ideaMaker Manual

RAISE3D

www.raise3d.com

1 Using ideaMaker ... 3
1.1 What is ideaMaker? ... 3
1.2 Where to download ideaMaker? ... 3
2 Install ideaMaker ... 4
3 Let's Print! ... 8
3.1 Import .STL files ... 8
3.2 Slice the model ... 9
3.2.1 Select your printer type and filament type ... 9
3.2.2 Choose a slicing template ... 10
3.2.3 Edit the selected slicing template ... 11
3.2.4 Click "Slice" button to start slicing ... 12
3.3 Estimated Print Result ... 13
3.3.1 Check the sliced model by layers ... 13
3.4 Save the sliced files ... 16
4 Interface ... 35
4.1 Menu Bar ... 36
4.2 Tool Bar ... 55
4.3 Operation property ... 65
4.4 Model list ... 66
4.5 Uploading queue ... 68
4.6 Instant operation bar ... 69
4.7 Model preview ... 70
4.8 Perspective transformation ... 71
4.9 Detecting Information ... 72
5 Slicing settings ... 73
5.1. Main Template ... 73
5.1.1 Create Template ... 73
5.1.2 Duplicate Template ... 75
5.1.3 Edit Template ... 76
5.1.3.1 Layer ... 81
5.1.3.2 Extruder ... 84
5.1.3.3 Infill ... 89
5.1.3.4 Support ... 94
5.1.3.5 Platform Additions ... 102
5.1.3.6 Cooling ... 108
5.1.3.7 Temperature ... 111
5.1.3.8 Advanced ... 113
5.1.3.9 Ooze ... 115
5.1.3.9 Other ... 122
5.1.3.10 GCode ... 125
5.1.4 Compare Template ... 130
5.1.5 Delete Template ... 131
5.1.6 Import Template ... 132

5.1.7 Export Template 134
5.2. Group and Layer Settings 135
5.2.1 Setting Group 135
5.2.1.1 Add New Setting Group 136
5.2.1.2 Delete setting groups 137
5.2.1.3 Edit the selected setting group 137
5.2.1.4 Add Settings 138
5.2.1.5 Delete Settings 139
5.2.2 Per-Layer Settings 140
5.2.2.1 Add Per-Layer Settings 140
5.2.2.2 Delete Per-Layer Settings 140
5.2.2.3 Edit Per-Layer Settings 140
5.2.3 Import and Export Group and Layer Settings 142
5.3 Per-Model Extruder 143
6 Multiple Extruders 144
6.1 Calibrate XY offset 144
6.2 Advanced Settings 150
7 Filament Settings 160
7.1 Edit Filament Settings 160
7.2 Add/Duplicate Filament 163
7.3 Remove Filament 165
8 Printer Settings 166
8.1 Edit Printer Settings 169
8.2 Add/Duplicate Printer 171
8.3 Import Printer 173
8.4 Remove Printer 174

1 Using ideaMaker

1.1 What is ideaMaker?

ideaMaker is a slicing software which prepares 3D models for printing and turns them into G-Code 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 select your language preference. Then click ***Next*** to move on to the next menu. If you are using an older version of ideaMaker or reinstalling ideaMaker, you will need to close all open processes of ideaMaker. So the installation can be completed without error. Please check your task manager to make sure that there is no ideaMaker process before installation.

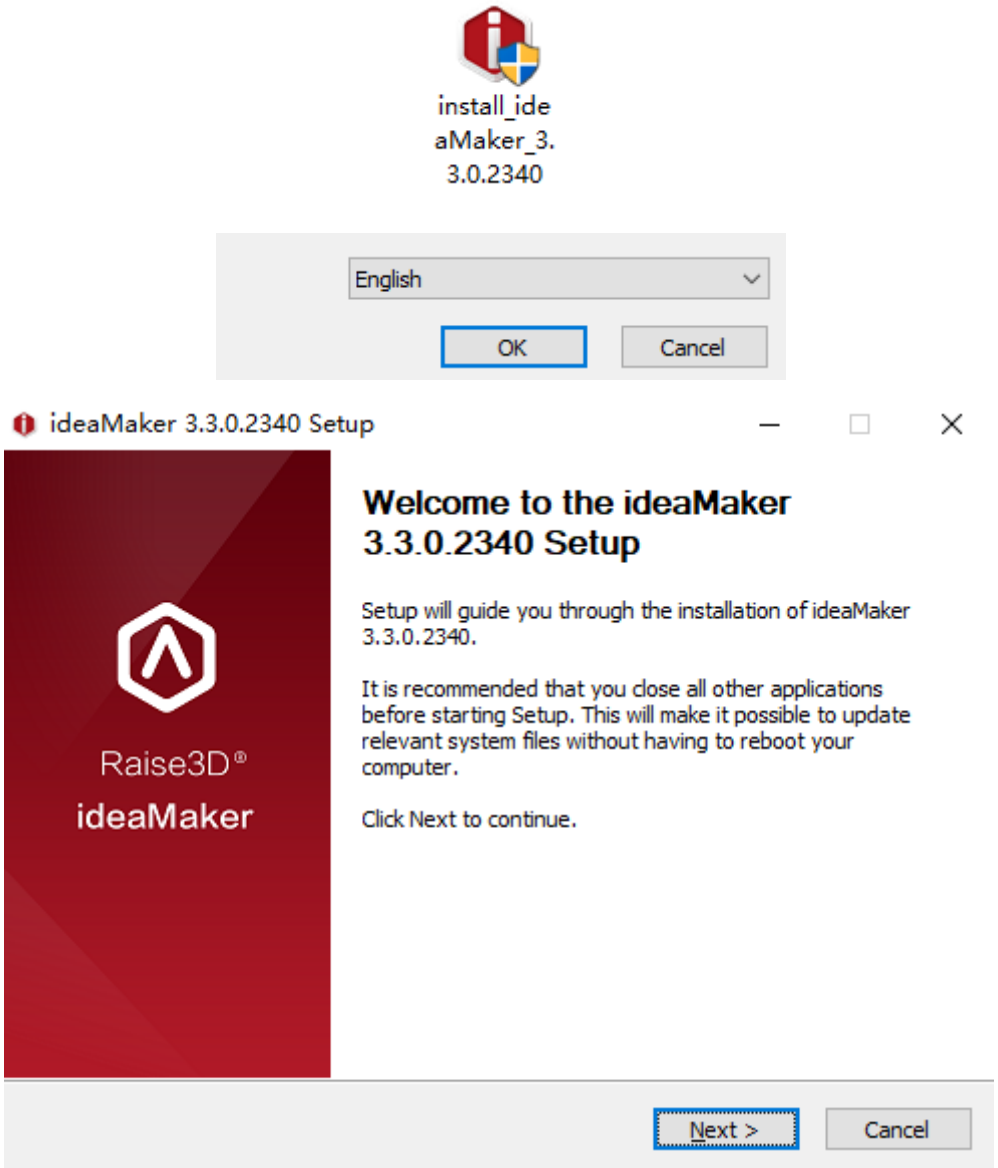


Figure 2.1: Select a language for installing ideaMaker.

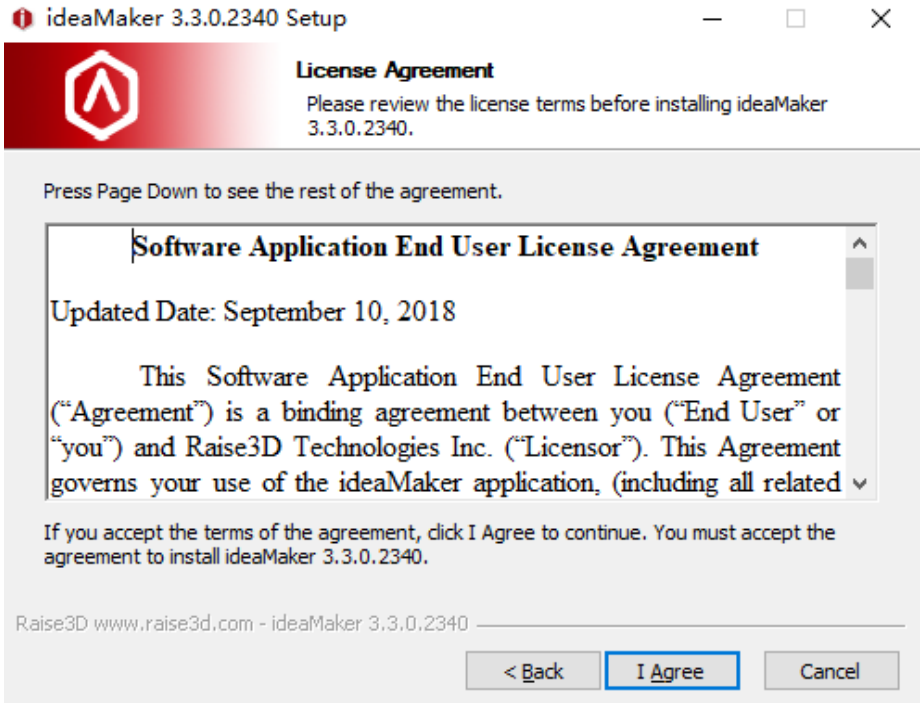


Figure 2.2: License Agreement.

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

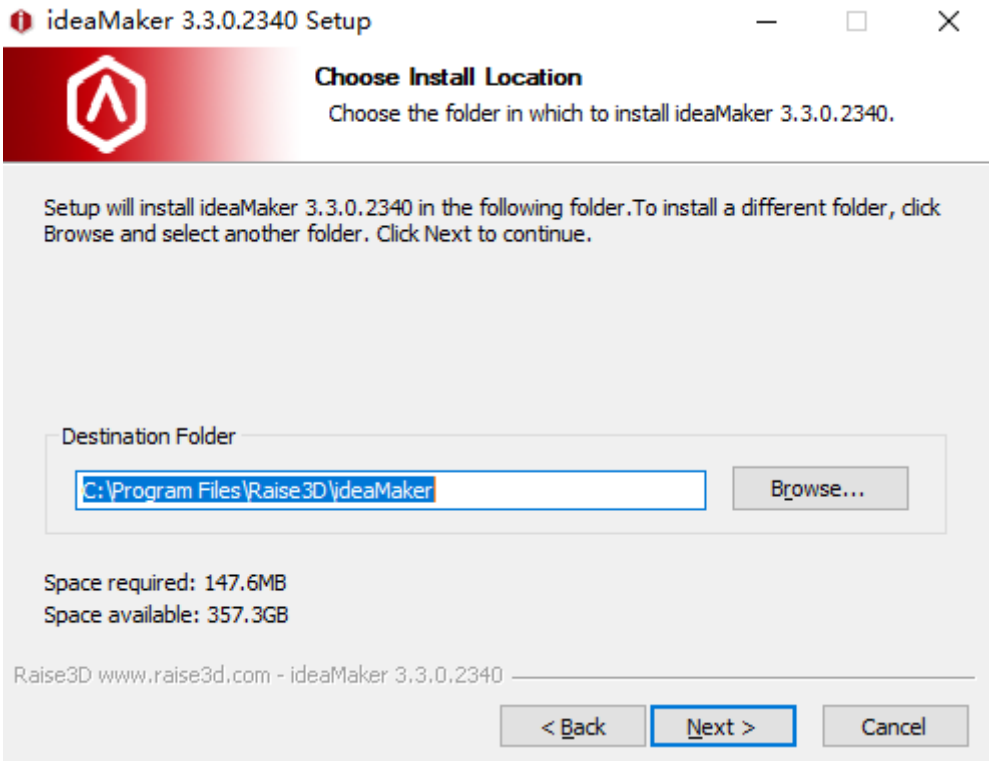


Figure 2.3: Select a path to install your ideaMaker.

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.

Microsoft Visual C++ 2008 SP1 Redistributable is auto-checked by the installation software. If you have installed that in your computer before, you don't need to check it again.

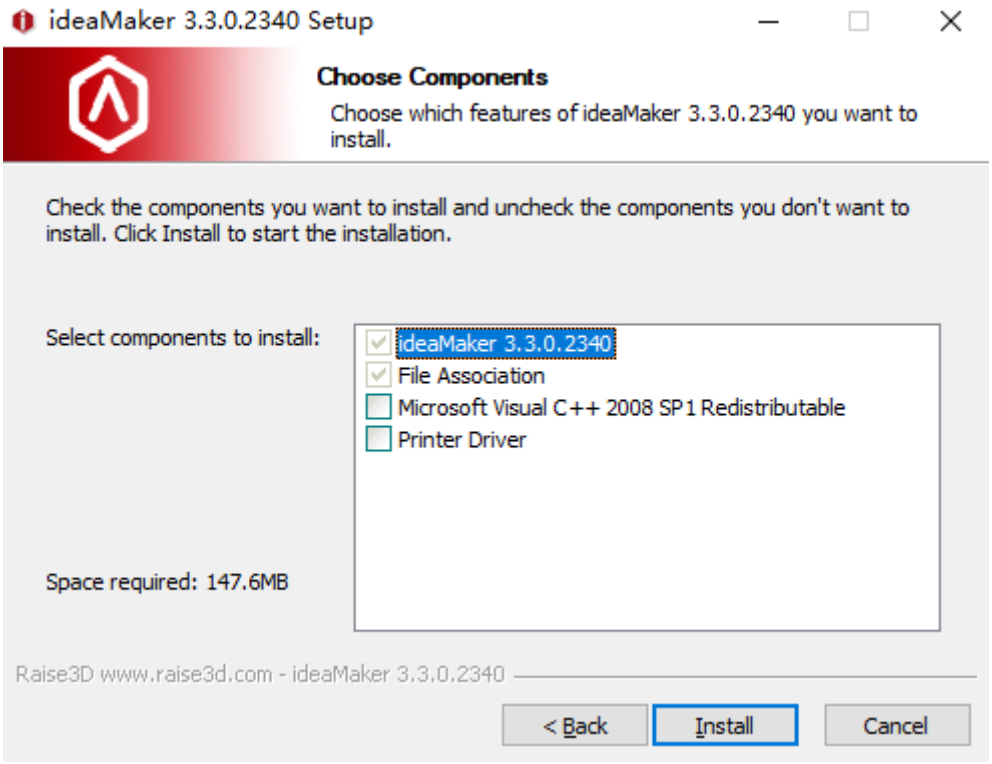


Figure 2.4: You don't need to select Printer Driver if your printer is N-series Printer.

4. After the installation is completed, click ***Next*** to move onto the next step.

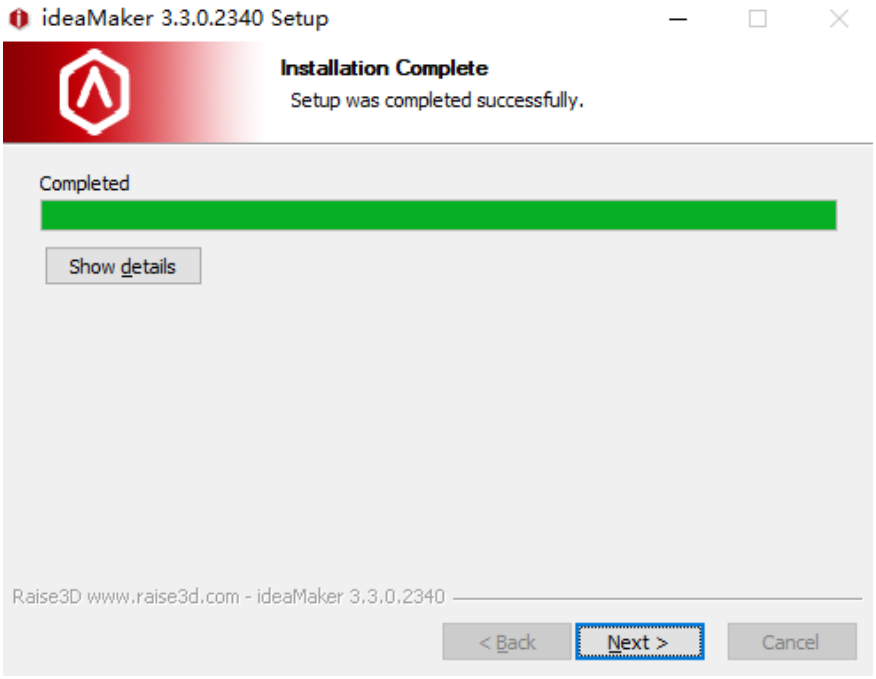


Figure 2.5: Press "Next" to complete the installation.

