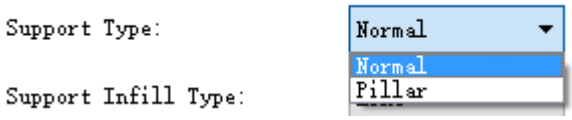


Figure 4.49: Two types support structure.

Support Infill Type refers to the infill structure of support.

Lines refer to the segments at the same direction.

Rectilinear refers to the fill structure with a continuous moving path. No moving direction changes during printing means that there will be no risk of nozzle clogs from retraction.

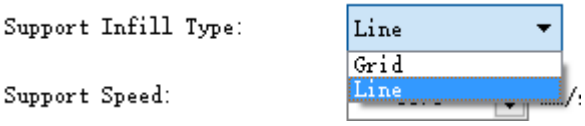


Figure 4.50: Two types support infill.

Support Speed refers to the speed of printing the model's support structure.

Infill Ratio refers to the density of infill structure of the support material.

Max Overhang Angle refers to the max overhanging angle which needs support to print. Overhang Angle refers to the angle between the overhang surface and Z axis.

Horizontal Offset refers to the distance of the support material from the print in the horizontal directions.

Vertical offsetTop Layersrefers to the distance of the support material from the top in the vertical directions.

Vertical offset Down Layersrefers to the distance of the support material from the bottom in the vertical directions.

Support Flowrate refers to the amount of material extruded which will be multiplied by this value while printing support.

Pillar Size refers to the size of support each support pillar.

Sparse Connection refers to disabling retraction from one support to another which will cause strings between supports which add connections between supports.

Dense Support:

Dense Support Layers refers to the amount of layers of dense support layer. It will only exist in the layers to approaching the model surface which can make the support connecting points smoother after removing the supports.

Dense Support Infill Ratiorefers to the density of the material using for printing the dense support infill.

Dense Support Infill Type refers to the type of dense support structure which is divided by ***Grid*** and ***Line***.

Grid type is more solid to be the basement. ***Line*** type is easier to peel.

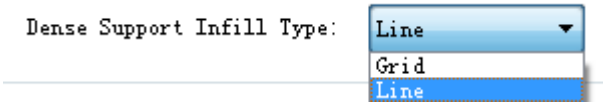


Figure 4.51: Two types of dense support infill.

Interlaced Support Infill Angle: You can define the direction of each layer of the support which can decrease the potential of hangs in corners due to all the supports at the same direction. If you add value 1, value 2 and value 3, the angle of the first layer of support will be value1. Next layer will be value 2. Next will be value 3. Next will back to value 1 and change by the order.

Angle refers to the direction of support.

Add Infill Angle refers to adding a value.

Remove Infill Angle refers to removing the selected value.

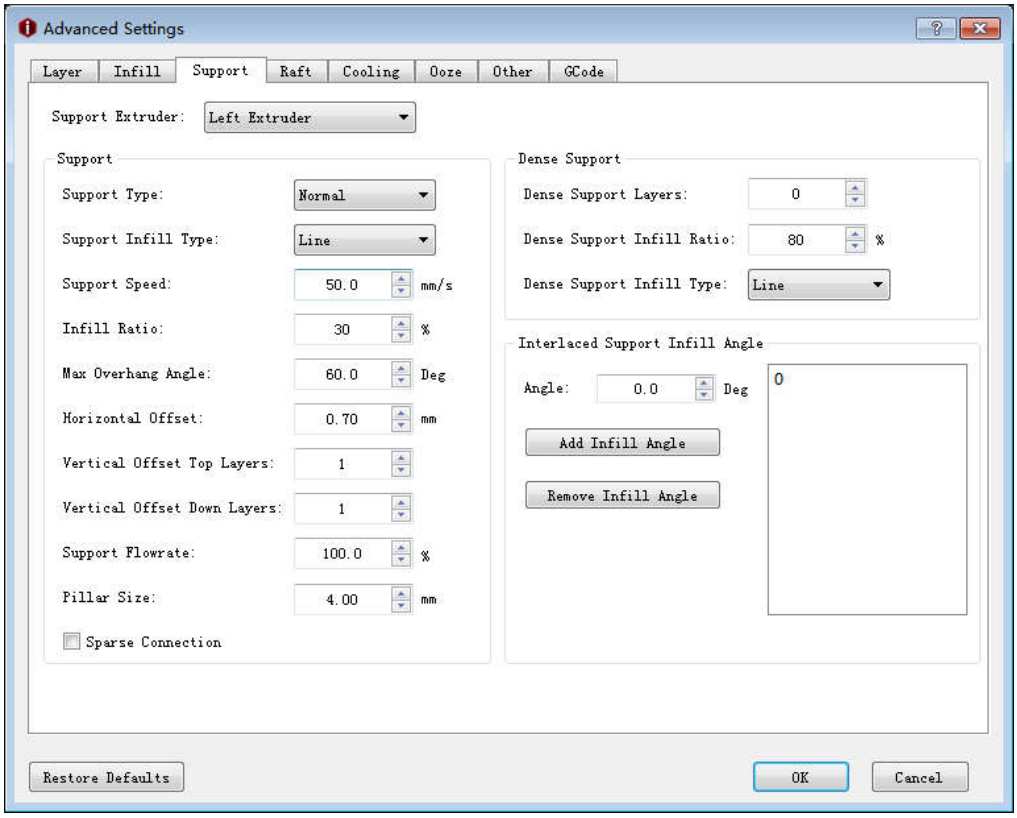


Figure 4.52: The Support tab.

4.2.4 Raft

Raft Extruder refers to choosing which extruder to print raft. (if you set number of extruder to 1，then it will only show ***Primary Extruder***;if you set it to 2，then it will show ***Left Extruder*** *and you can switch to* ***Right Extruder***)

Raft:

Raft Offset refers to the extra raft area around the object which is also rafted.

Raft Gap from Model refers to the gap between the last layer of the raft and the first main body layer.

First Layers refers to the amount of layers of the first layer.

Surface Layers refers to the amount of surface layers put on top of the raft, these are fully filled layers.

First Layer Speed refers to the print speed of the first layer.

Interface Layer Speed refers to the print speed of the interface layer.

Surface Layer Speed refers to the print speed of the raft's surface.

Surface Layer Infill Ratiorefers to the density of the material using for printing the surface layer infill structure.

Raft Lines Type refers to the fill structure of Raft.

Lines refer to the segments at the same direction.

Rectilinear refers to the fill structure with a continuous moving path. No moving direction changes during printing means no risk of nozzle clogs from retraction.

Figure 4.53: Two types of Raft.

Keep Holes in raft structure refers to generating a Raft according to shape of your model. With being unchecked, there will generate a whole piece of ***Raft*** without holes inside in case overlapping lines between different printing sections on ***Raft***.

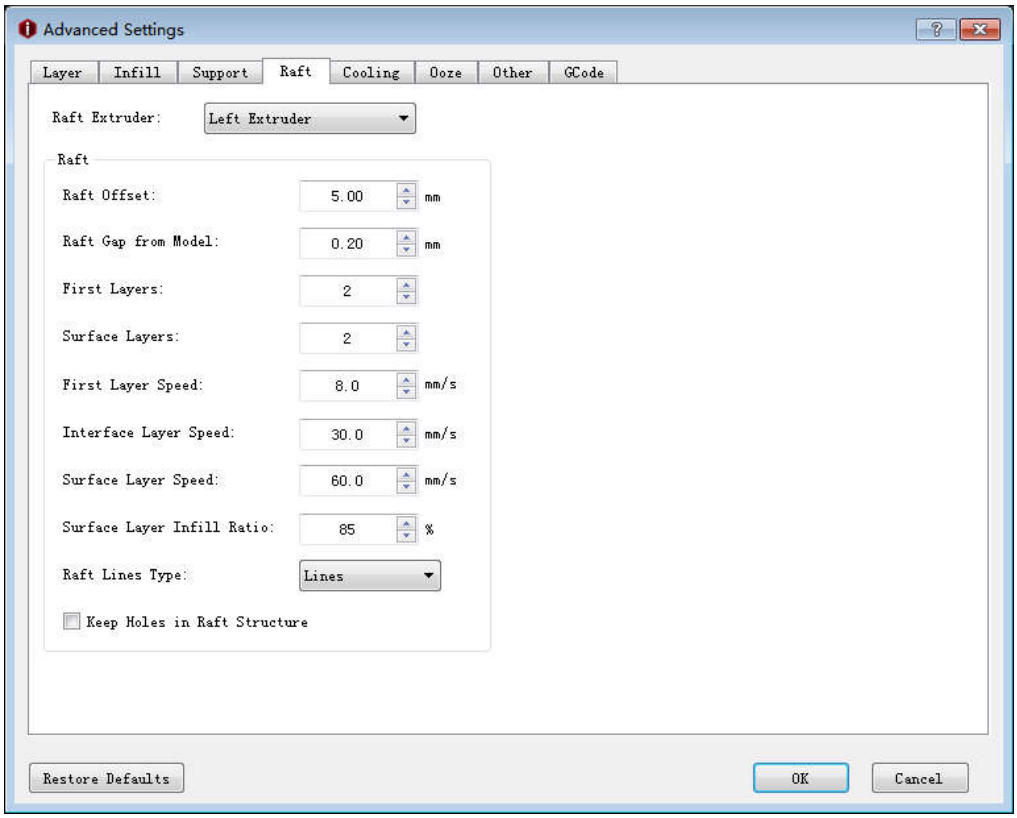


Figure 4.54: The Raft tab.

4.2.5 Cooling

Cooling:

Minimal Layer Print Time refers to the minimum time spending in a layer, which gives the layer time to cool down before the next layer is put on top.

Default Fan Speed refers to the speed of the fan in the unspecified condition.

Maximum Fan Speed means the maximum speed of the extra cooling fan. If the cooling setting slows down the layer, the fan is adjusted between the min and max speed. Maximal fan speed is used if the layer is slowed down due to cooling setting by more than 200%.

Minimal Printing Speed refers to the minimal speed when the layer printing is slowed down due to cooling settings.

Temperature:

Bed Temperature refers to the temperature of the bed when printing.

Primary Extruder refers to the temperature of the primary extruder when printing. We default set the left extruder as the primary extruder when you choose ***Extruder Count*** as 1 in ***Printer Settings***.

Left Extruder refers to the manual control temperature of the left extruder.

Right Extruder refers to the manual control temperature of the right extrude.

The ***Left Extruder*** and ***Right Extruder*** setting can only be seen after choosing ***Extruder Count*** as 2 in ***Printer Settings***.

Fan Control: Only the downward model fan on single extruder printer is controllable. The side extruder cooling fan is constant on. We don’t suggest you to switch the extruder cooling fan to be controllable which may cause filament in throat tube be soften before get into the hot zone due to heat.

Layer refers to the layer at which the fan is turn on.

Fan Speed refers speed of the extra cooling fan.

You can add the specific layer where you want to change the fan speed at by clicking the ***Add Fan Point*** button and input the speed in right box. And delete it by choosing the value in right box and clicking the ***Remove Fan Point*** button.

Temperature:

Bed Temperature: refers temperature of build plate.

Primary Extruder refers to the temperature of the primary extruder when printing. We default set the left extruder as the primary extruder when you choose ***Extruder Count*** as 1 in ***Printer Settings***.

Left Extruder refers to the manual control temperature of the left extruder.

Right Extruder refers to the manual control temperature of the right extrude.

The ***Left Extruder*** and ***Right Extruder*** setting can only be seen after choosing ***Extruder Count*** as 2 in ***Printer Settings***.

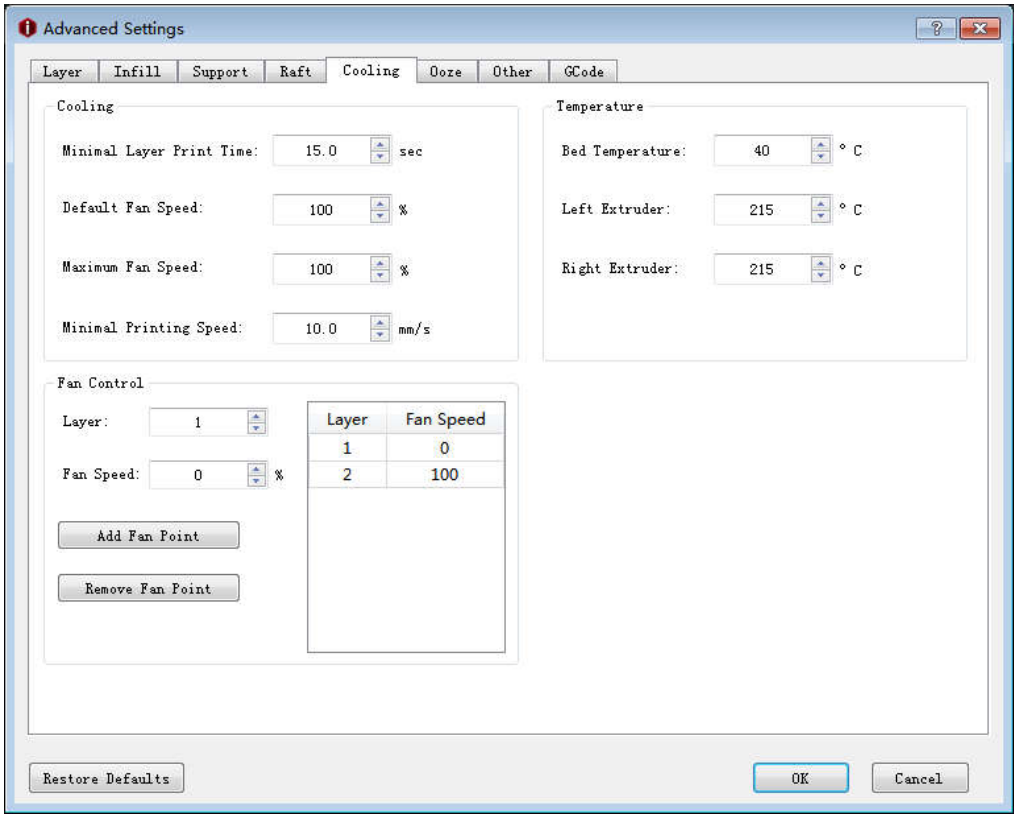


Figure 4.55: The Cooling tab.

4.2.6 Ooze

Retract:

Enable Retraction refers to enabling retract filament when the nozzle travels to another print point.

Retract Speed refers to the speed which the filament is retracted at, a higher retraction speed works better. But a very high retraction speed can lead to filament grinding.

Restart Speed refers to the speed when the filament is pushed in before continuing the extrusion.

Retract Material Amount refers to the amount of retraction. Set at 0 means that there is no retraction at all.

Z hop refers to the distance the nozzle travels on z direction after retraction before moving the next print point.

Minimal Travel of Retraction refers to the minimum amount of travel needed for a retraction to happen at all. Set this item to make sure you do not get a lot of retractions in a small area.

Force Retract On Layer Change means when you enable it, the printer will automatically do retract when start a new layer.

Minimal Amount of Retraction refers to the minimal amount of extrusion that needs to be done before retracting again if a retraction needs to be happen before this minimal is reached the retraction is ignored. This avoids retracting a lot on the same piece of filament which flattens the filament and causes grinding issues.

Avoid Traveling Through Holes means when you enable it, the nozzles will evade moving above the empty print section to improve the print quality of outer surfaces.

Extra Restart Amount refers to the amount of extrusion compensation before continuing the extrusion.

Avoid Retract Inside Models means when you enable it, when printing the infill or the area between shells, the retracts will be automatically stopped. Less retract, less risk of nozzle clogging.

Multiple Extruders Ooze Control:This section can only be seen after choosing ***Extruder Count*** as 2 in ***Printer Settings***.

Enable WipeWall means when printing a model with dual-extrusion, a nozzle will print a thin WipeWall around the model. This WipeWall will help to clean the nozzle while printing.

Retract Speed of Extruder-switch refers to the speed which the filament is retracted at when switching nozzle between dual-extrusion.

Retract Amount of Extruder-switch refers to the amount of retraction when switching nozzle between dual-extrusion. 0 refers to no retraction at all.

Restart Speed of Extruder-switch refers to the speed when the filament is pushed in after switching nozzle between dual-extrusion.

Extra Restart Amount of extruder-switch refers to the amount of retraction will add when switch extruder.

WipeWall Offset refers to the distance between WipeWall and model.

WipeWall Angle refers to the largest angle the WipeWall will generate according to shape of your model (from 0 to 90).

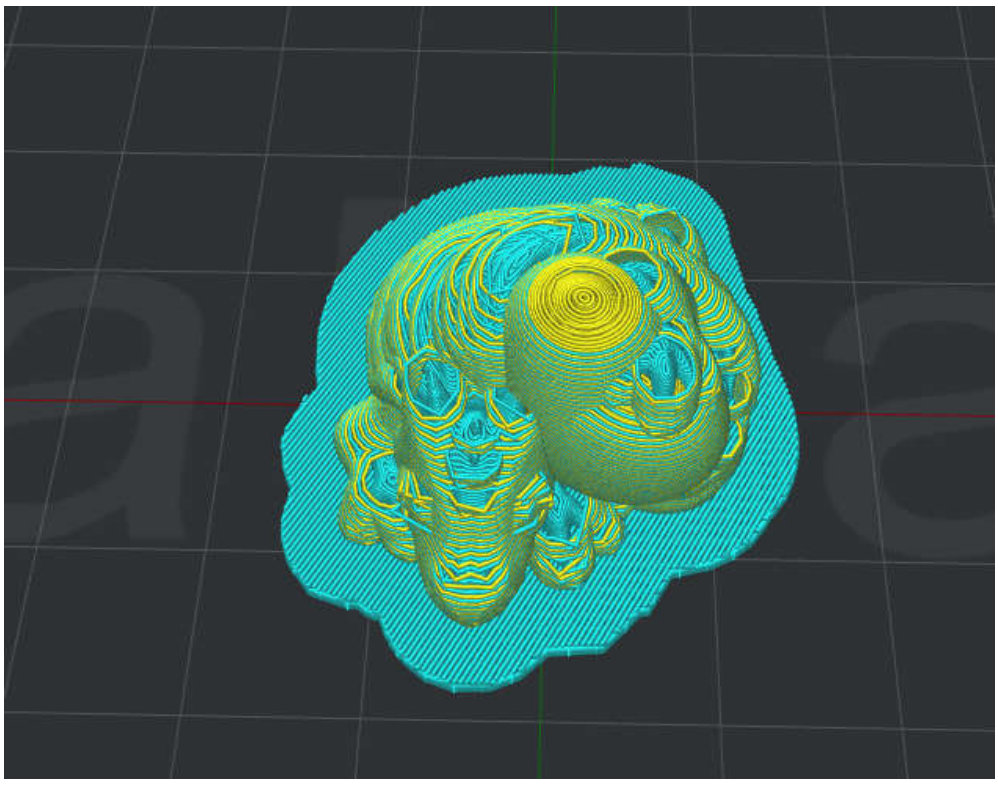


Figure 4.56: Set Angle of WipeWall to be 90.

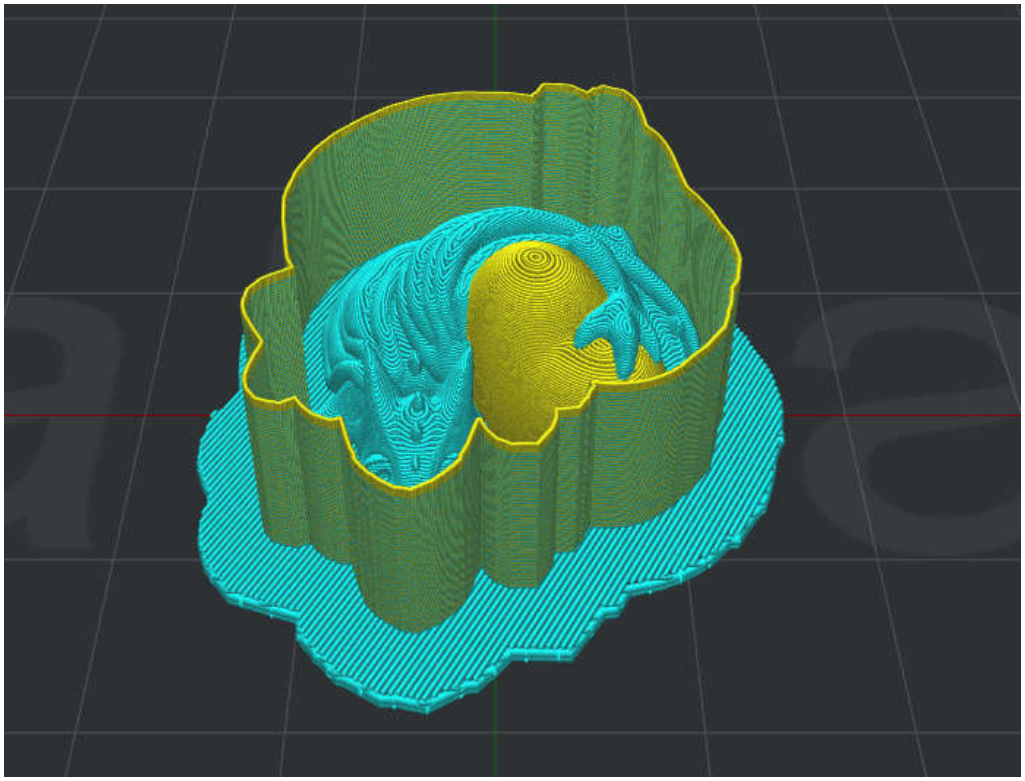


Figure 4.57:*Set Angle of WipeWall to be 0.*

WipeWall Type refers to the type of WipeWall. The primary difference is the distance of model and WipeWall.