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

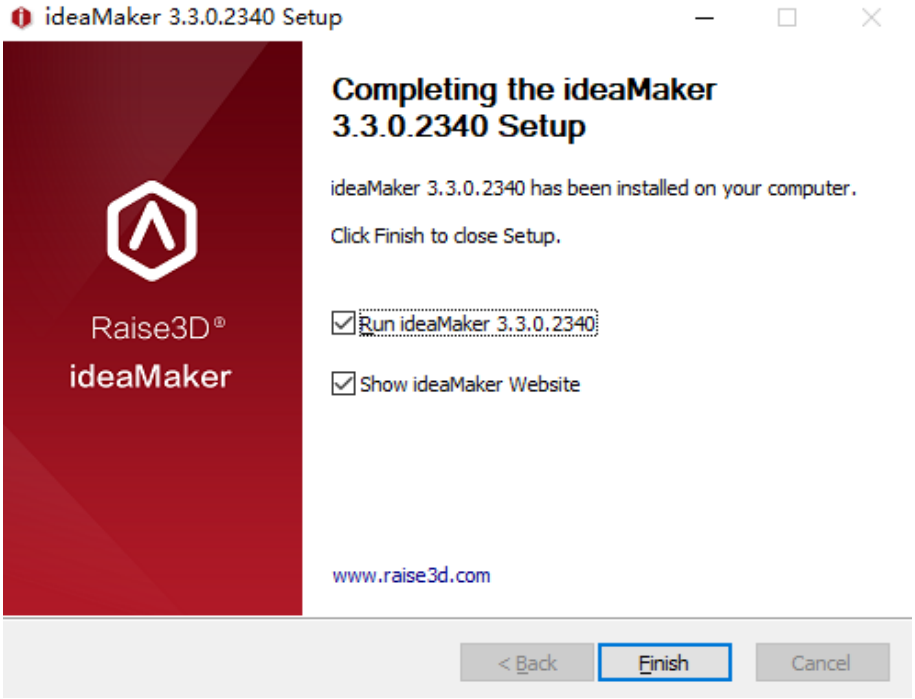


Figure 2.6: Complete the installation.

3 Let's Print!

As you have finished the installation, you can now begin to use ideaMaker and prepare files to print. Here are the first steps.

3.1 Import .STL files

Click the button **"+"** to import .STL, .OBJ or .3MF files. Click the ***Repair*** button to execute an auto-repair of your model if the box at bottom-right corner shows any error warnings.

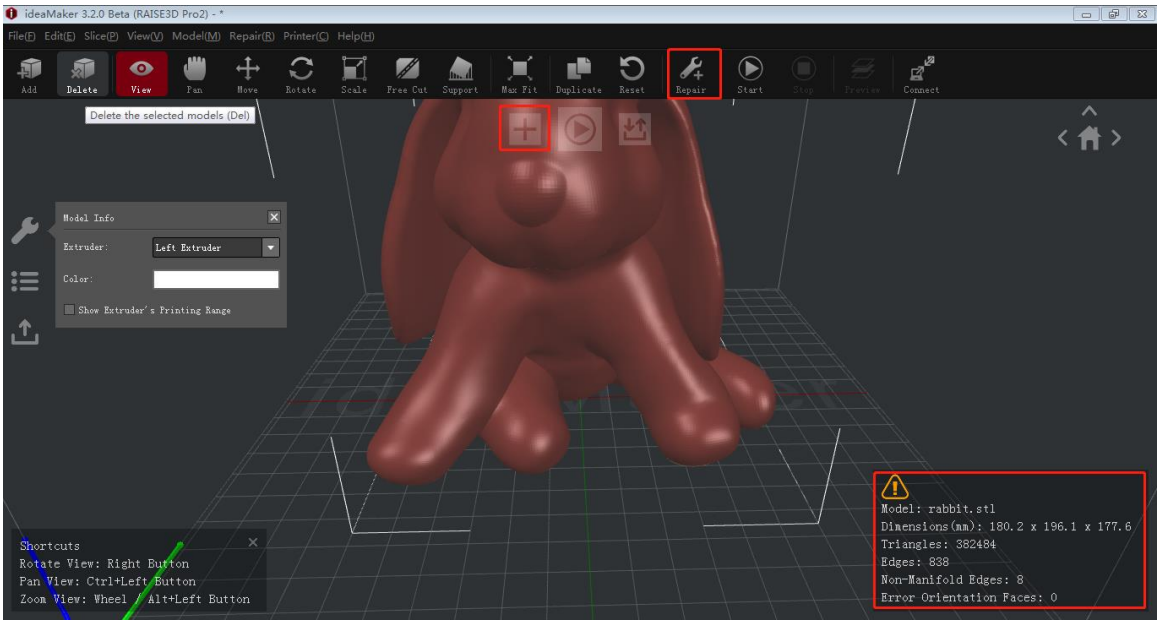


Figure 3.1: Press the "+" icon to import STL files. If it doesn't display a green tick below, press the "Repair" button to execute an auto-repair.

3.2 Slice the model

Click the start button to slice the model.

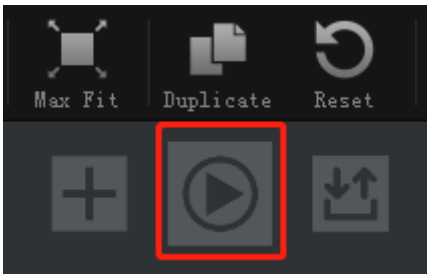


Figure 3.2: start a slice.

3.2.1 Select your printer type and filament type

If you are using V2 hot end, please select both printer and filament as V2. If you are not using V2 hot end, make sure both are selected as standard profiles.

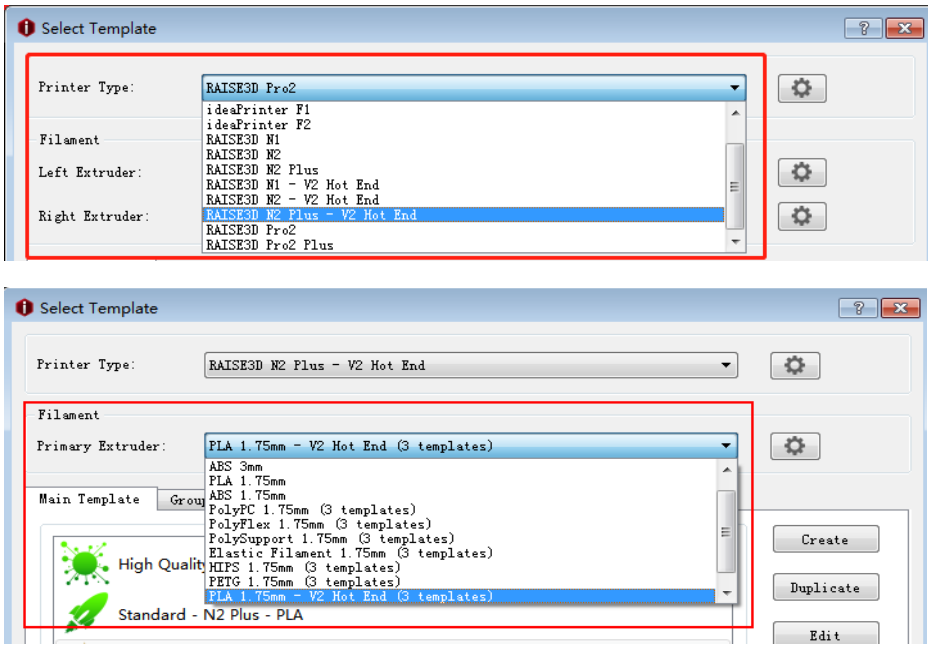


Figure 3.3: Select the printer and filament you want to use.

3.2.2 Choose a slicing template

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

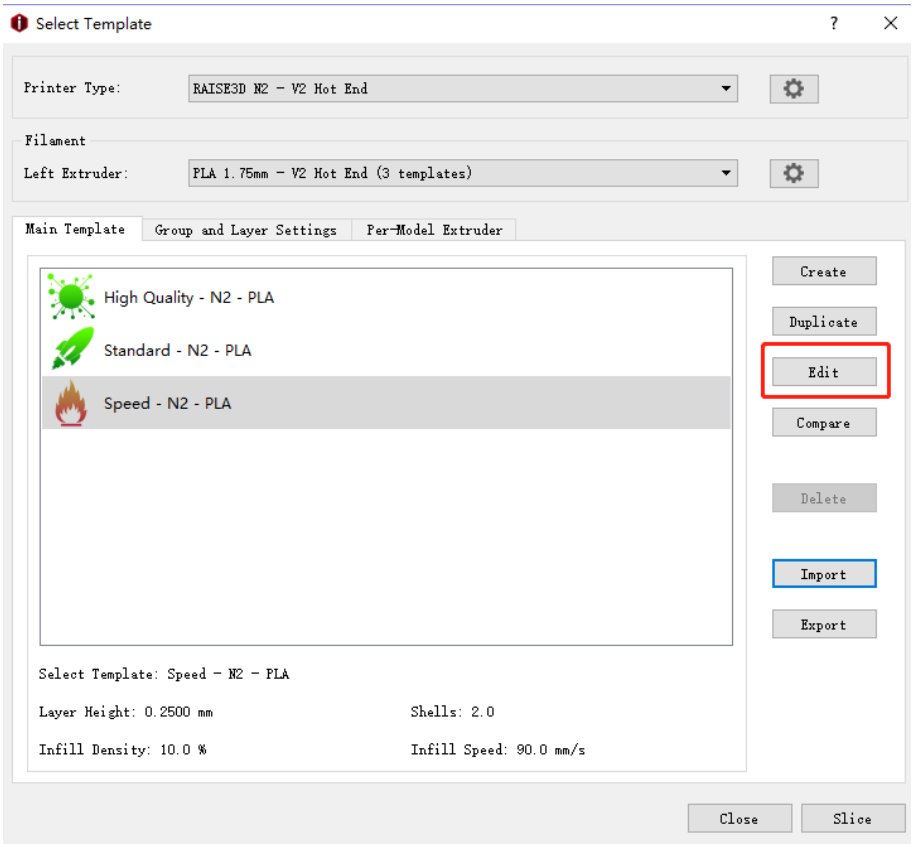


Figure 3.4: Select a template to start with. Click "Edit" to adjust the settings.

3.2.3 Edit the selected slicing template

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 ***Advanced*** as well if you want. We will talk about that in the following chapters.)

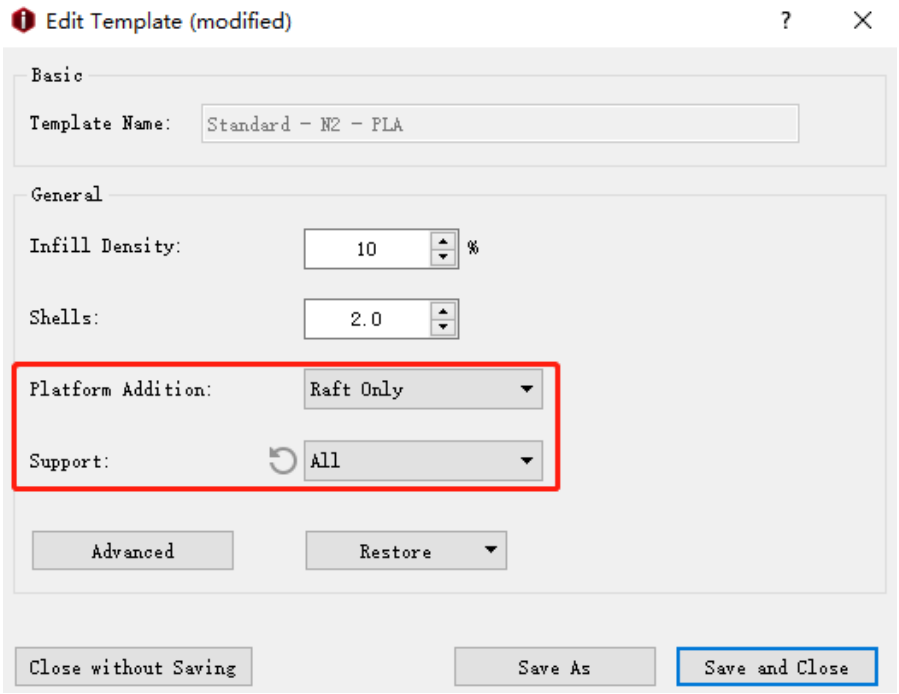


Figure 3.5: Select a suitable "Platform Addition" and "Support" type for your model.

3.2.4 Click "Slice" button to start slicing

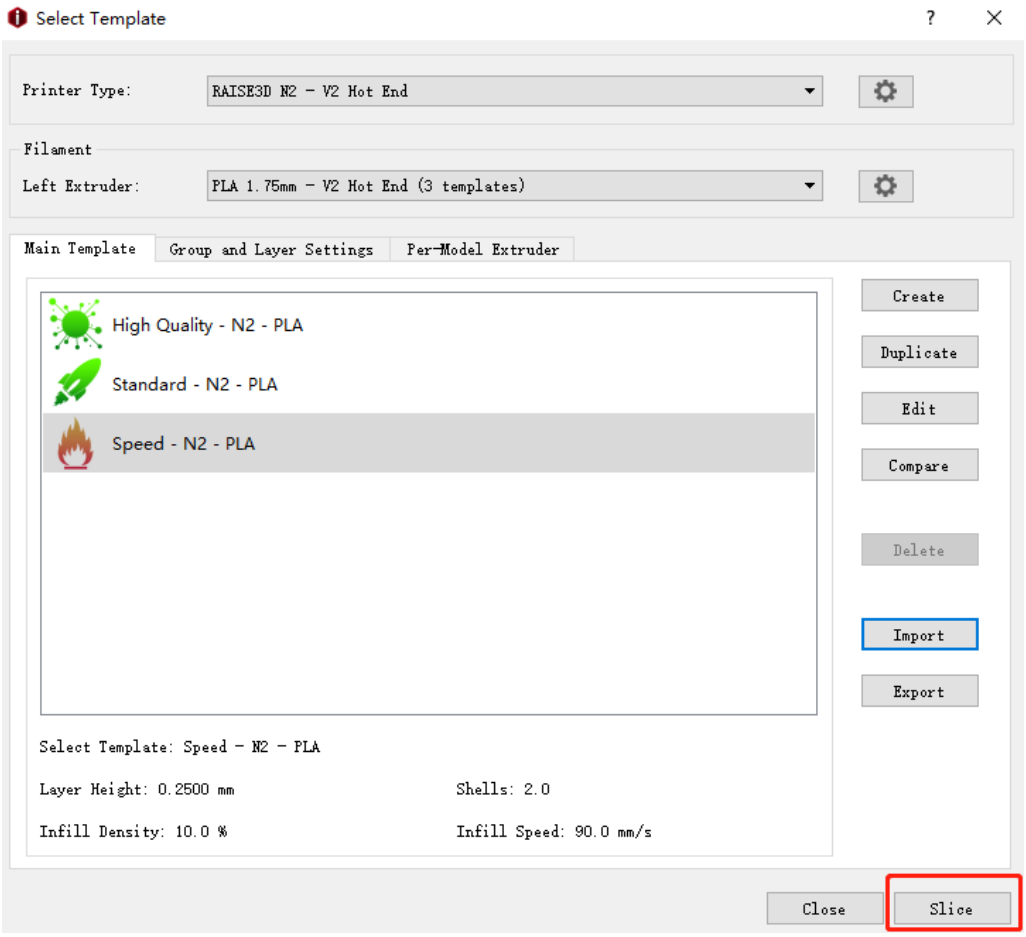


Figure 3.6: Click "Slice" to start slicing.

3.3 Estimated Print Result

After completing the slicing, ideaMaker will advise the estimated data for your reference as ***Figure 3.7***.

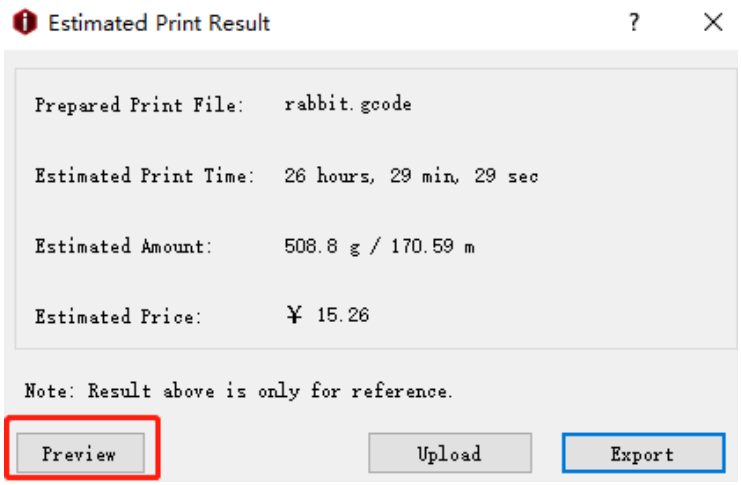


Figure 3.7: Here you can check estimated print time and amount of filament needed.

3.3.1 Check the sliced model by layers

Click ***Preview*** button so that you can check the sliced model by height as ***Figure 3.8***.