Contoured is the nearest by model. **Vertical** is the farthest.

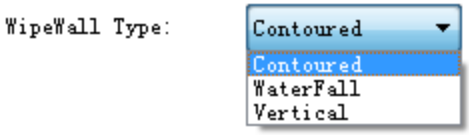


Figure 4.58: Three type of WipeWall.

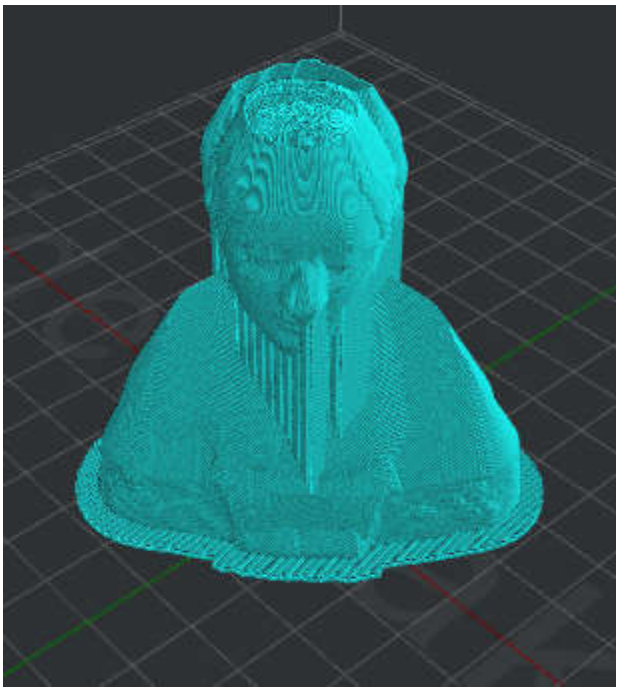


Figure 4.59: Set type of WipeWall to be "Contoured".

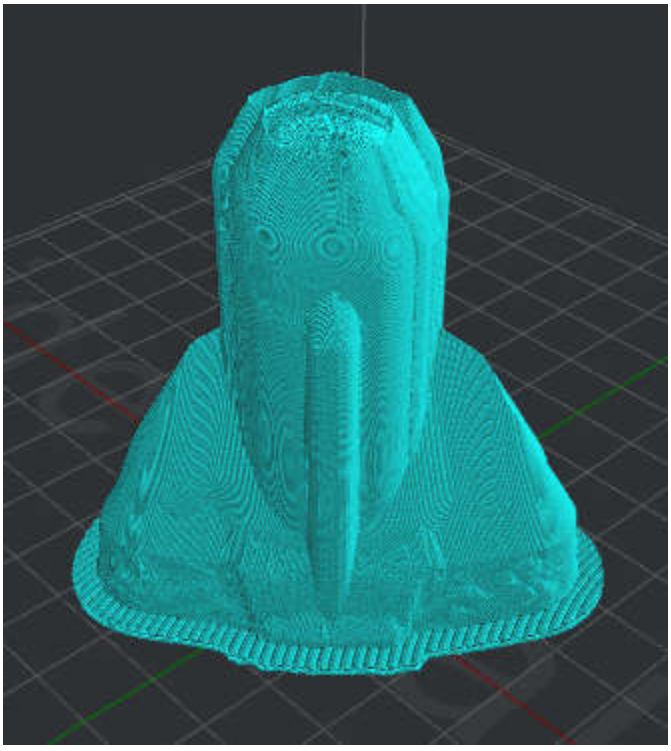


Figure 4.60: Set type of WipeWall to be "WaterFall".

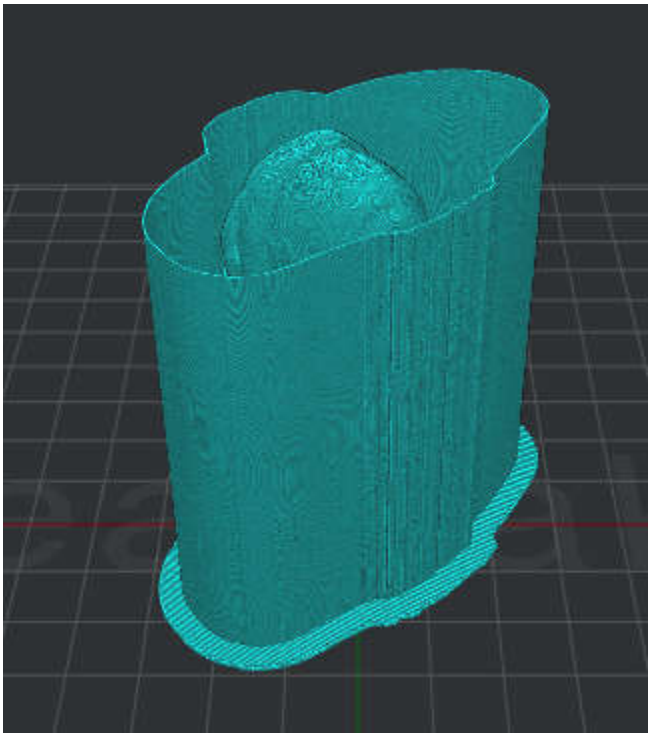


Figure 4.61: Set type of WipeWall to be "Vertical".

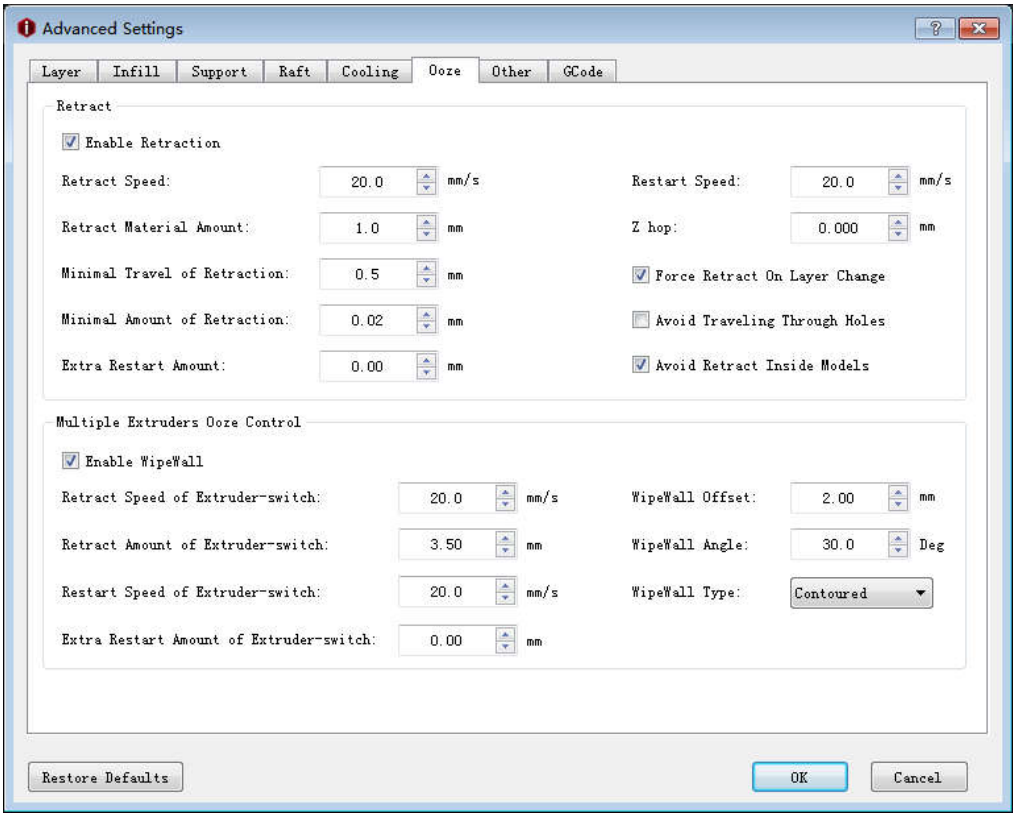


Figure 4.62: The Ooze tab.

4.2.7 Other

Spiral Vase Modemeans that the model will be printed depending on its outline and only one single shell will be printed in every layer. This mode will transfer the model to vase-like structure with only outer shell, no infill and open top surface. Z axis will move slowly when one layer is close to finishing.

Merge Open Segments of Model Parts means that the ideaMaker will fix the open segments for the model.

Merge Internal Overlapping Parts means that the ideaMaker will merge the parts which overlap in internal of the model.

Print External Shells First refers to printing the outer shell then printing the inner shell.

Check Thin Wall (Single Extrusion Width) means that ideaMaker will check the model's thickness while slicing it, and ignore the little details which are thinner than the setting.

Thin Wall Width refers to the width of thin wall check threshold.

Layer Start Point Type refers to the position of starting point of each layer.

Nearest refers to the optimal moving path of the print head.

Fixed refers to a customer position. If the customer position is not available in some layer, ideaMaker will find a closest position to instead.

Random refers to separate position.