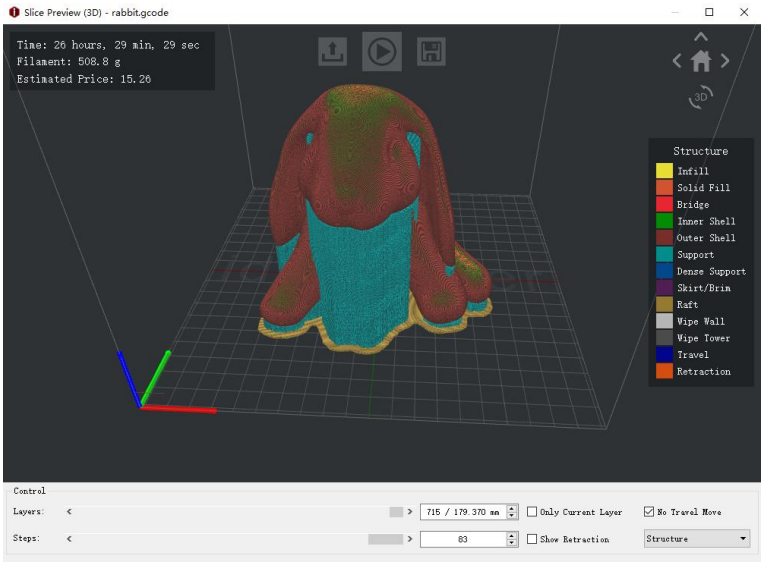


Figure 3.8: Preview how the model will be printed out.

In this page you can check the ***Time***, ***Filament***, ***Estimated Price***, ***Retraction*** and ***Travel*** move.

Time refers that the time required to the printing task.

Filament refers that the filament quantity required to the printing task.

Estimated Price refers that the cost required to the printing task.

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

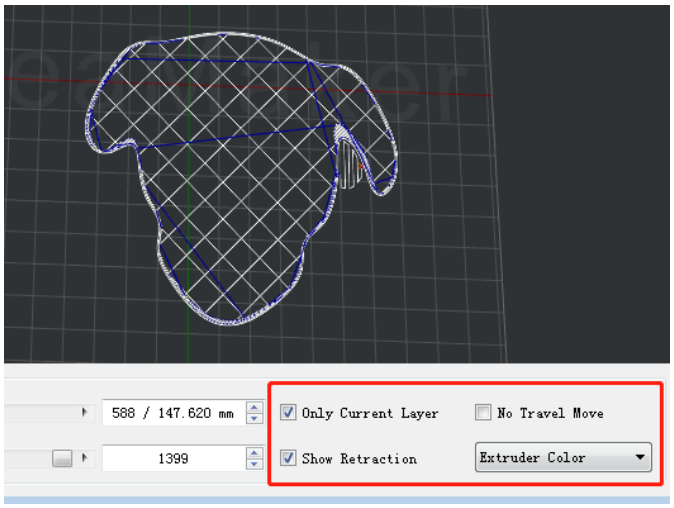


Figure 3.9: Check the retraction points with "Show Retraction". Check one layer with "Only Current Layer".

It can also show different structures in different colors by selecting ***Structure*** in ***drop list***. 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.

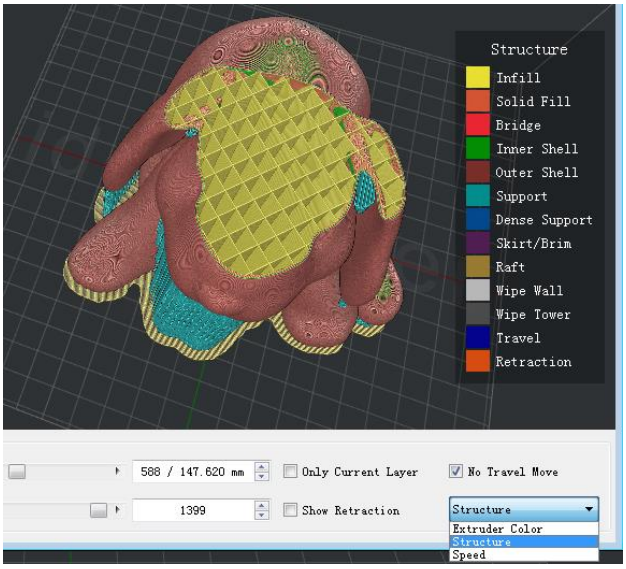


Figure 3.10: Show different structures with different colors.

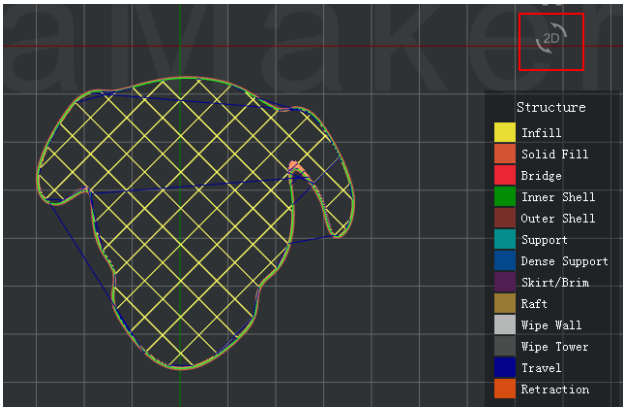


Figure 3.11: Enable "2D" to check the current layer with orthographic view.

: Upload the printing file to your printer.

: Start slicing your model.

: Save the printing file to local disk.

: Check model with home view.

3.4 Save the sliced files

Close the preview dialogue box after the confirmation.

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

Option1: Export the sliced files to USB storage or SD card.

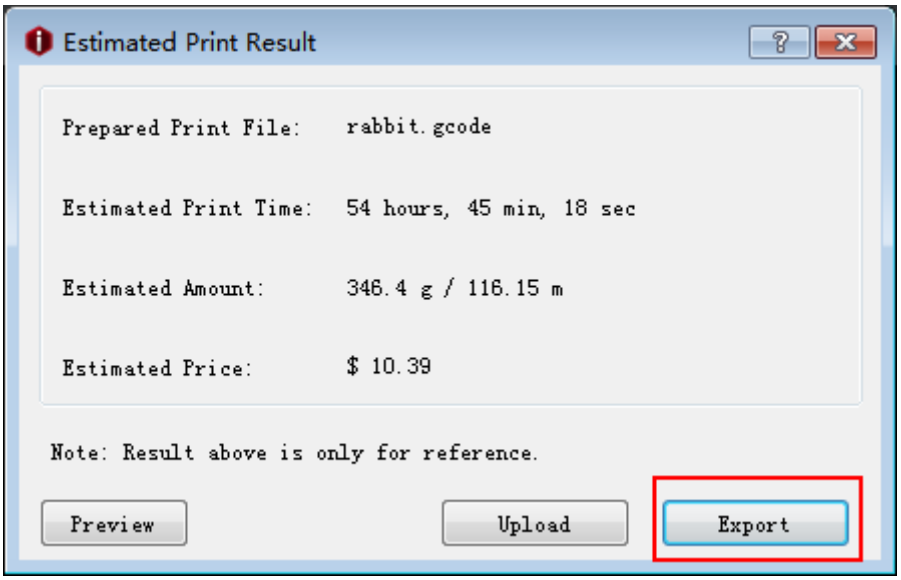


Figure 3.12: Click "Export" in "Estimated Print Result" window to save the sliced files.

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 (G-Code file and .data file) to USB storage. It is important to copy both files.

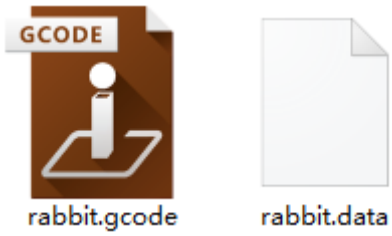


Figure 3.13: G-Code file and .data file are both necessary for N-series printers.

2. Insert the USB or SD storage unit into your printer and select the file to start your first print.

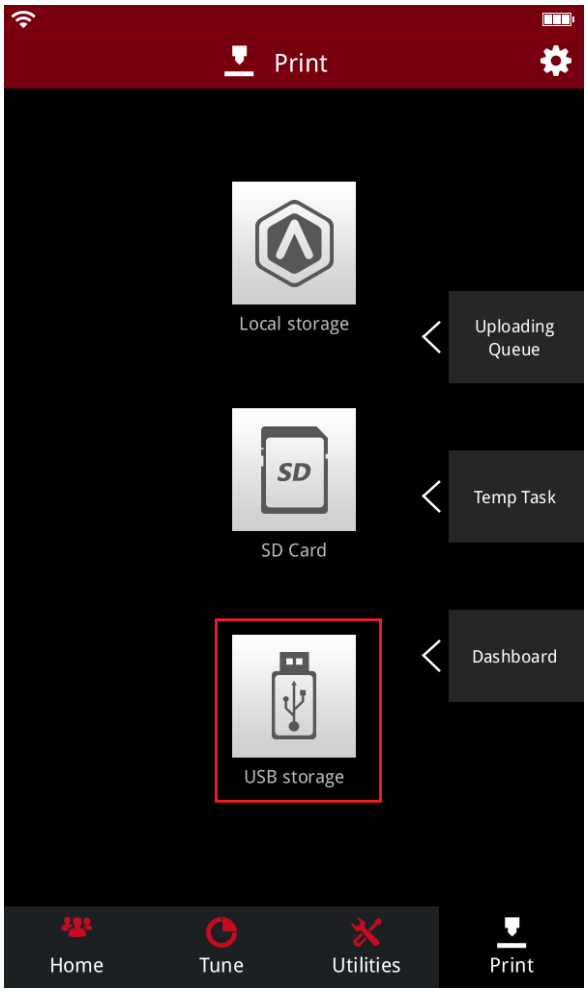


Figure 3.14: If you are using USB driver, press the "USB storage" to check the files inside. If you are using SD card, press "SD Card" to check the files inside.

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.

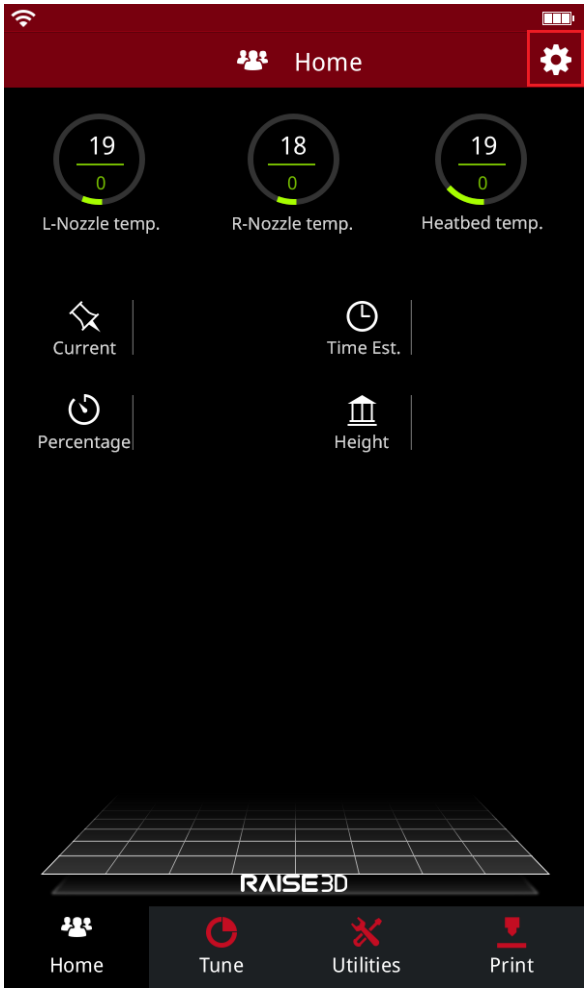


Figure 3.15: Press the gear button to enter the setting page.

2. Select ***WLAN*** tab. Enable ***WLAN*** and select your network from the list then input the password.

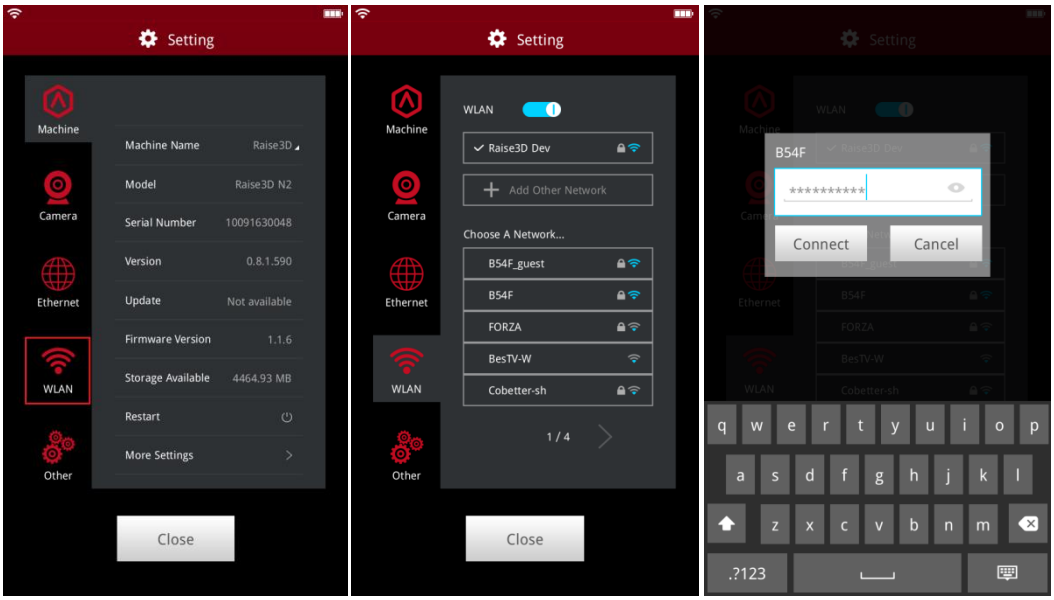
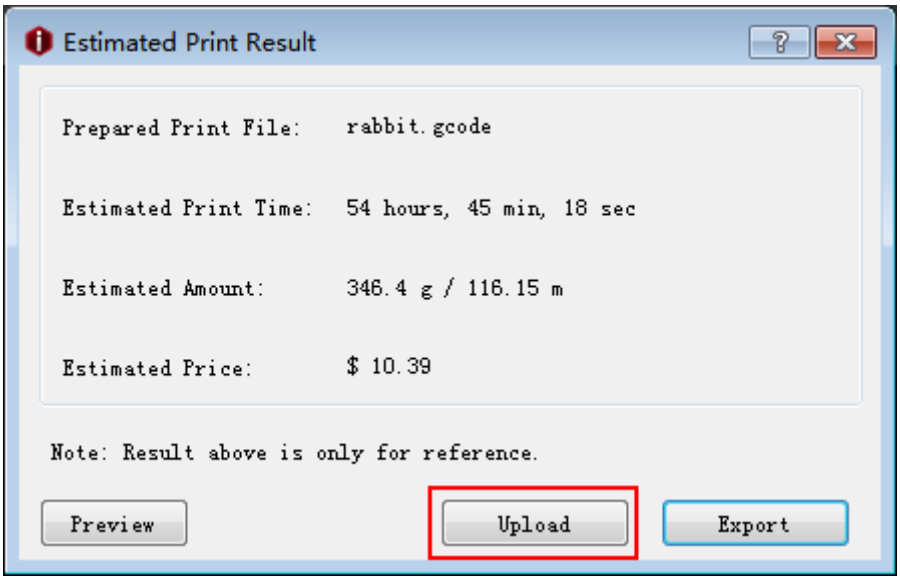


Figure 3.16: Connect the printer with your "WLAN".

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



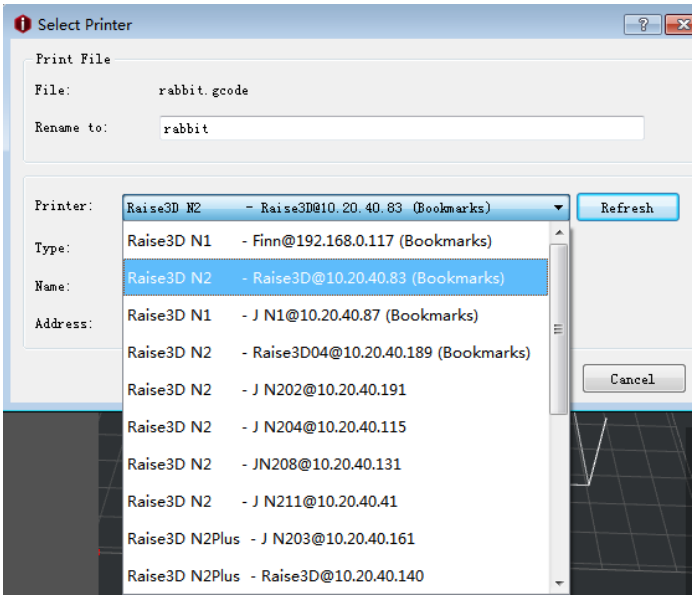


Figure 3.17: Click "Upload" in "Estimated Print Result" page and select a printer you want to use from the drop list.