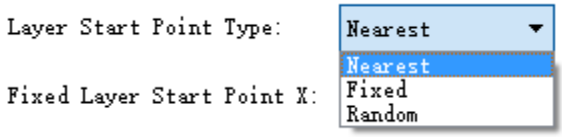


Figure 4.63: There are 3 types of Layer Start Point. If you choose Fixed as your start point type, you can set X & Y's value to define a position. ideaMaker will set a closest position based on the position you set.

Fixed Layer Start Point X refers to defining the X position of fixed start point.

Fixed Layer Start Point Y refers to defining the Y position of fixed start point.

Pause at Height means that you can set a height and the printer will automatically pause at the defined height. You can do several operations such as changing filament, and resume the print then.

You can define the height by clicking the ***Add Pause Point*** button. And delete the defined height by choosing the height in right box and clicking the ***Remove Pause Point*** button.

Note: This height doesn't include ***Raft***.

Global Offset means that you can set offset to the location of your model.

You can separately set ***X-Offset***, ***Y-Offset*** and ***Z-Offset***.

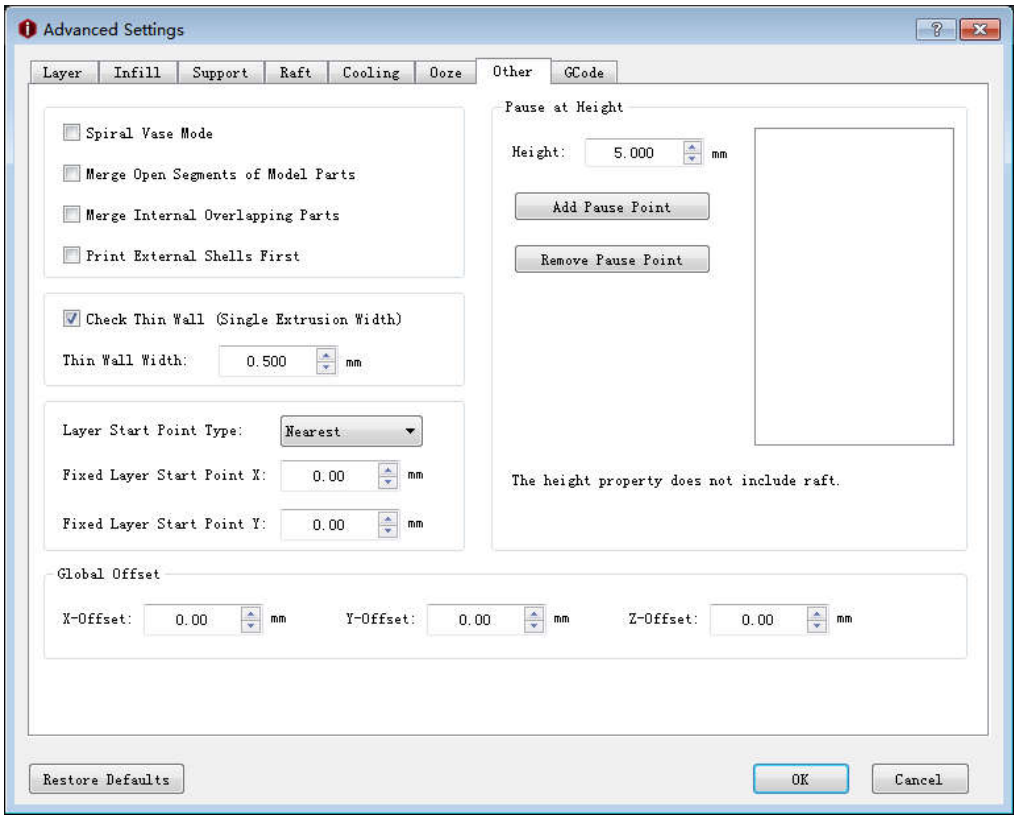


Figure 4.64: The Other tab

4.2.8 GCode

Start GCode refers to the program that printer will do or set before printing the model. ***End GCode*** refers to the program that printer will do or set after finish printing the model. Select the extruder you want to check by choosing ***Number of Extruders***. ***1*** refers to only one extruder. ***2*** refers to two extruders.

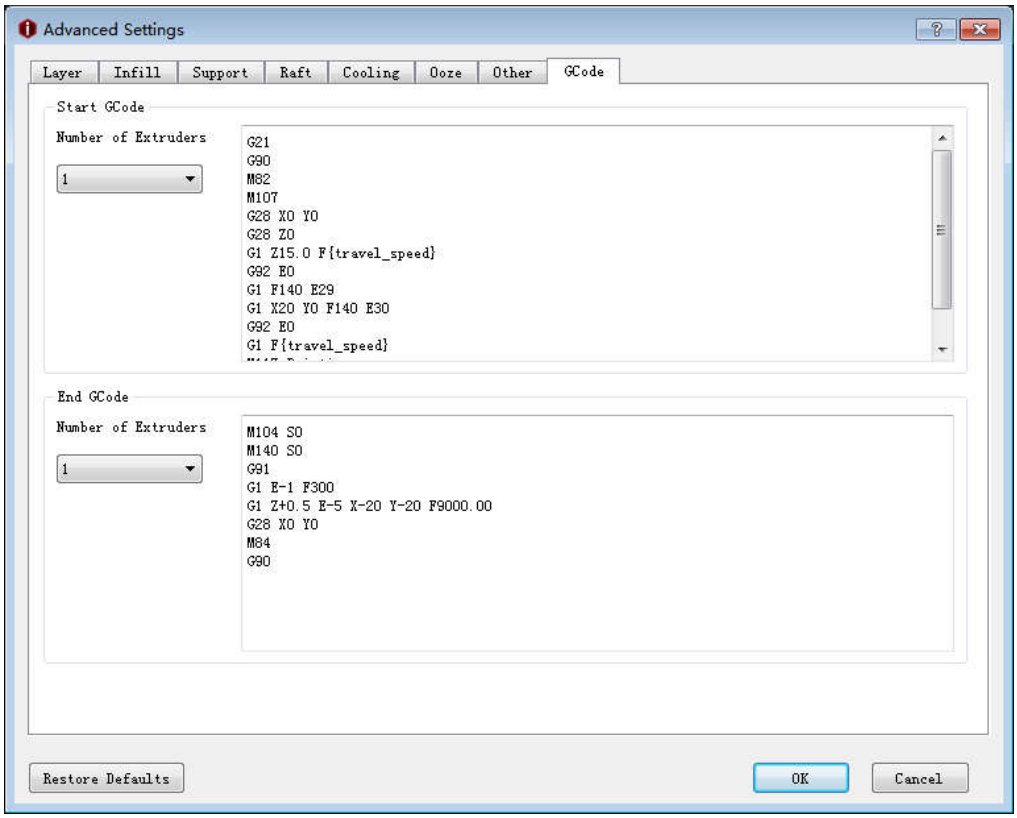


Figure 4.65: The GCode tab.

4.3 Multiple Extruders

When you want to print model by multiple extruders, you need to know something below:

1. Set printer setting to multiple-extruder-mode.

Printer->Printer Settings->Multiple Extruders

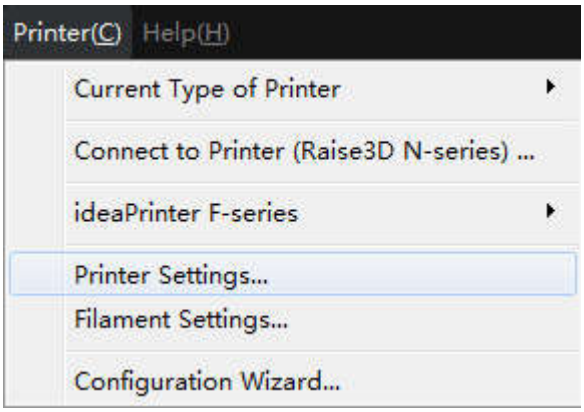


Figure 4.66: Choose "Printer" ->"Printer Settings" from menu bar.

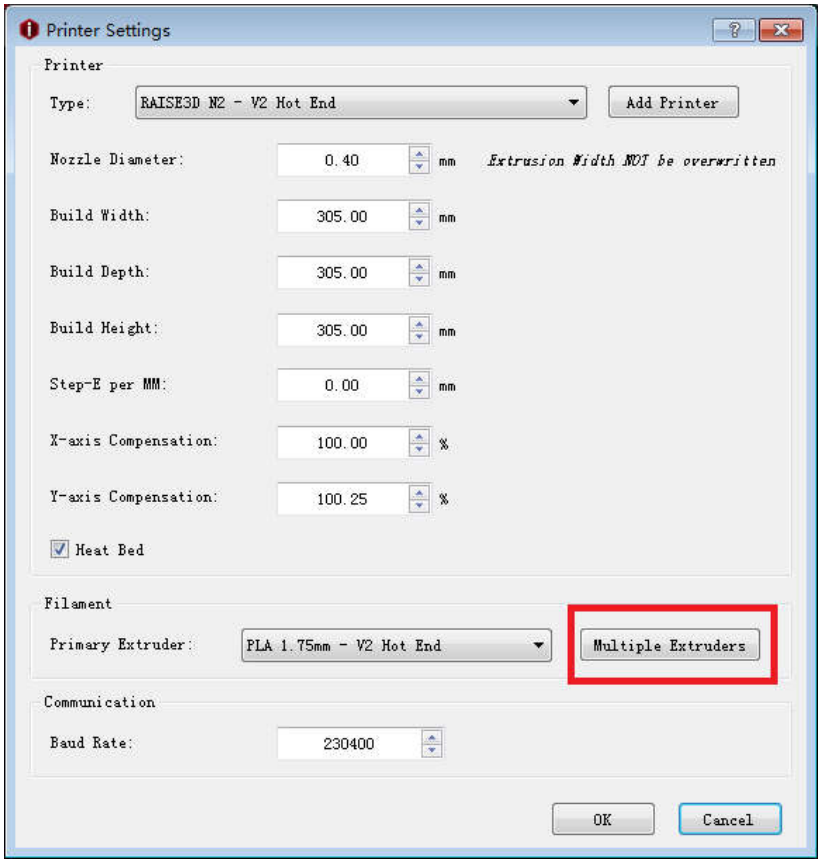


Figure 4.67: Click "Multiple Extruders".

Select 2 in ***Extruder Count***, then assign filament to each nozzle in ***Filament***.