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

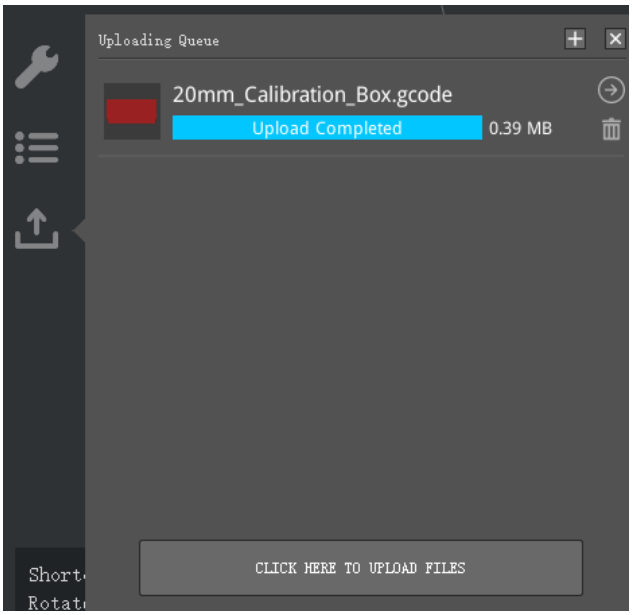


Figure 3.18: Check the uploading progress at the left side of ideaMaker.

You also can press ***CLICK HERE TO UPDATE FILES*** button to update files which saved in computers.

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

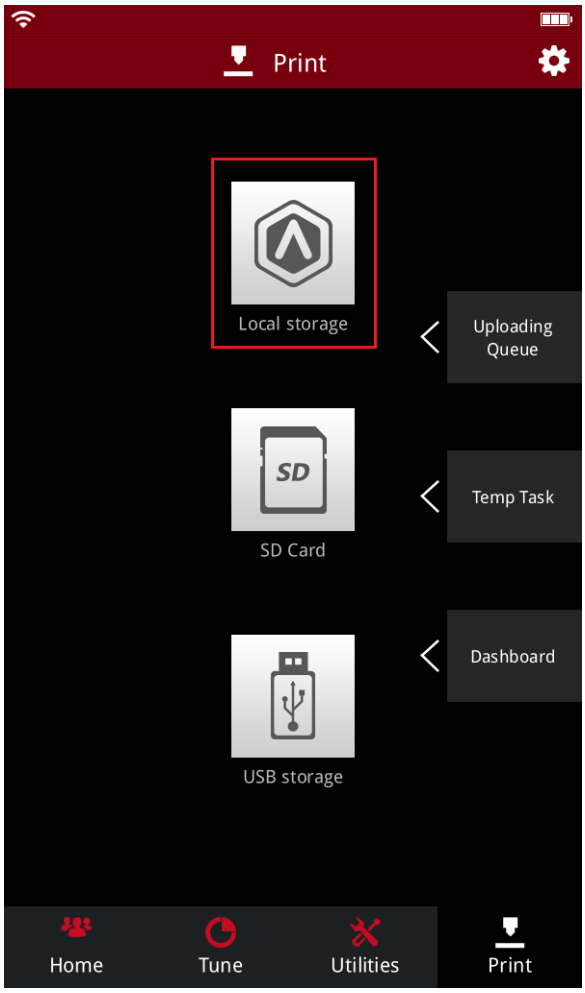


Figure 3.19: Uploaded files will be inside "Local Storage".

6. Remote connect.

You can also connect to your printer through ideaMaker remotely. Select ***Printer*** ->***Connect to Printer (Raise3D N-series)***.

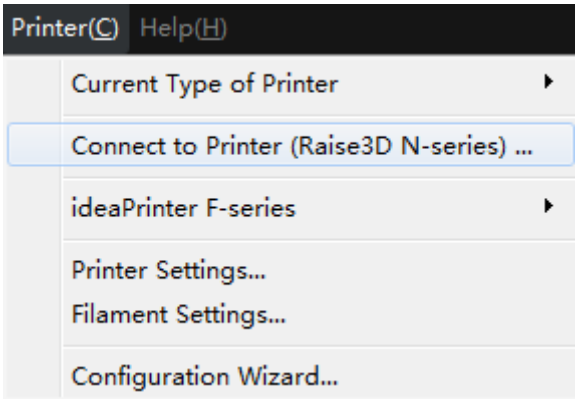


Figure 3.20: Select Printer ->Connect to Printer (Raise3D N-series).

Select the printer you want to connect with in this page.

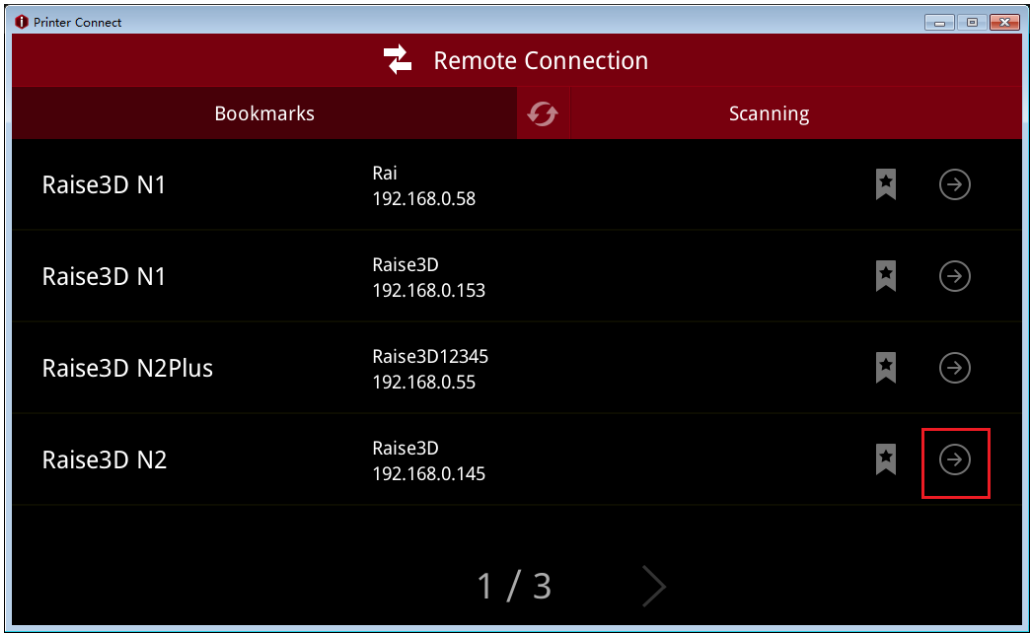


Figure 3.21: Select the printer you want to connect in "Scanning" list. Click the arrow icon to enter.

Waiting for connecting.

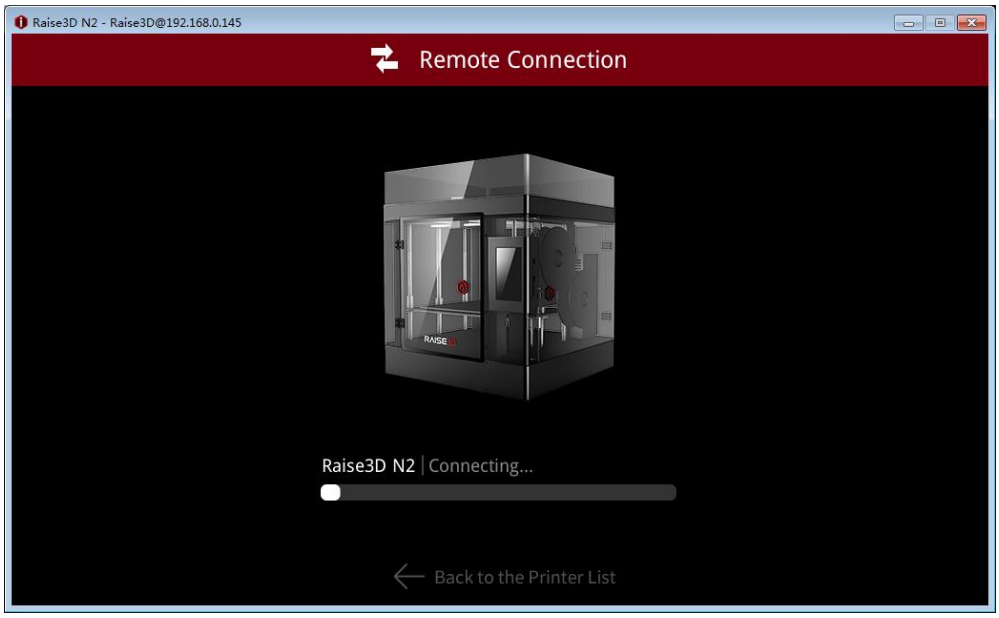


Figure 3.22: Connecting page.

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

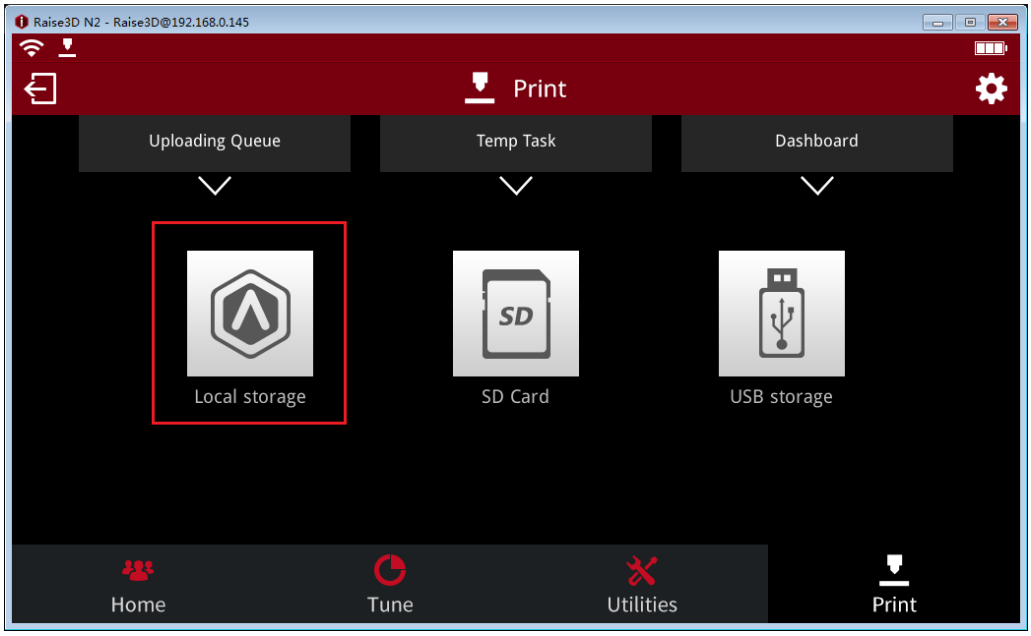


Figure 3.23: Click "Local storage" to check the uploaded files.

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

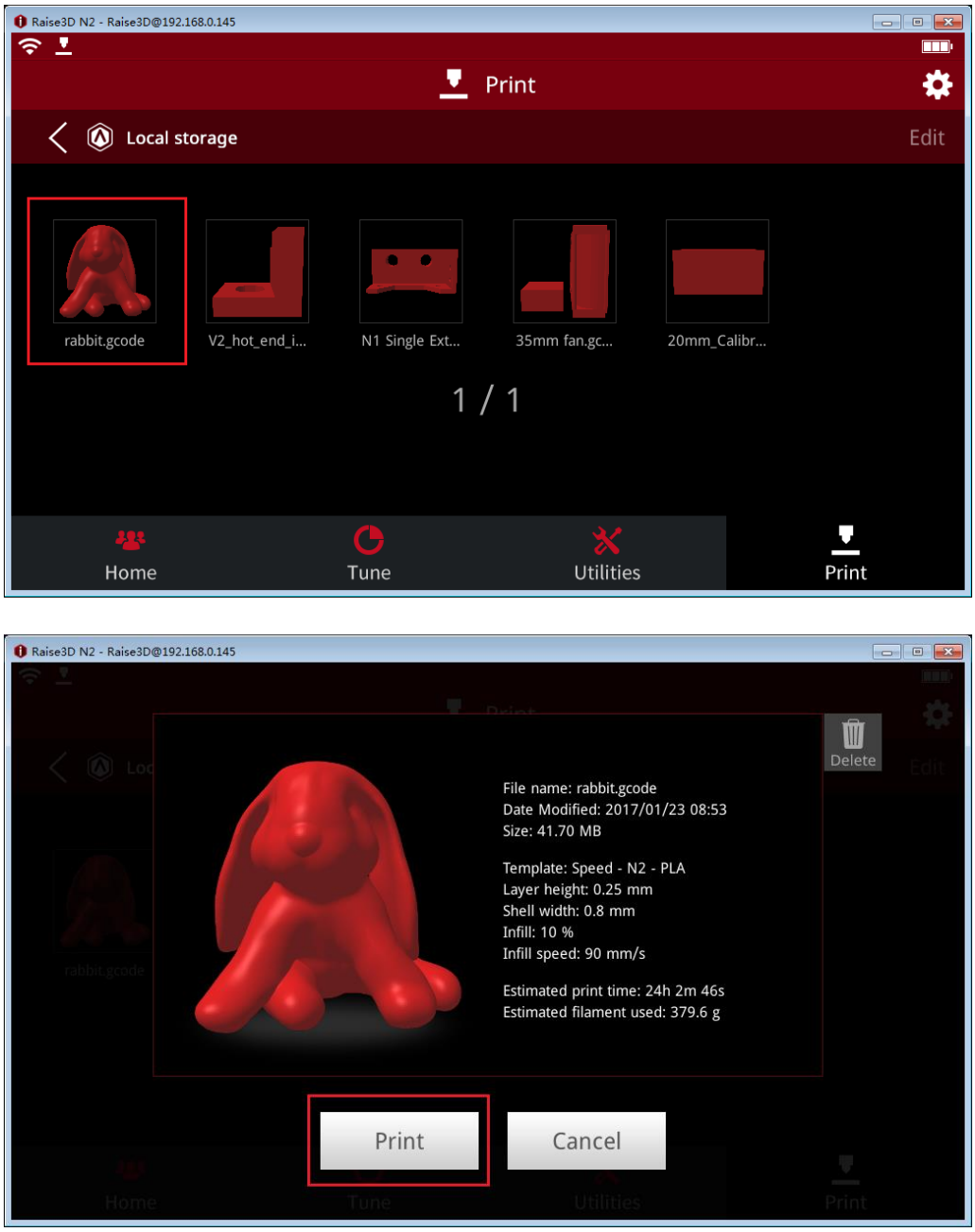


Figure 3.24: Select one file to print and check the file information.

Home:

L-Nozzle temp: the temperature of the left nozzle. Upper one is current temperature. Below one is target temperature.

R-Nozzle temp: the temperature of the right nozzle. Upper one is current temperature. Below one is target temperature.

Heated bed temperature: the temperature of the heating bed. Upper one is current temperature. Below one is target temperature.

Current: the name of the printing model as ***Figure 3.25***.

Figure 3.25: Current

Time Est.: the total time and the remaining time of the printing model. The bigger one refers to the remaining time. The small below one refers to the total time.

Figure 3.26: Time Est.

Percentage: the percentage of completed model.

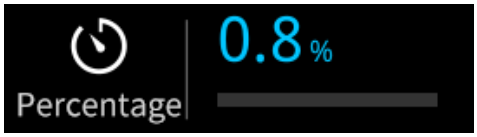
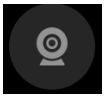


Figure 3.27: Percentage.

Height: the height of the completed model. Click the values then you can check the layer information, the current layer and how many layers in total.

Figure 3.28: Height.

: pause or stop printing task.

 : Press camera button to take photos and save it to local disk.

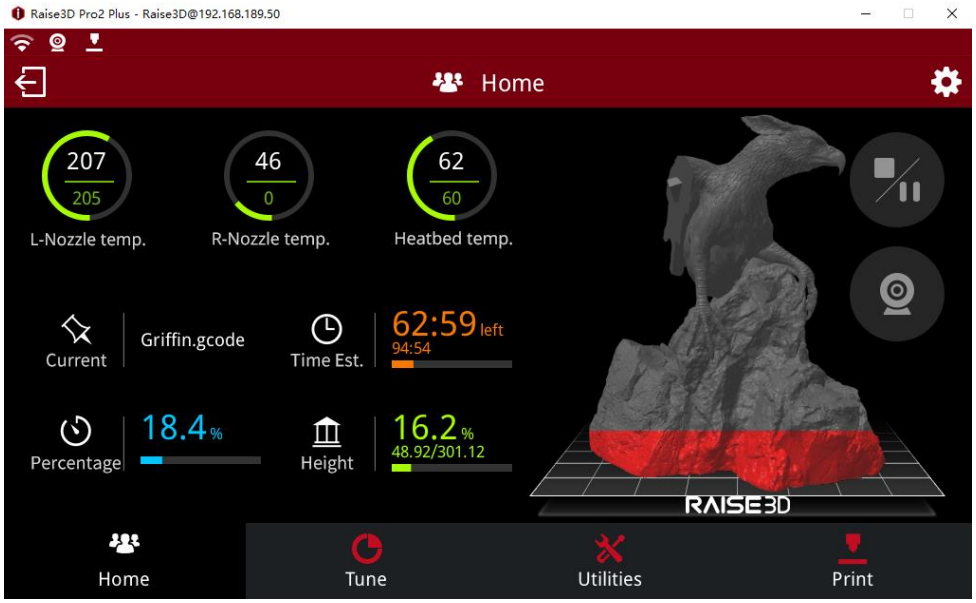


Figure 3.29: Home.