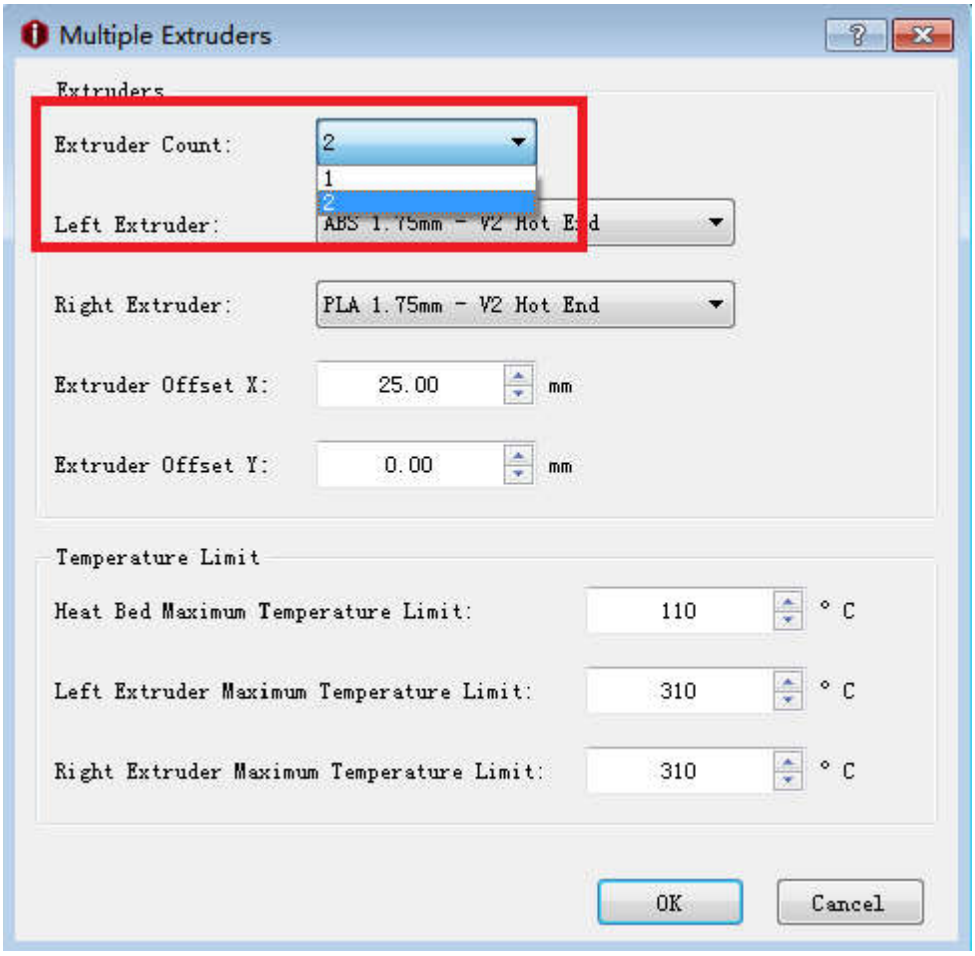


Figure 4.68: Set the "Extruder Count" to be 2. And set filament type for each extruder.

Extruder offset X refers to the offset between two nozzles in the direction of X.

Extruder offset Y refers to the offset between two nozzles in the direction of Y.

2. If you want to print a multiple-filament model, you need to arrange which extruder should print which part of the model, and then assemble the model.

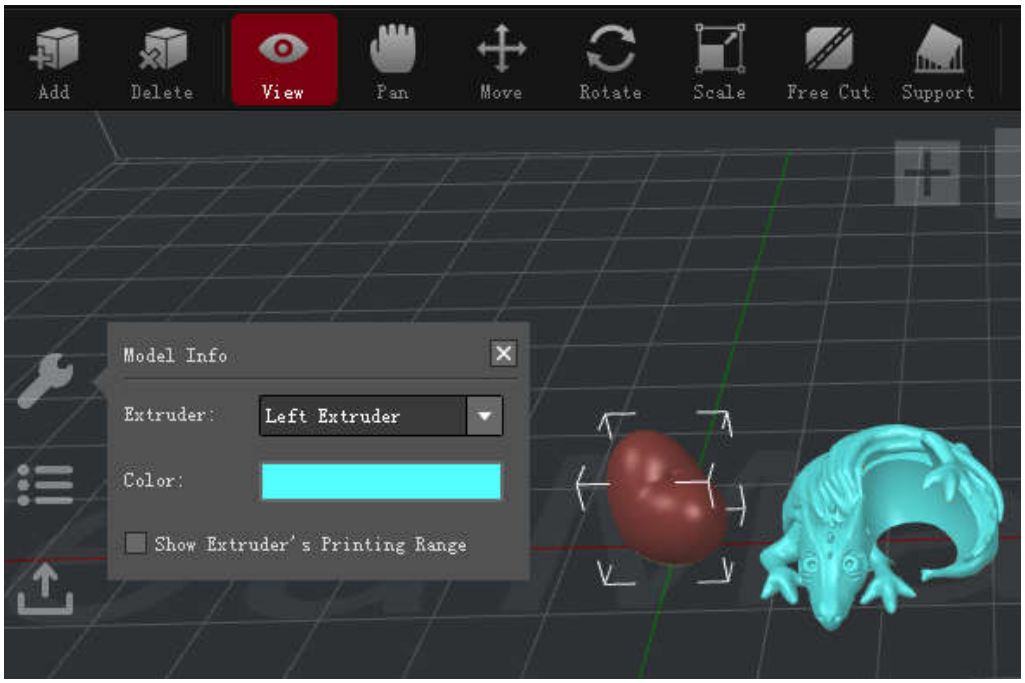


Figure 4.69: Choose the different extruders to print different parts of the model.

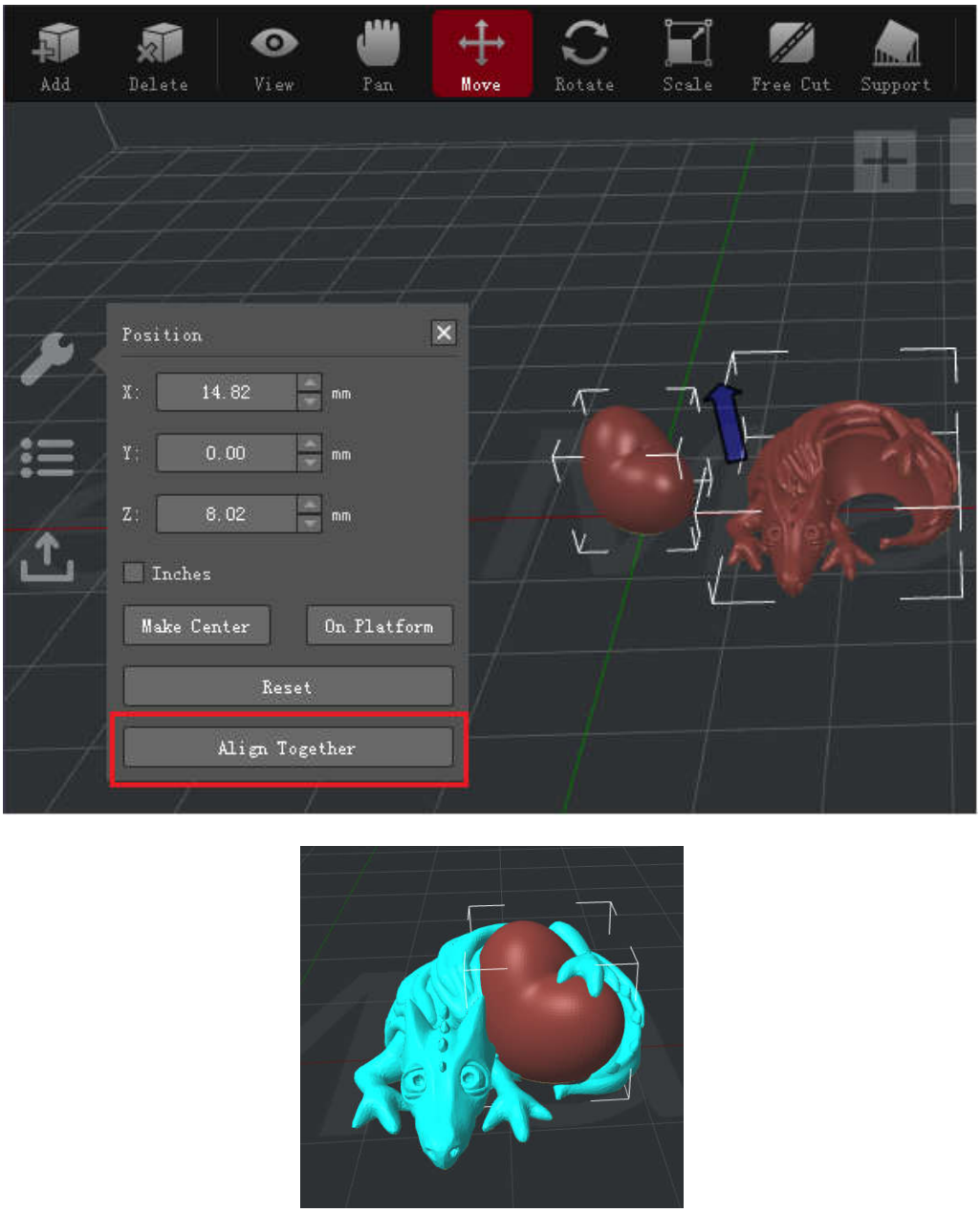


Figure 4.70: Try "Align Together" to align the models with "Move" enabled. If it doesn't work well, try move the models manually.

You can enable ***Show Extruder's Printing Range*** to check whether your model is at correct range for each extruder when moving the models.