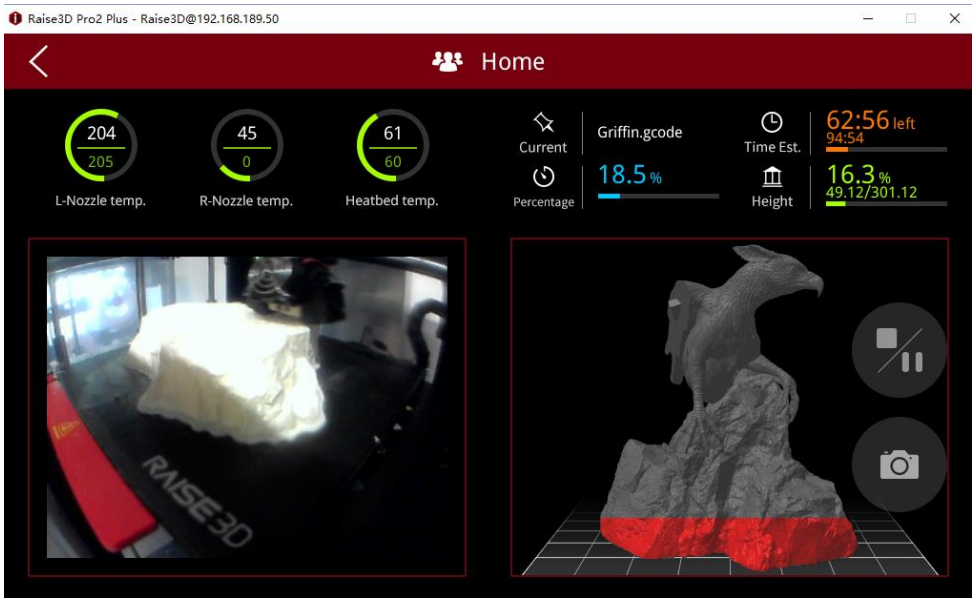


Figure 3.30: Take photos.

Tune:

L-Nozzle temperature: the temperature of the left nozzle.
R-Nozzle temperature: the temperature of the right nozzle.
If your machine is a dual-extruder machine, it will show ***L-Nozzle temperature*** and ***R-Nozzle temperature.***
If your machine is a single-extruder machine, it will only show Nozzle temperature here.
Heated bed temperature: the temperature of the heated bed.
Feed rate: the overall printing speed percentage.
Fan speed: the speed of the fan which in controlled slot.
Note: only the right-side downward fan on single-extruder machine can be controlled.
L-Nozzle flow rate: the flow rate of the left nozzle.
R-Nozzle flow rate: the flow rate of the right nozzle.

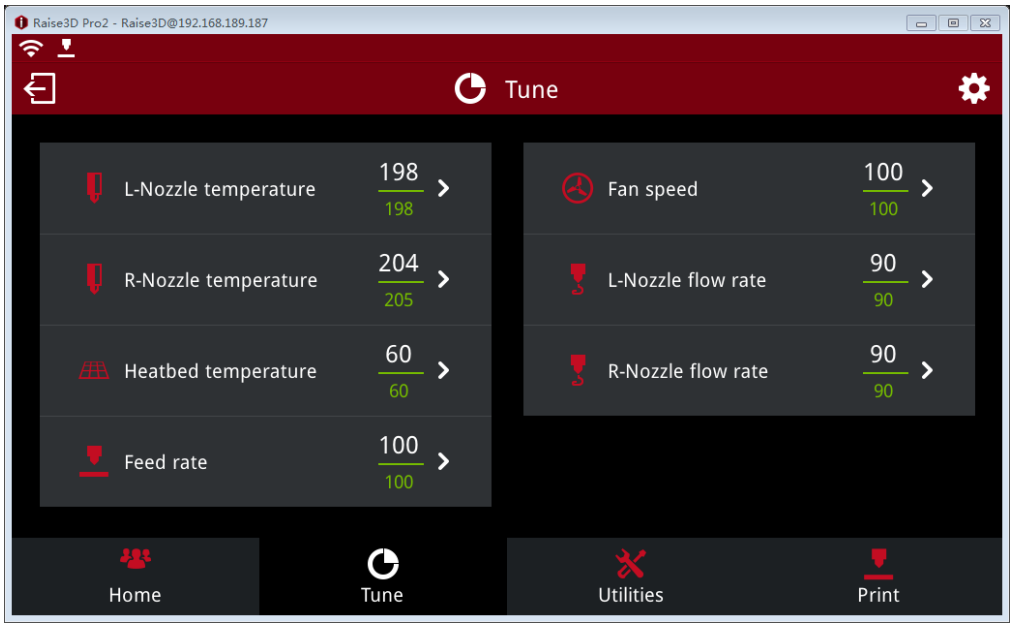


Figure 3.31: Tune.

Utilities:

Move Steps: the distance X/Y/Z will move per click, there are 3 types: 10.0mm, 1.0mm and 0.1mm.

: move the extruder in X direction.

: move the extruder in Y direction.

: home the extruder in X & Y directions.

: move the build plate in Z direction.

: home the build plat in Z direction.

: move the feeding gear to load or unload filament.

If your machine is a dual-extruder machine it will show L and R nozzle icon here.

: the value of X/Y/Z/E have moved. Will back to 0 after homing or reboot.

: disable all the five motors. (If the machine is a single-extruder one, then there are four motors in total.) You can push the head by hands after triggering this function.

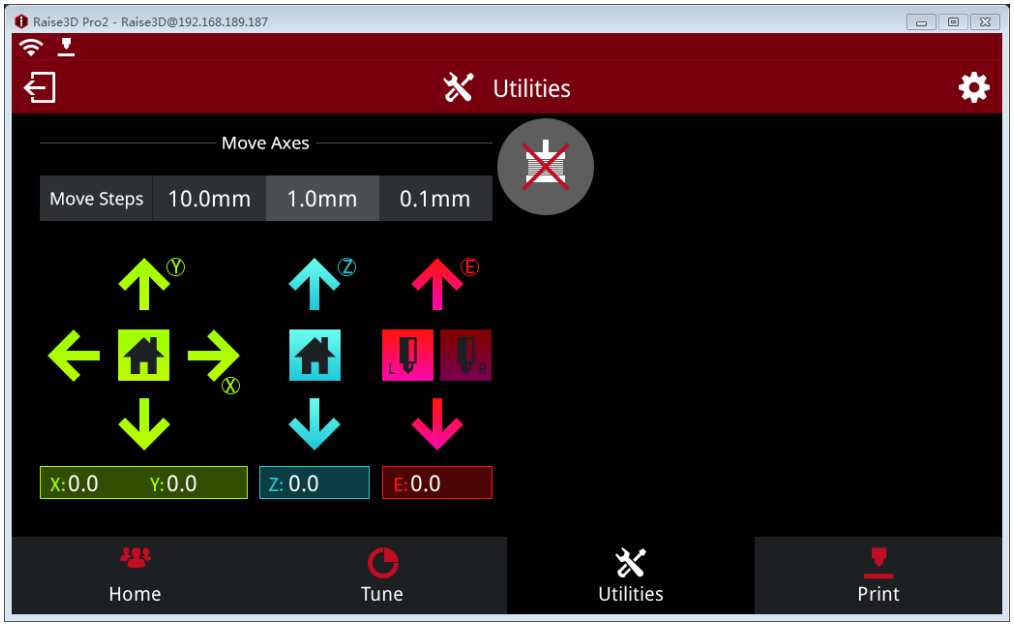


Figure 3.31: Utilities.

Print:

Local Storage: store files which be uploaded by Ethernet or WLAN.

USB Storage: the printing files which saved by USB disk.

Uploading Queue: the files which uploaded Ethernet or WLAN.

Temp task: the printing task which be disturbed by the power loss.

Dashboard: the completed printing task will be stored in here as ***Figure 3.33***.

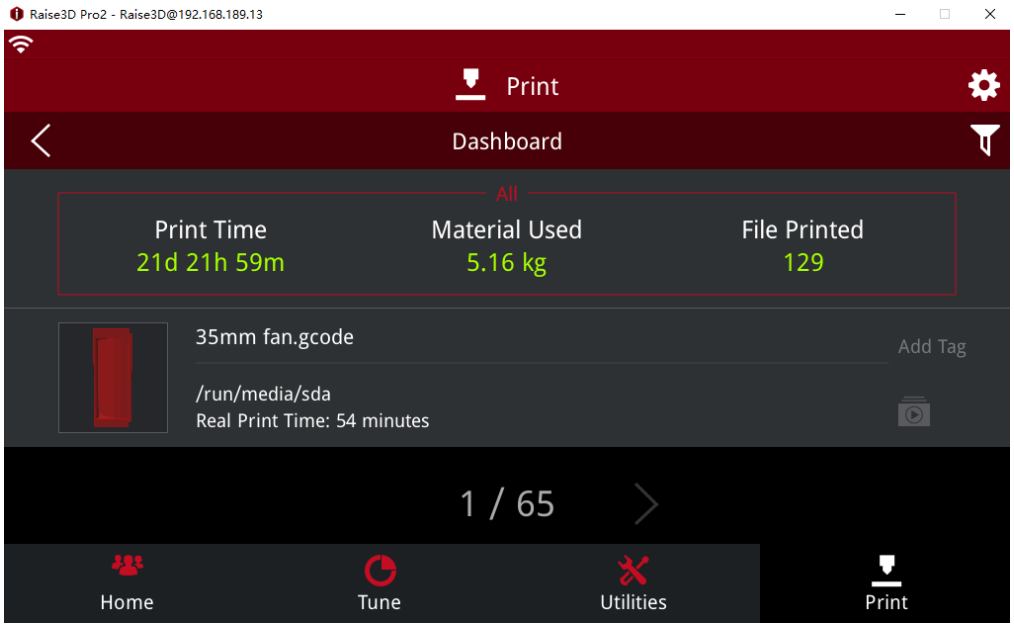


Figure 3.33 Dashboard.

Timelapse: the printing process will be saved as time-lapse photography as ***Figure 3.34***.

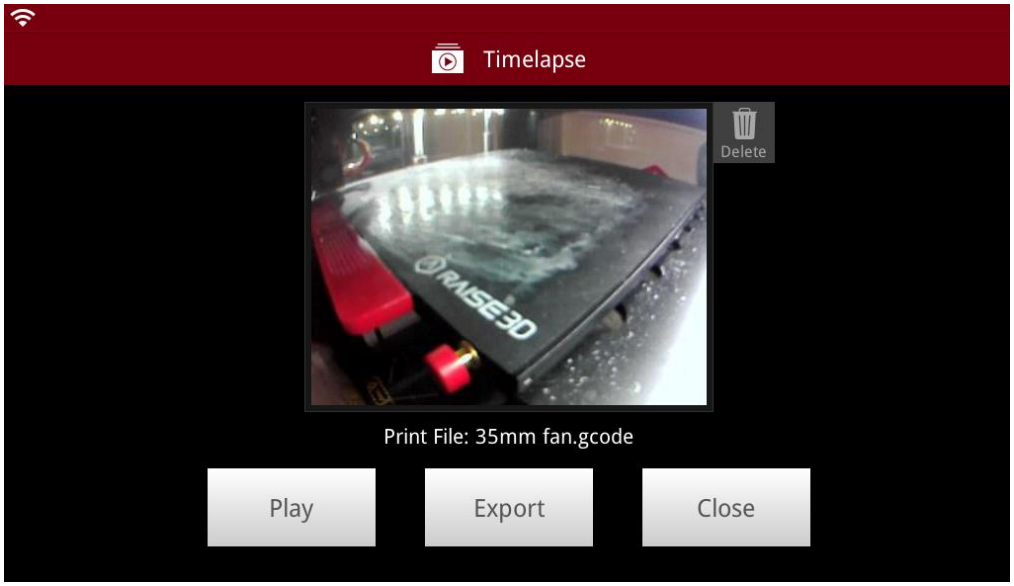


Figure 3.34: Timelapse.

Play refers to playing back the time-lapse video.
Export refers to exporting the timelapse from ideaMaker to your computer.
Close refers to closing this interface.

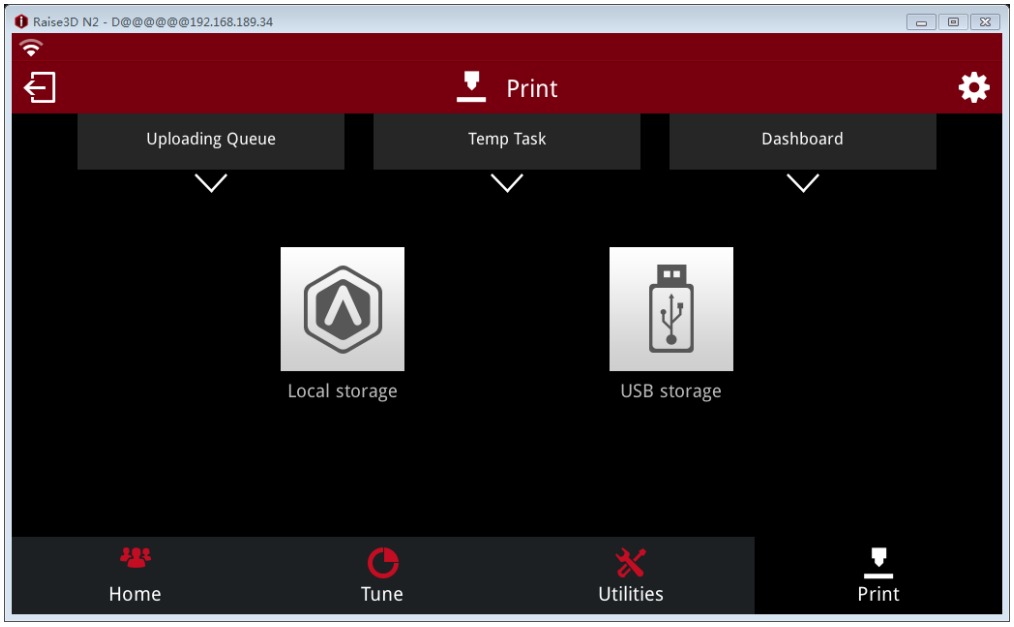


Figure 3.35: Print.

: WIFI connection status.

: camera icon, only shown when camera is enabled.

: printing status icon, only shown when print starts.

: exit icon.

: settings icon.

Machine name refers to the name of the machine and you can change it to be whatever you like.
Model refers to the type of the printer.
Serial Number refers to the identity number of the printer.
Version refers to the version of Raise Touch- touchscreen firmware currently used on this machine.

Update: the update information of touchscreen firmware.

Firmware version: the version of the motion controller board firmware.

Storage Available: the remaining memory space can be used in the machine.

About: included the feedback address.

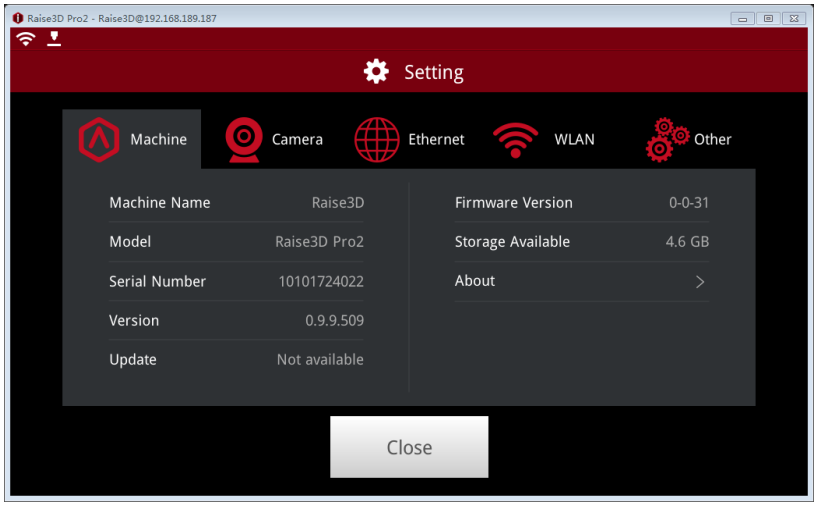


Figure 3.36: Machine.

Camera function is for Pro2 or Pro2 Plus.

Step 1: Choose a camera on the printer ***Figure 3.37***.

Figure 3.37: Choose a camera on touch screen.

Step 2: Connect to this camera as ***Figure 3.38.***

Figure 3.38: Connect to the chose camera.

Step 3: Take photo and save it in your computer ***as Figure 3.39.***

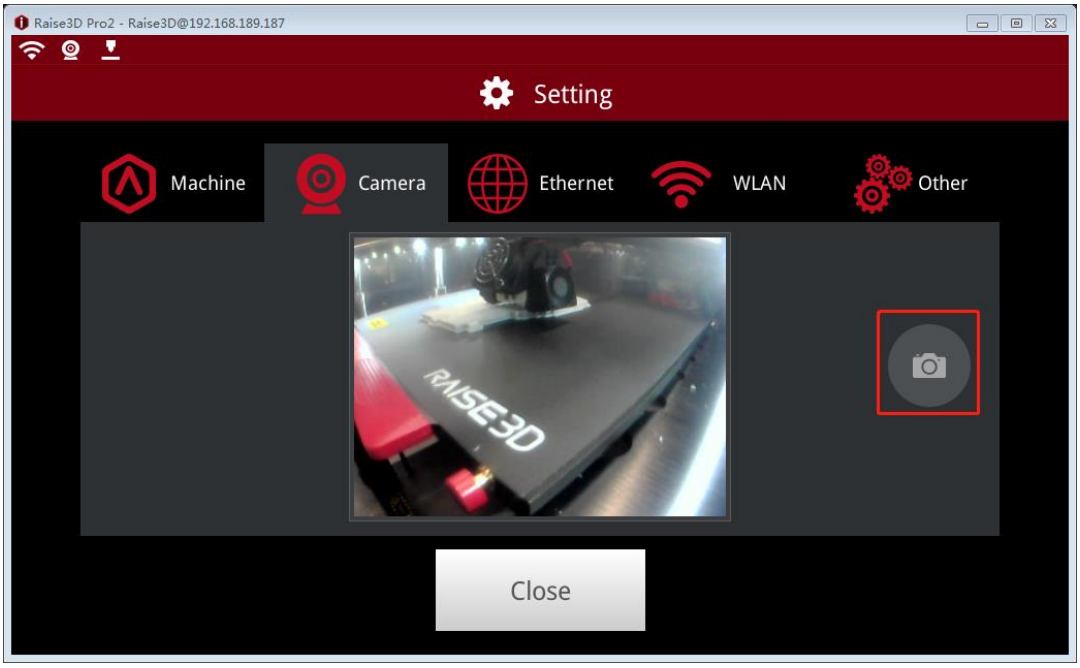


Figure 3.39: Take photo.

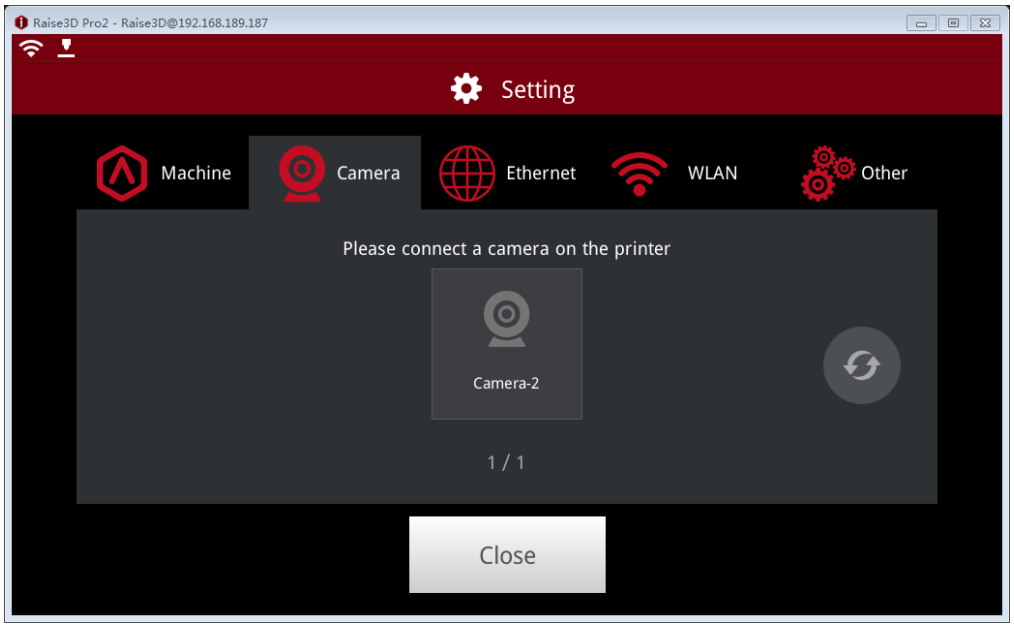


Figure 3.40: Camera.

Ethernet: connect printer and computer by net cable.

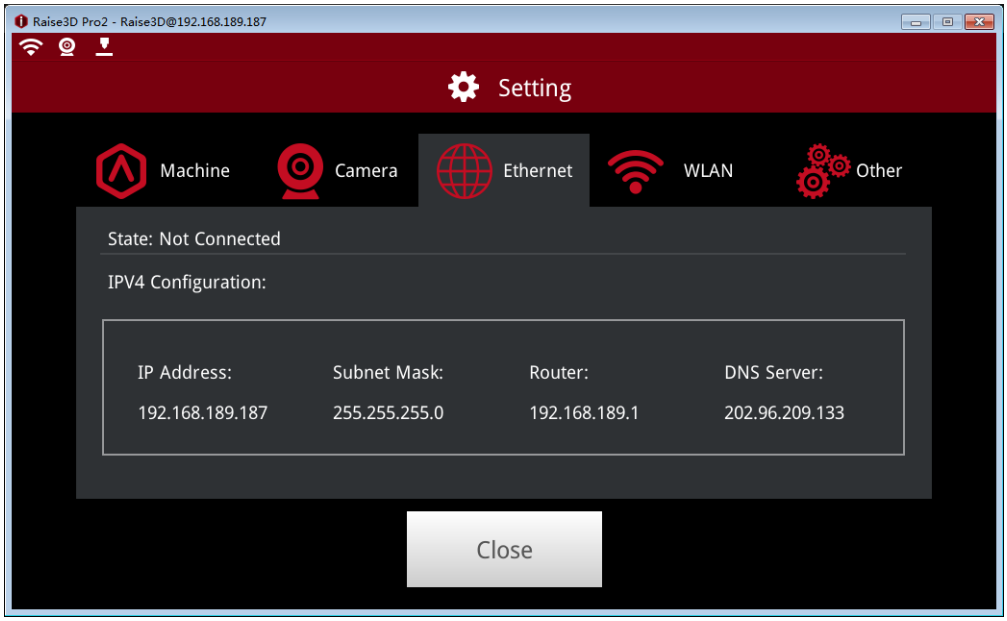


Figure 3.41: Ethernet.

WLAN: connect printer and computer by WLAN.

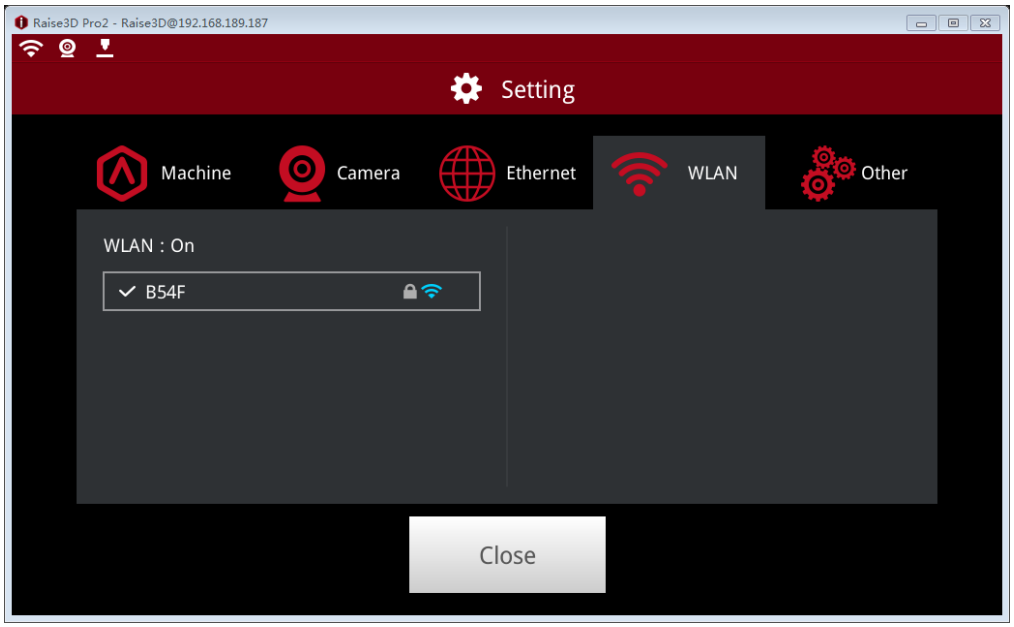


Figure 3.42: WLAN.

Other:

Brightness: control the brightness of the screen, you can adjust it by pressing the "+" or "-" button.

Language: the language applied to your printer.

Number of Nozzles: the number of the nozzle.

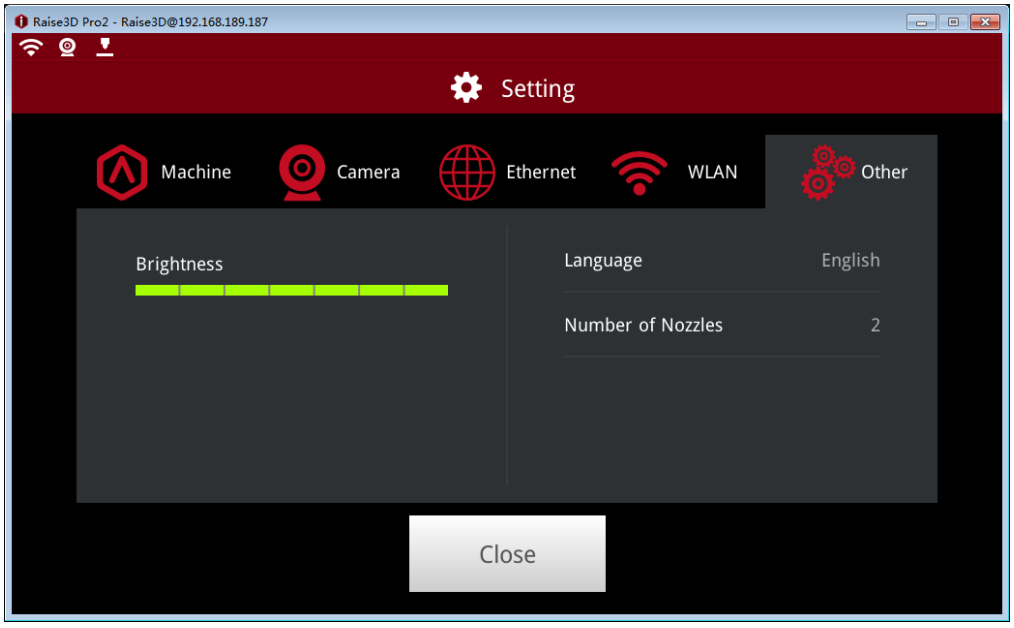


Figure 3.43: Other.

4 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.

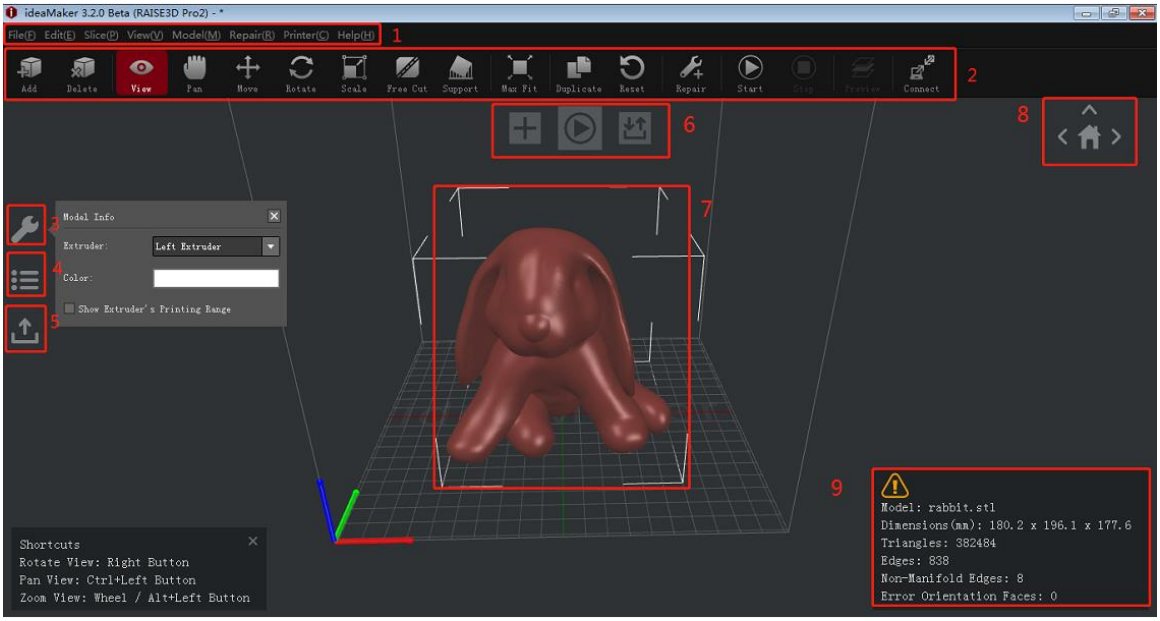
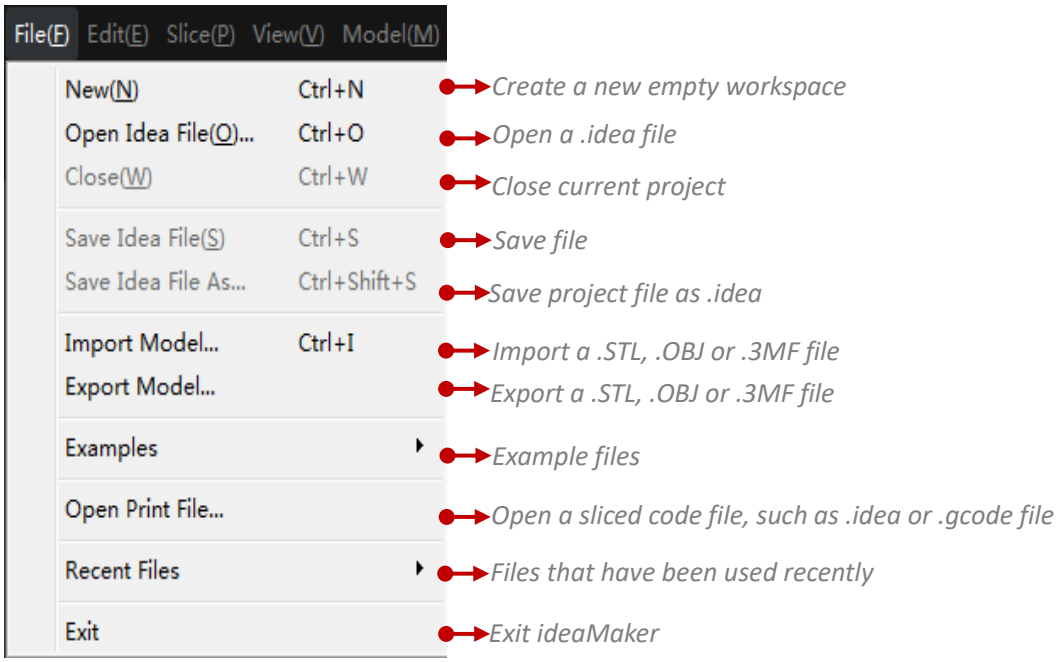


Figure 4.1: An overview over the ideaMaker interface.

4.1 Menu Bar

Menu bar includes all the operation commands and advanced setting.