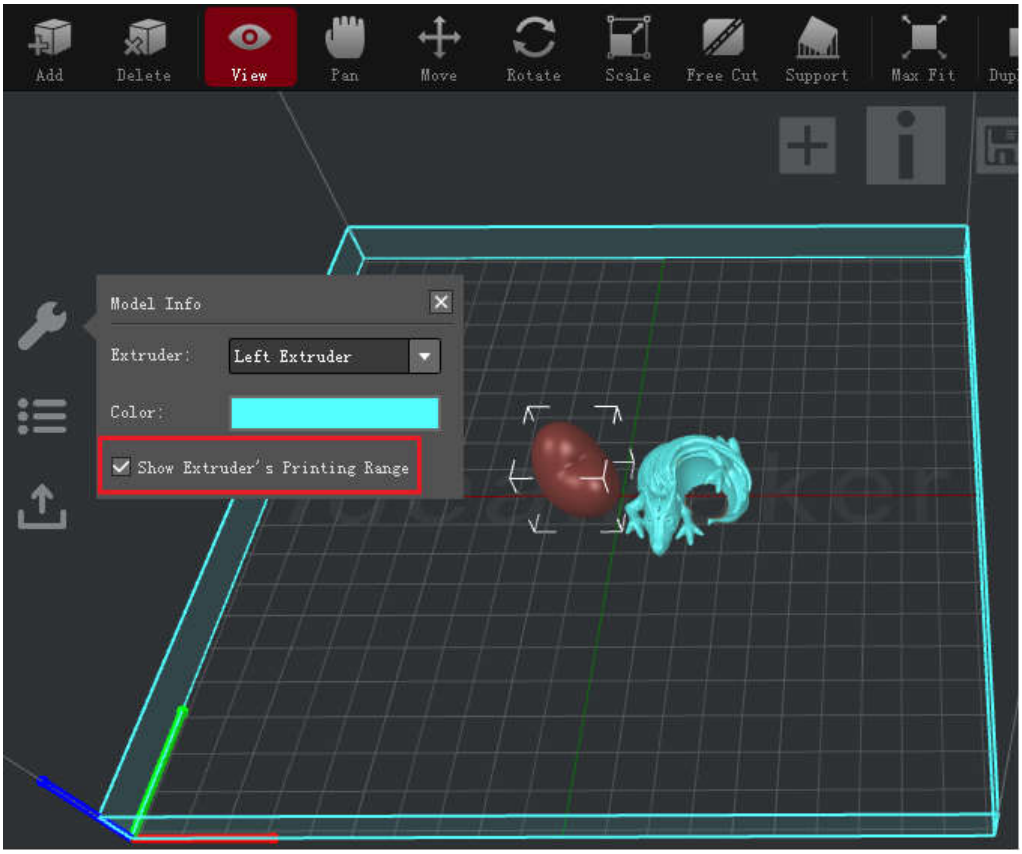


Figure 4.71: Check Right Extruder's printing range.

3. When slicing the model, if you want to use specific filament to print support and raft (such as soluble filament), you need to setthe extruder for printing support and raft.

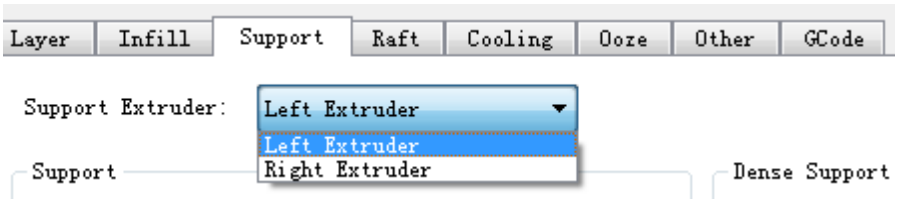


Figure 4.72: Choose an extruder to print support.

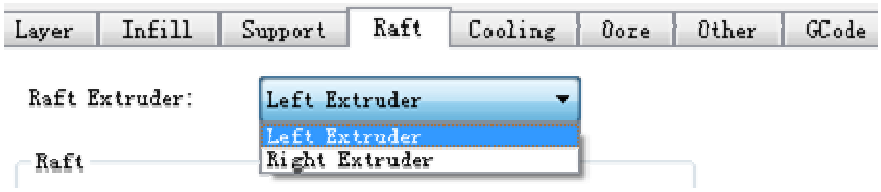


Figure 4.73: Choose an extruder to print raft.

4. If you use different filament while printing in multiple-extruder-mode, you need to set the extruders' temperature separately. We usually suggest set the right nozzle a little higher than the left nozzle (if you are printing the same filament in left and right) due to the much longer loading path of right extruder）.

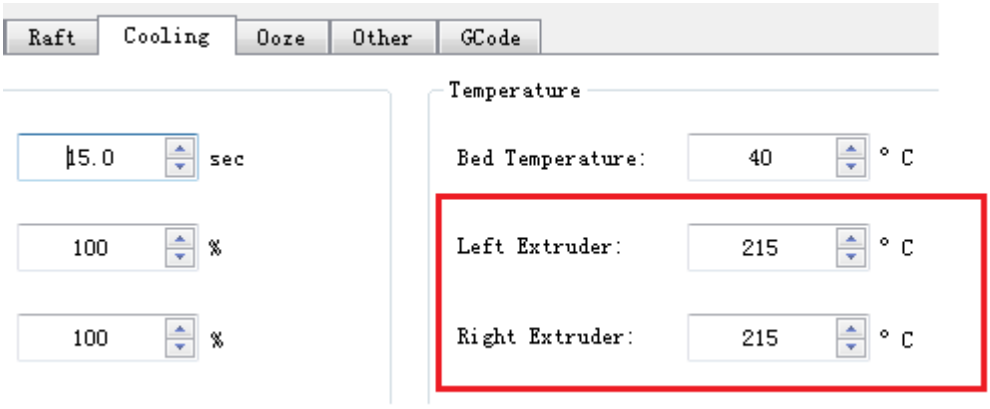


Figure 4.74: Check whether the temperature is suitable for your filament.

5. In order to print a better model that printed by multiple extruders, you also need to control multiple extruders' ooze.

These settings as follows:

Multiple Extruders Ooze Control:

Enable WipeWall means when printing a model with dual-extrusion, a nozzle will print a thin WipeWall around the model. This WipeWall will help to clean the nozzle while printing.

Retract Speed of Extruder-switch refers to the speed which the filament is retracted at when switching nozzle between dual-extrusion.

Retract Amount of extruder-switch refers to the amount of retraction when switching nozzle between dual-extrusion. 0 refers to no retraction at all.

Extra Restart Amount of extruder-switch refers to the amount of retraction will add when switch extruder.

The ***Multiple Extruders Ooze Control*** section can only be seen after choosing ***Extruder Count*** as 2 in ***Printer Settings***.

WipeWall Offset refers to the distance between WipeWall and model.

WipeWall Angle refers to the largest angle the WipeWall will generate according to shape of your model (from 0 to 90).

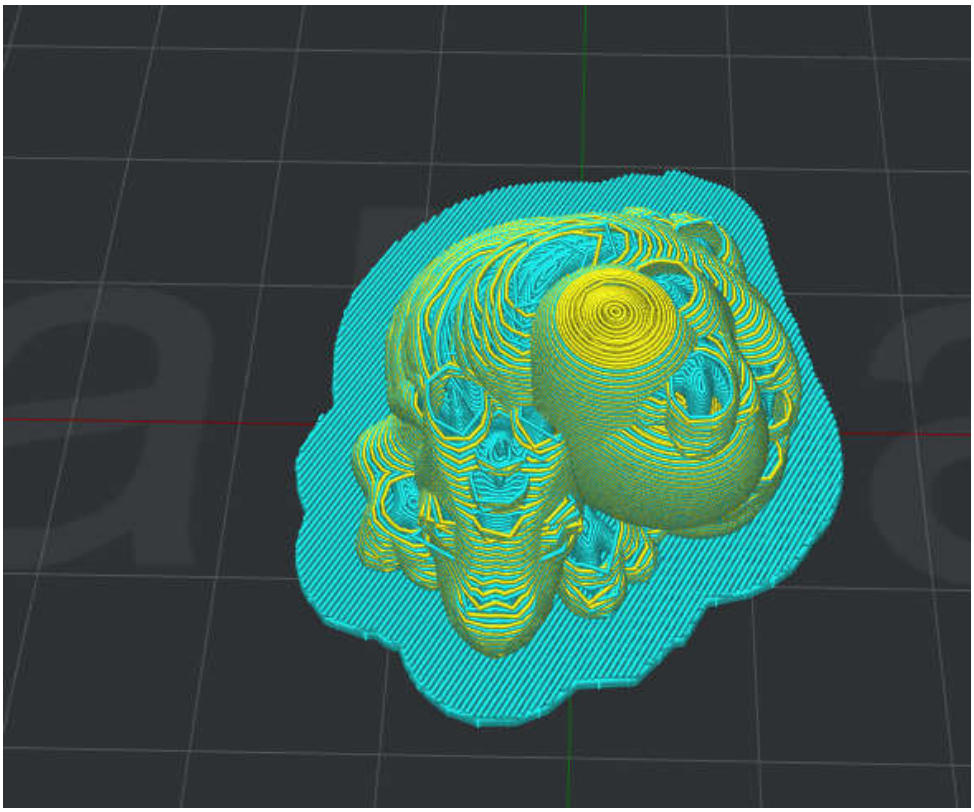


Figure 4.75: Set Angle of WipeWall to be 90.

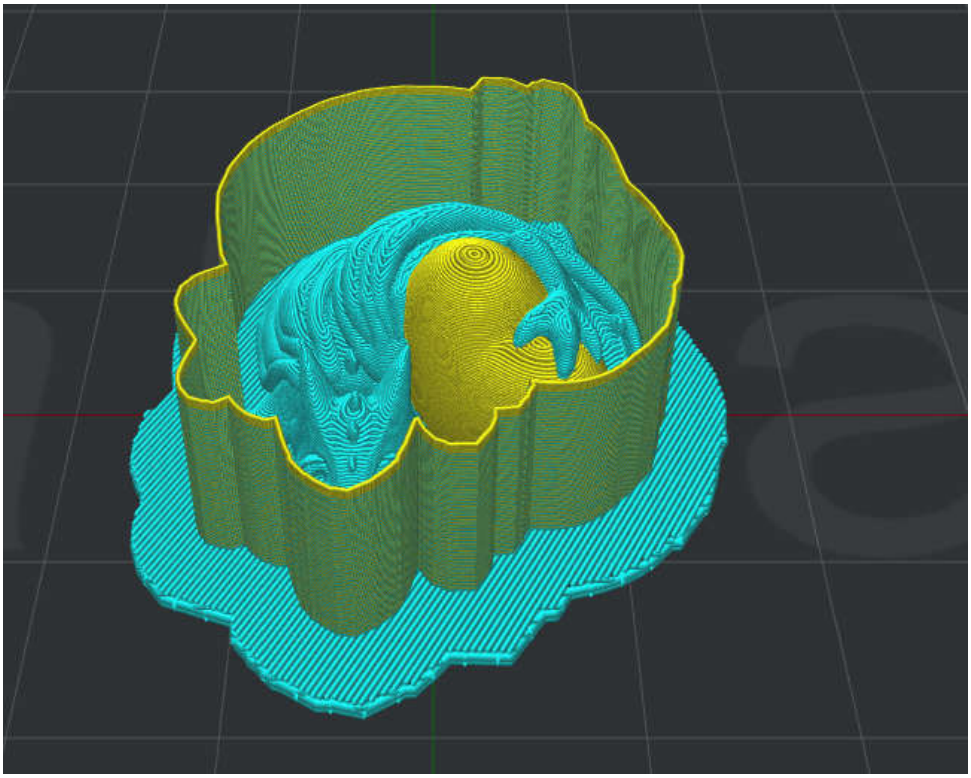


Figure 4.76:*Set Angle of WipeWall to be 0.*

WipeWall Type refers to the type of WipeWall. The primary difference is the distance of model and WipeWall.