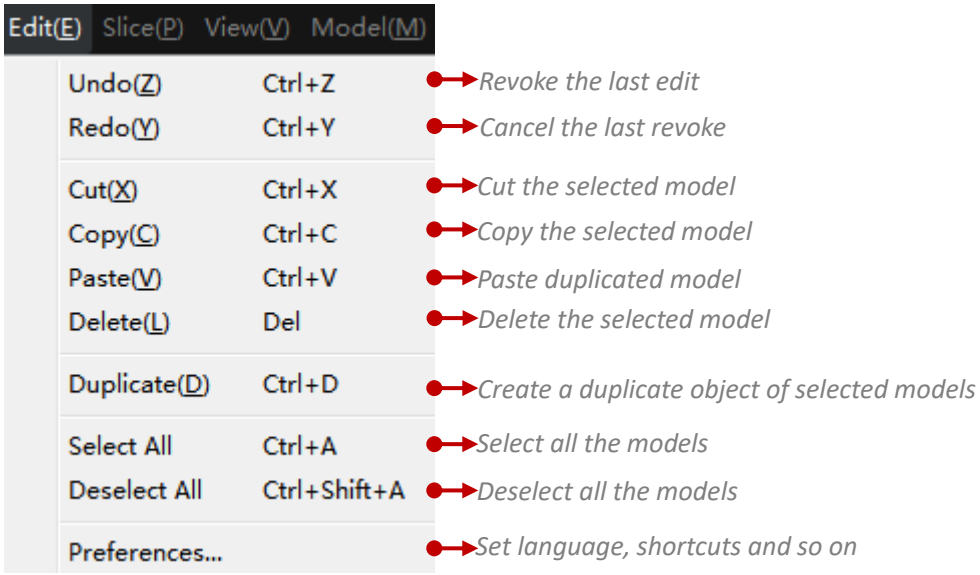
1>File



Note:

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

2>Edit



Note:

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

(1) General

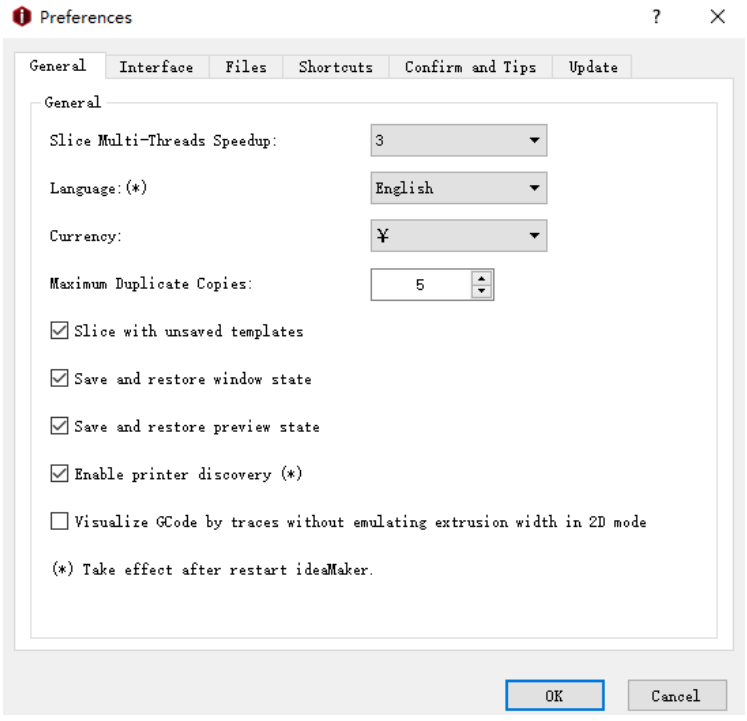


Figure 4.2: The General tab of Preference.

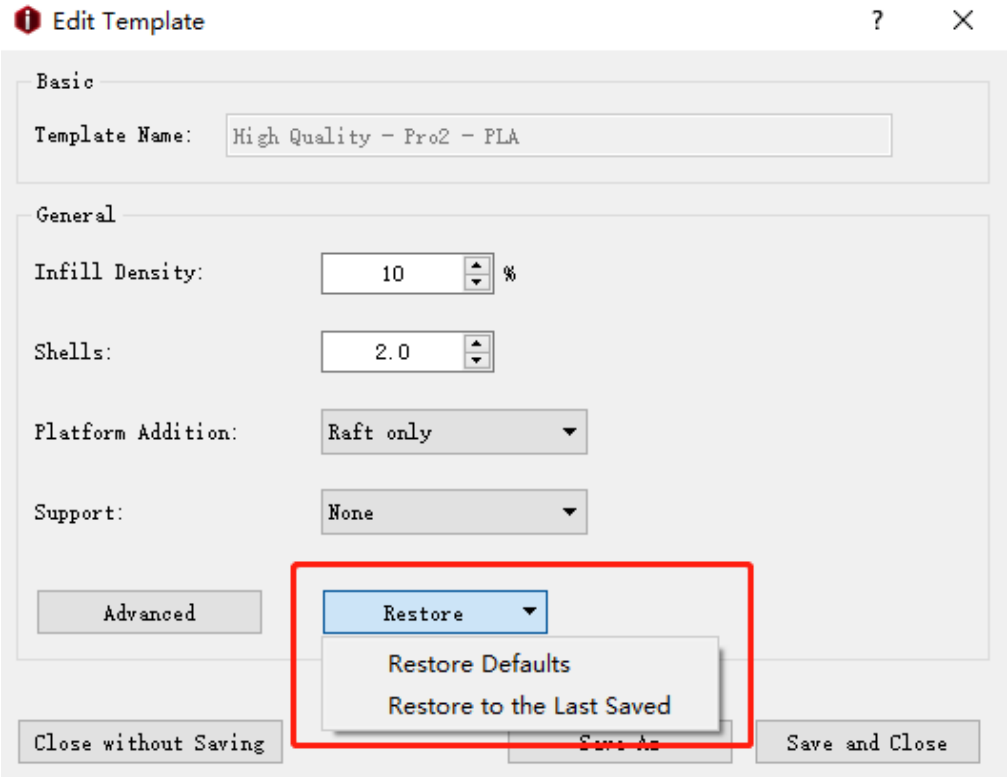
Slice Multi-Threads Speedup refers to the speed of slicing, the greater the ***Multi-Threads Speedup*** value, the faster the slicing speed. This value depends on your computer's CPU threads.

Language refers that you can select a language for your software.

Currency refers that you can select a currency unit for your software.

Maximum Duplicate Copies refers that you can increase the maximum value for models duplication, the maximum value is 1000, and the minimum is 1.

Slice with unsaved templates refers to carrying out a slicing without saving the settings edited. With this function enabled, you can select ***Restore Defaults*** or ***Restore to the Last Saved***.



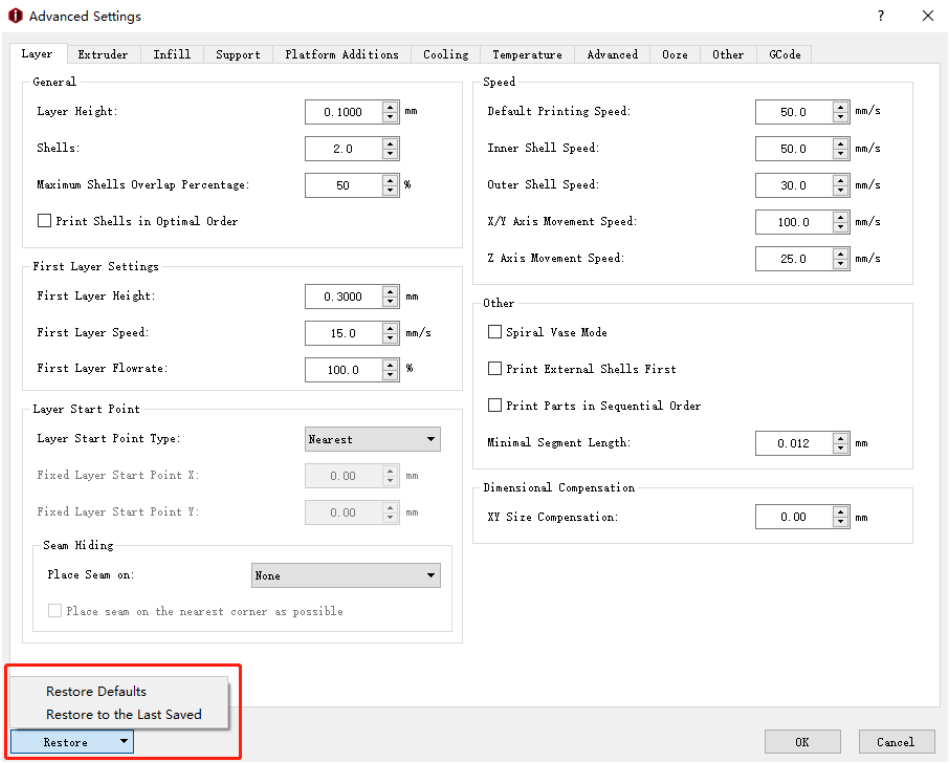


Figure 4.3: Enable "Slice with unsaved templates".

Save and restore window state refers that the window state can be saved and restored with this function enabled.

Save and restore preview state refers that the preview state can be saved and restored with this function enabled.

Enable printer discovery refers that with this function enabled ideaMaker will automatically search for printers under the same LAN as computer.

Visualize Gcode by traces without emulating extrusion width in 2D mode refers that enabled this function the tool paths will not be displayed with emulating extrusion width. It shows traces clearly.

Figure 4.4: Unable this function

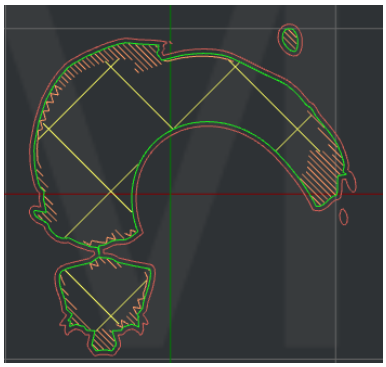


Figure 4.5: Enabled this function

(2) Interface

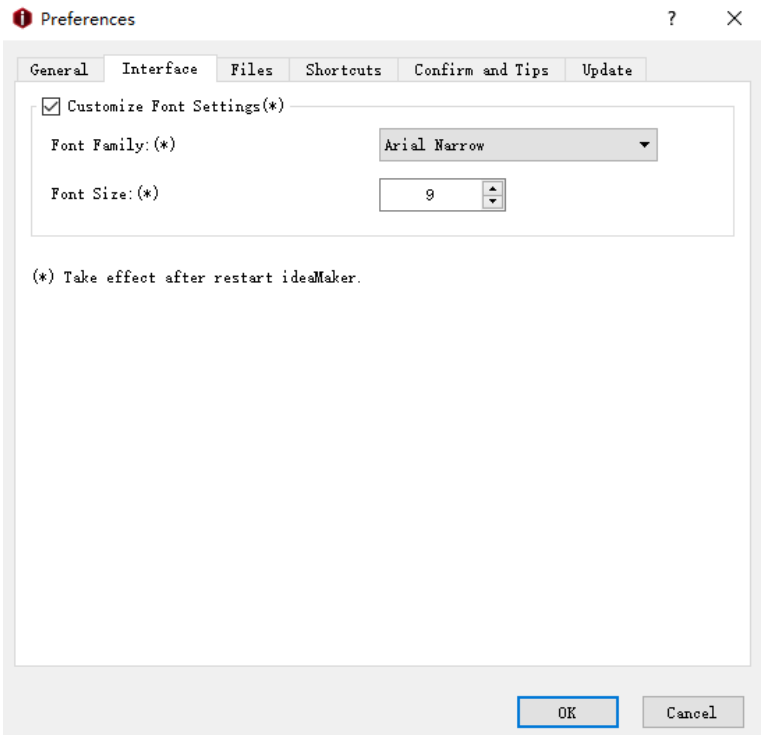


Figure 4.6: The Interface tab of Preference.

Customize Font Settings refers that with this function enabled you can adjust the font size and type of the interface in ideaMaker.

Font Family refers to the font type of the interface in ideaMaker.

Font Size refers to the font size of the interface in ideaMaker.

(3) Files

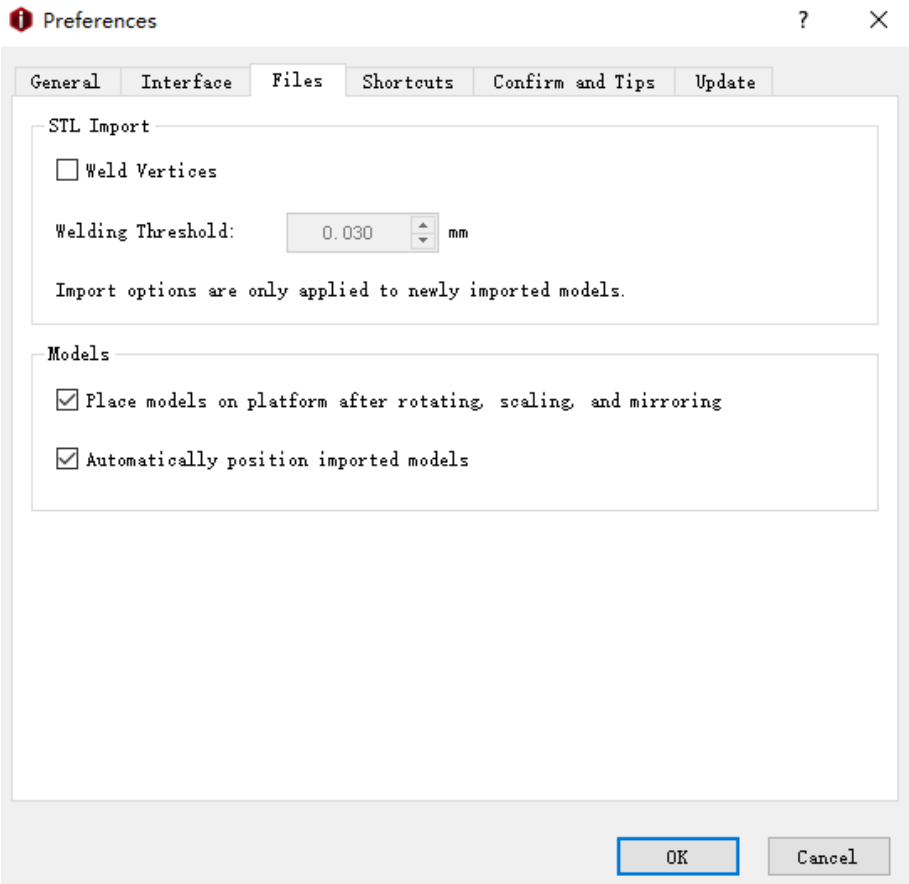


Figure 4.7: The Files tab of Preference.

Weld Vertices refers that the nearby points will be welded with this function enabled.

Place models on platform after rotating, scaling and mirroring refers that the model will be placed on the build platform after rotating, scaling and mirroring.

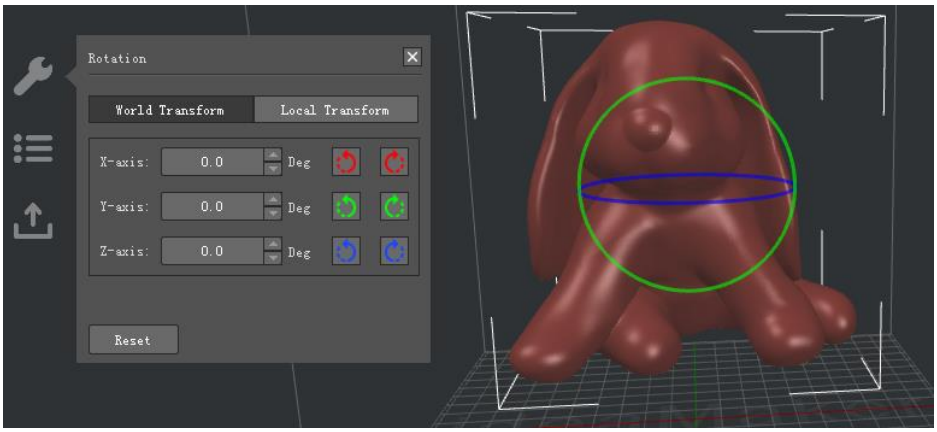


Figure 4.8: The model is floating in mid-air before rotating.

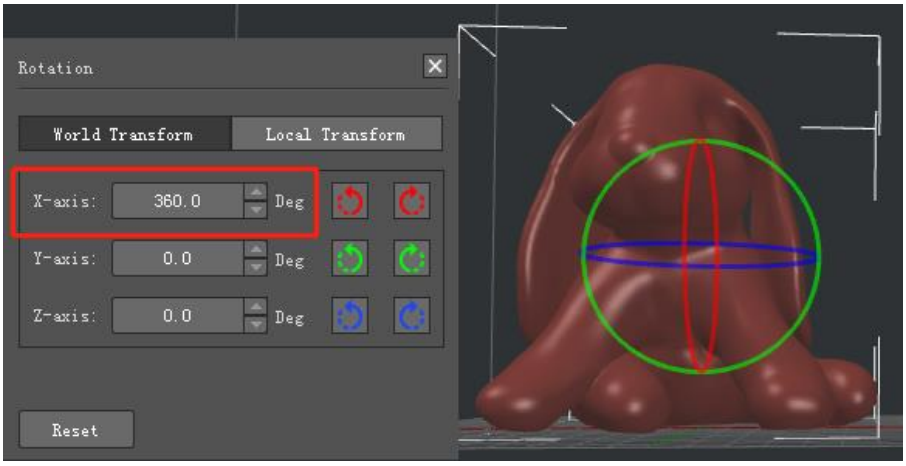


Figure 4.9: The model is placed on platform after rotating.

Automatically position import models refers that with this function enabled the models will be placed in the center of the build platform automatically.