Contoured is the nearest by model. **Vertical** is the farthest.

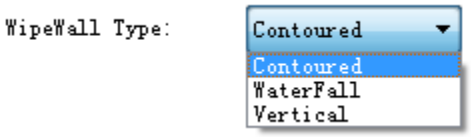


Figure 4.77: Three type of WipeWall.

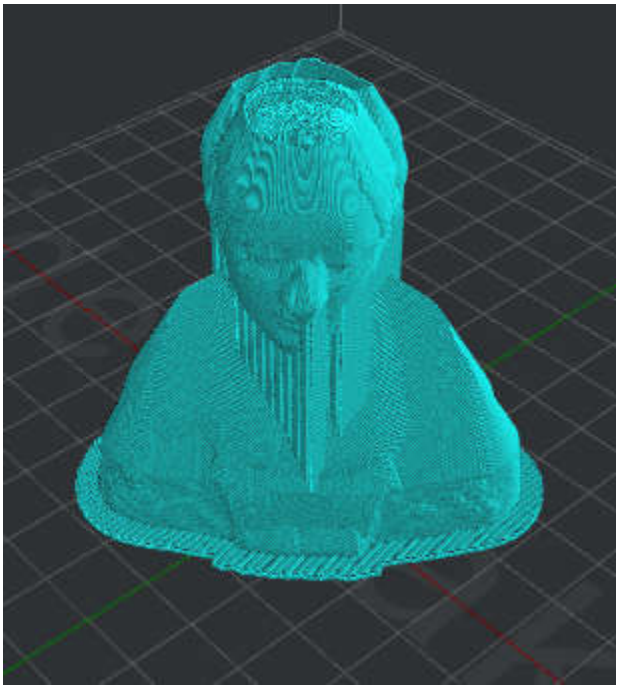


Figure 4.78: Set type of WipeWall to be "Contoured".

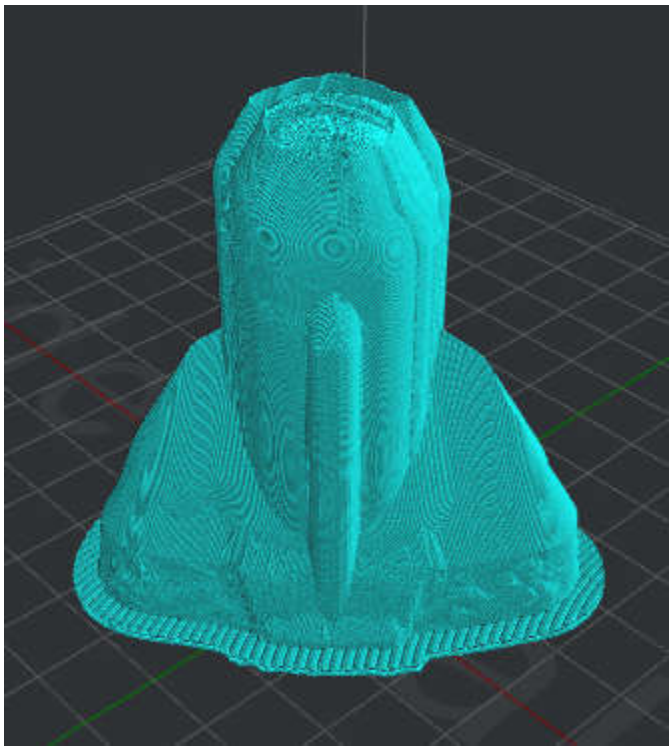


Figure 4.79: Set type of WipeWall to be "WaterFall".

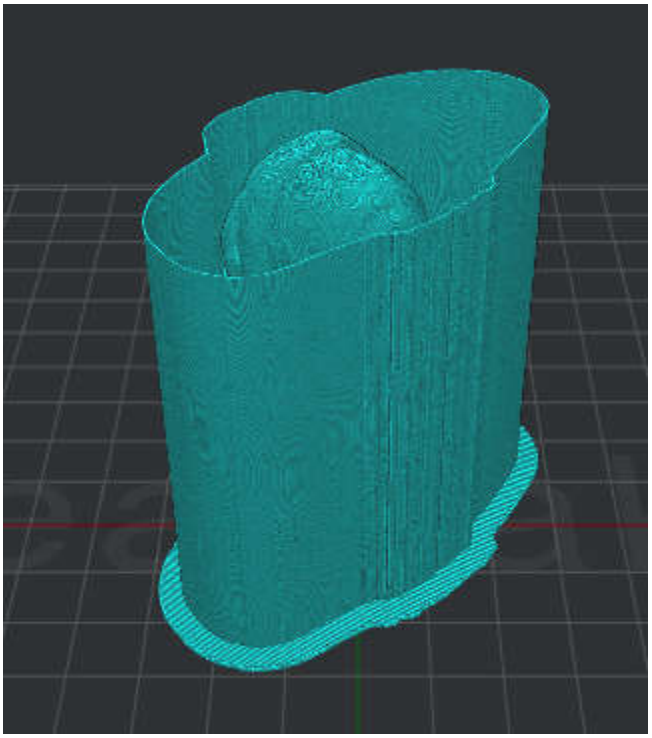


Figure 4.80: Set type of WipeWall to be "Vertical".

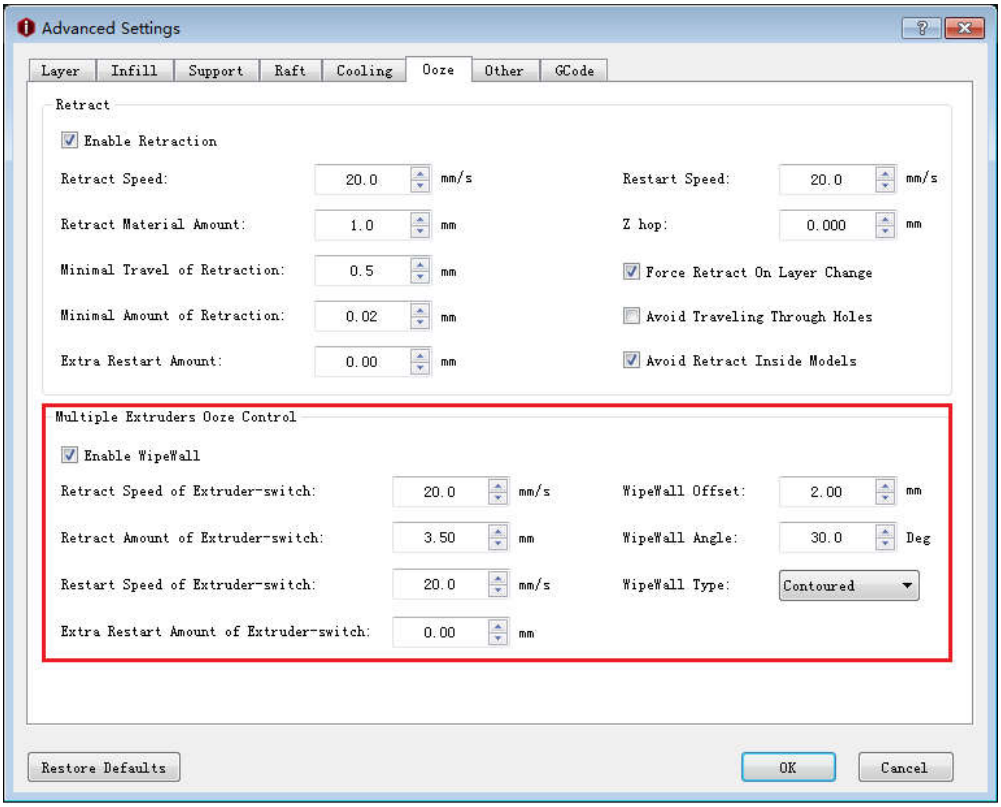


Figure 4.81: WipeWall will help decrease the effect of ooze from the other nozzle. But it is hard to totally avoid the ooze.

4.4 Filament Settings

When you want edit the parameters for your filament, follow check the steps as below:

1. Edit the existing template.

Printer->Filament Settings

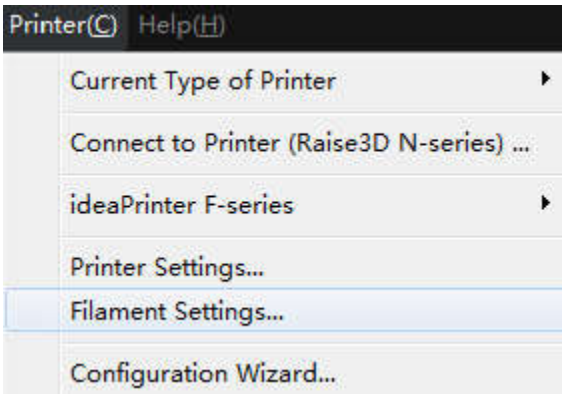


Figure 4.82: Choose "Printer" ->"Filament Settings" from menu bar.

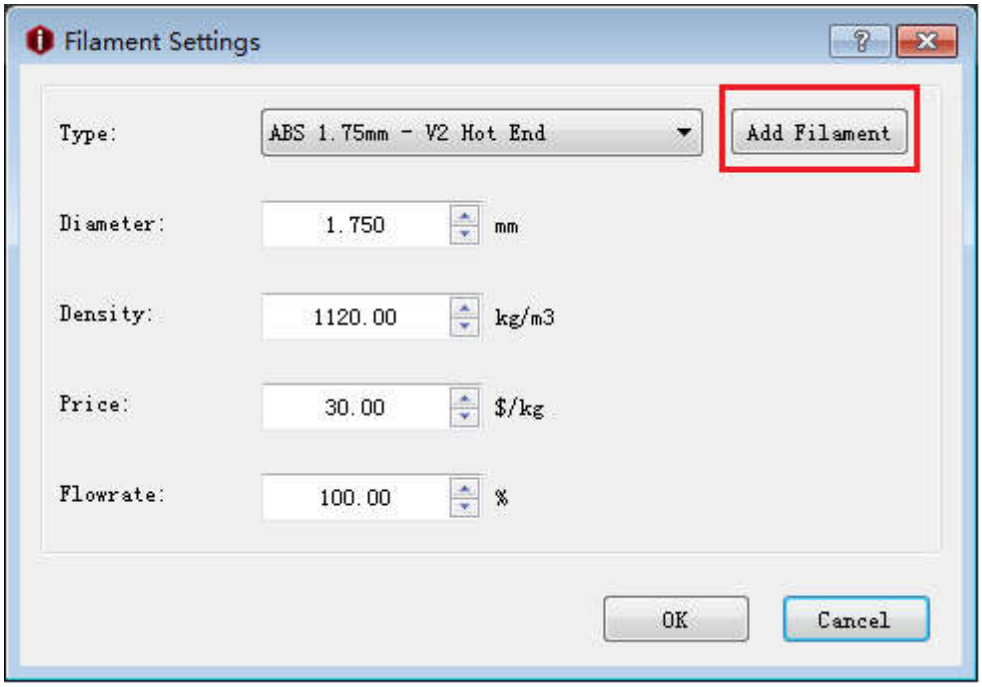


Figure 4.83: Choose a filament from the drop list and edit its parameters.

In this page, you can edit the parameters of your filament. The default settings are forRaise3D own filament and Raise3D certificated filament.

2. Add a new template.

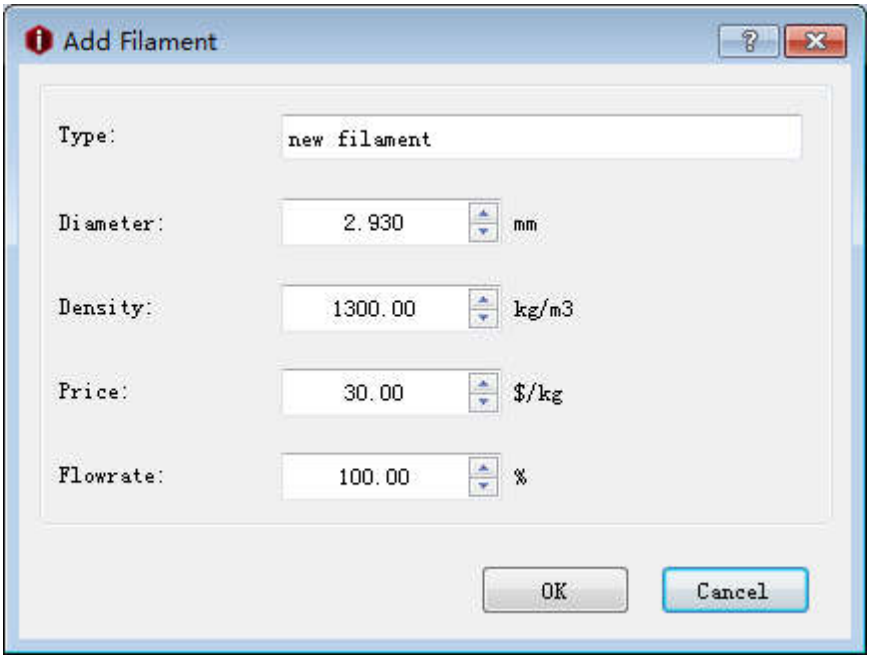


Figure 4.84: Set parameters for the new filament.

If you want to build a new template, choose ***Add Filament*** in last page.

Here we have a little tip, in most cases filament doesn't need flow rate compensation. But PLAand flexible materials needs to have flow rate compensation. PLA material has a default setting of flow rate at 94%. Print other materials with a PLA setting.gcodefile may cause wrong extrusion.

4.5 Printer settings

When you want edit the parameters for your printer, follow check the steps as below:

1. Edit the existing template in ***Printer->Printer Settings***

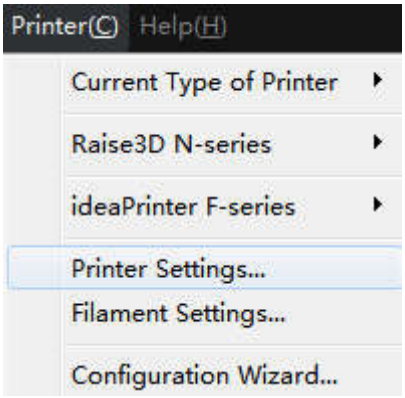


Figure 4.85: Choose "Printer" ->"Printer Settings" from menu bar.

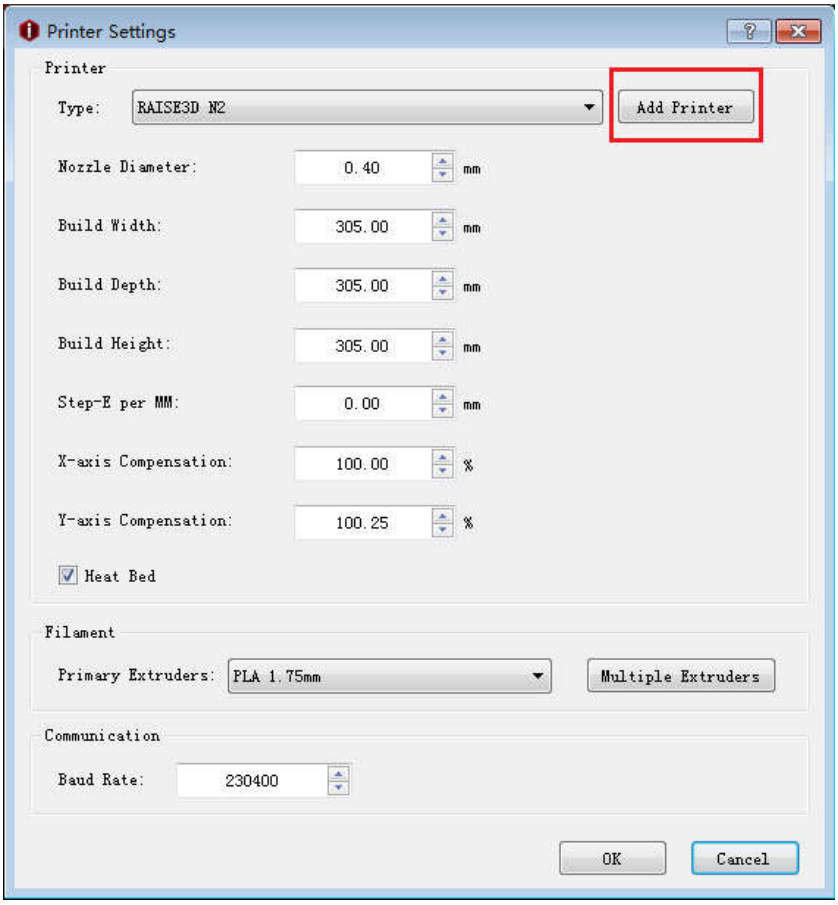


Figure 4.86: Choose a printer from the drop list and edit its parameters.

In this page, you can edit the parameter of your printer. The default settings are for Raise3D own printer.

2. Add a new printer.

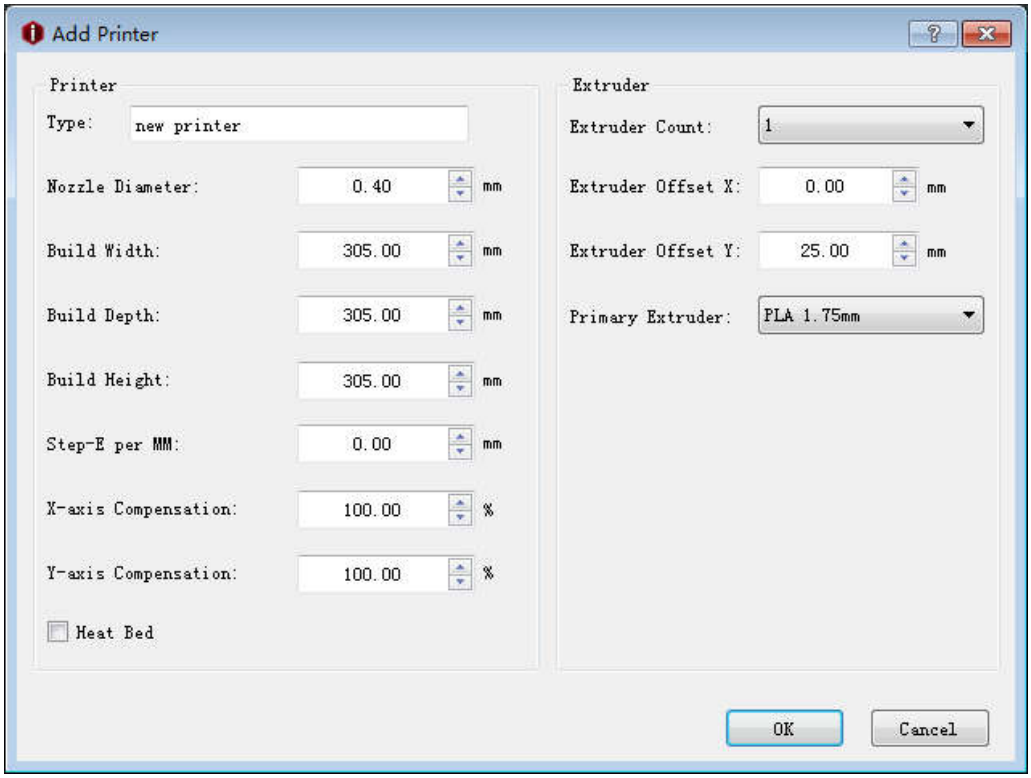


Figure 4.87: Set parameters for the new printer.

RAISE 3D

www.raise3d.com

US Office: 2398 Walsh Ave, Santa Clara, CA 95051
Tel: +1-888-963-9028
China Office: Floor 4 B5, 1688 North Guoquan Road, Yangpu District, Shanghai, 200000
Tel: +86 21 65337855