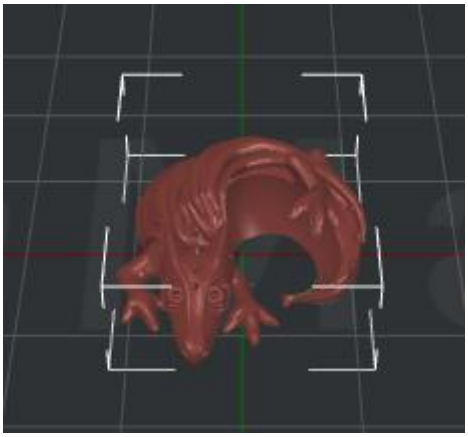


Figure 4.10: Enable the function of "Automatically position import models".

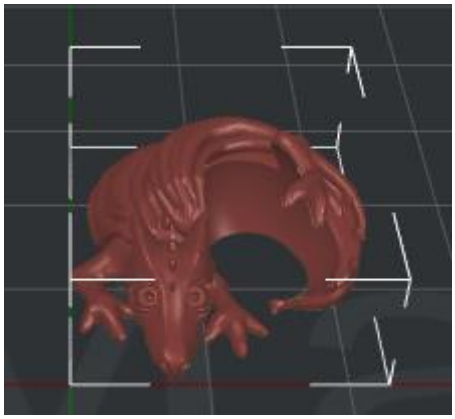


Figure 4.11: Unable the function of "Automatically position import models".

(4) Shortcuts

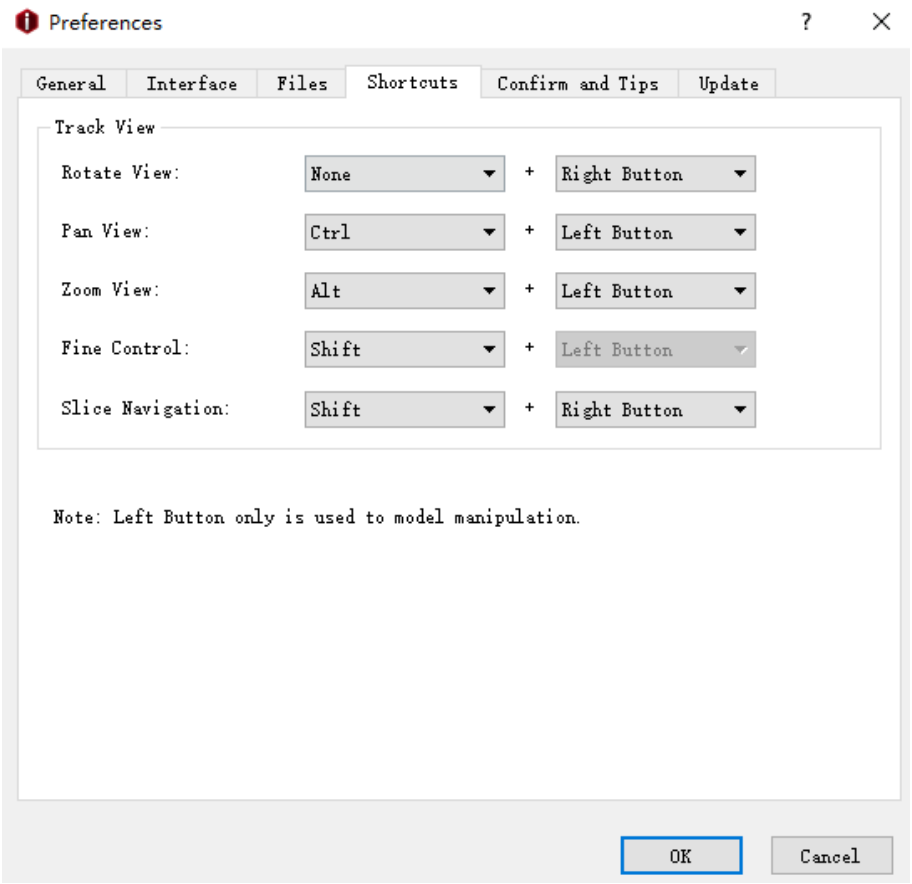


Figure 4.12: The Shortcuts tab of Preference.

Click left button and move mouse to rotate model.

Click ***Ctrl***, left button and move mouse to pan the view.

Click ***Alt***, left button and move mouse to zoom the view.

Click ***Shift***, left button and move mouse the fine control of mouse's movement.
Click ***Shift***, right button and move mouse to control the preview .gcode file. Move mouse up/down is to control ***layer*** in ***Slice Preview***, move mouse left/right is to control ***Step*** in ***Slice Preview***.

(5) Confirm and Tips

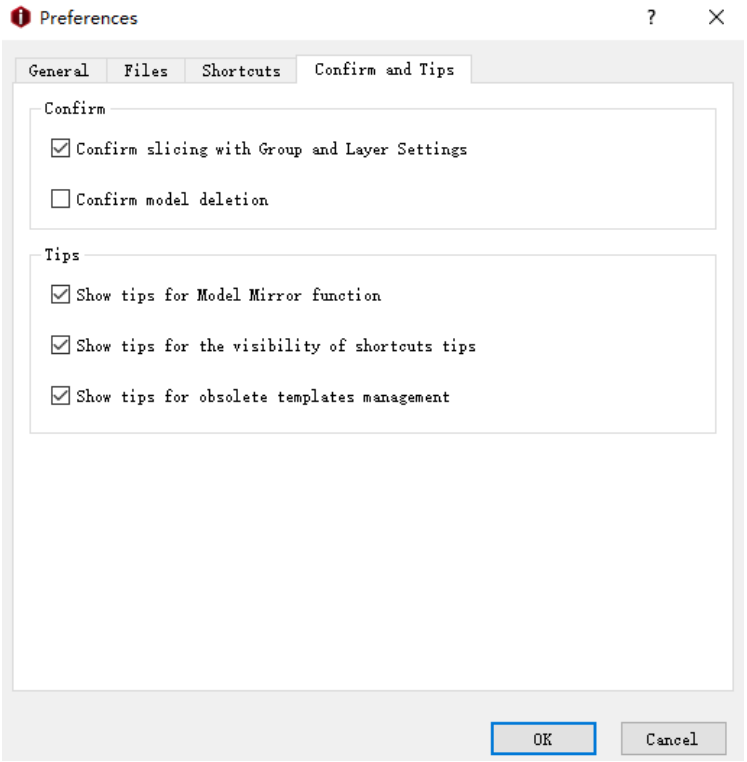


Figure 4.13: The Confirm and Tips tab of Preference.

Confirm:
Confirm slicing with Group and Layer Settings refers that with this function enabled ideaMaker will pop-up a dialog to confirm whether continue slice with override parameters in Group and Layer Setting as ***Figure 4.13***.

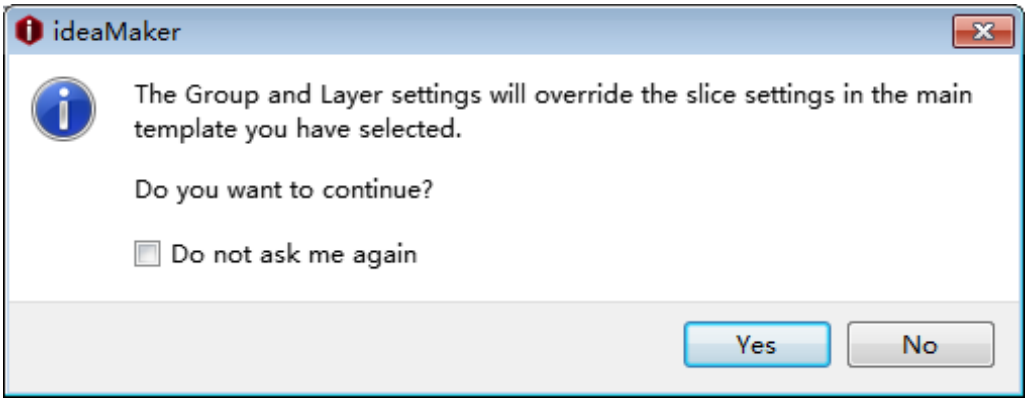


Figure 4.14: Confirm slicing with Group and Layer Settings.

Confirm model Deletion refers that with this function enabled ideaMaker will pop-up a dialog to confirm whether delete the model you selected as ***Figure 4.14***.

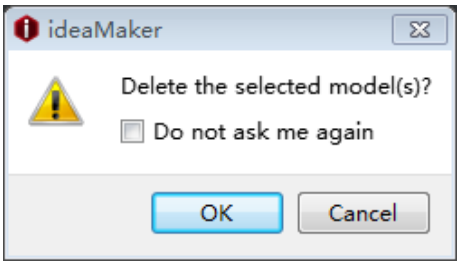


Figure 4.15: Confirm model Deletion.

Tips:

Show tips for Model Mirror Function refers that with this function enabled ideaMaker will pop-up a dialog to remind the tips of model mirror function as ***Figure 4.15***.

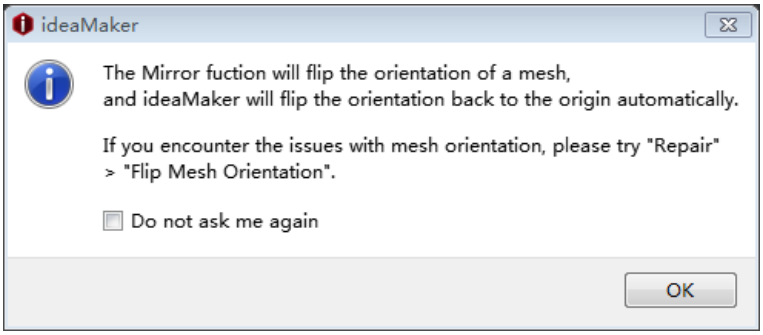


Figure 4.16: Show tips for Model Mirror Function.

Show tips for visibility of shortcuts tips refers that with this function enabled ideaMaker will pop-up a dialog to remind the tips for visibility of shortcuts function.

Show tips for obsolete templates management refers that with this function enabled ideaMaker will pop-up a dialog to remind the tips for obsolete templates management.

(6) Confirm and Tips

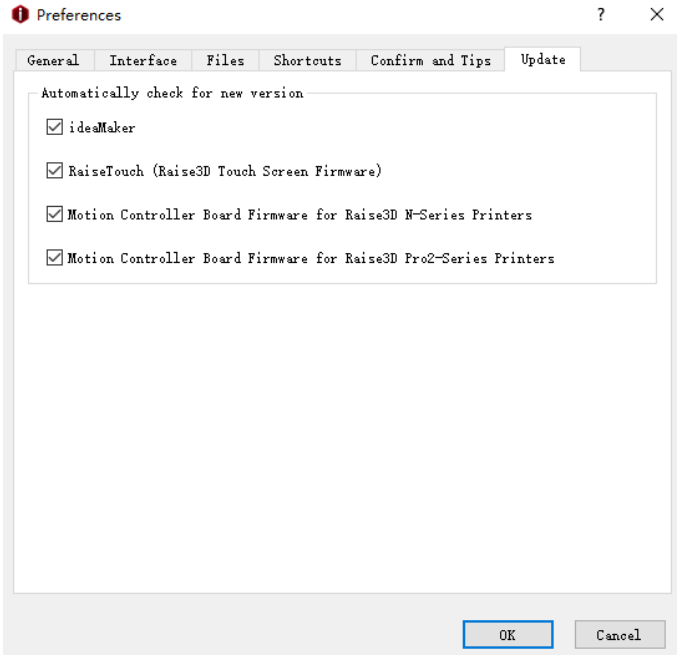


Figure 4.17: The Update tab of Preference.

Automatically check for new version

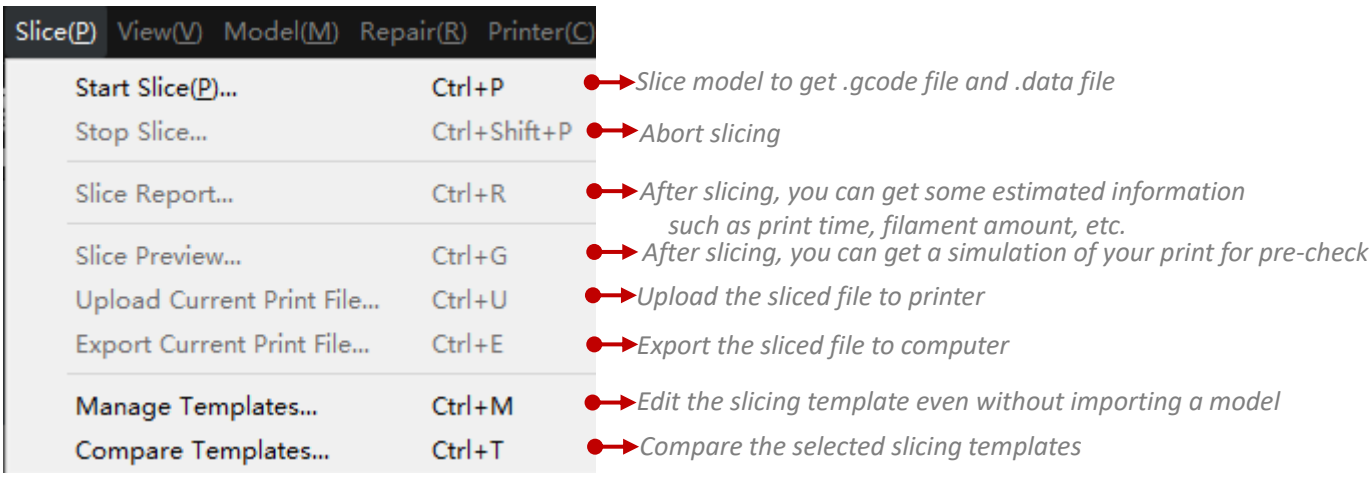
ideaMaker refers that with this function enabled ideaMaker will check the latest version of ideaMaker automatically.

RaiseTouch (Raise3D Touch Screen Firmware) refers that with this function enabled ideaMaker will check the latest version of RaiseTouch automatically.

Motion Controller Board Firmware for Riase3D N-series Printers refers that with this function enabled ideaMaker will check the latest version of motion controller board firmware for N-series printer automatically.

Motion Controller Board Firmware for Riase3D Pro2-series Printers refers that with this function enabled ideaMaker will check the latest version of motion controller board firmware for Pro2-series printer automatically.

3>Slice



Compare Templates refers to comparing the selected templates. You can choose different printer, filament and template. And you also can import the templates from local disk.

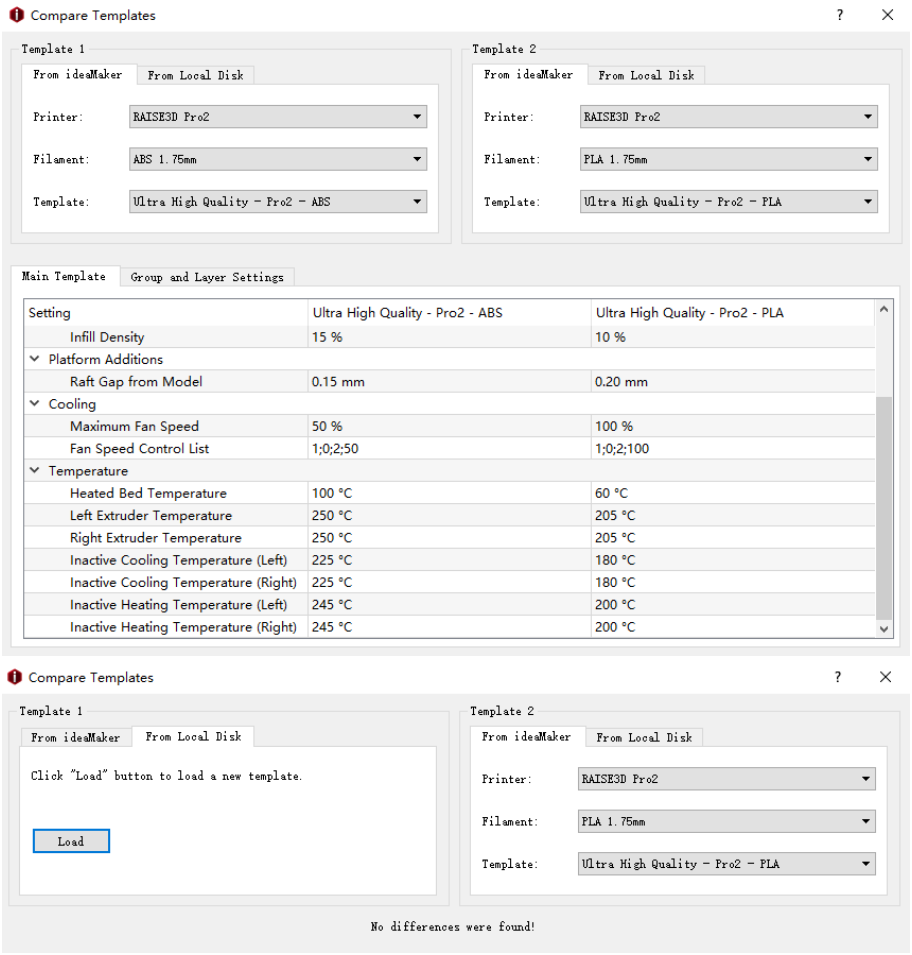
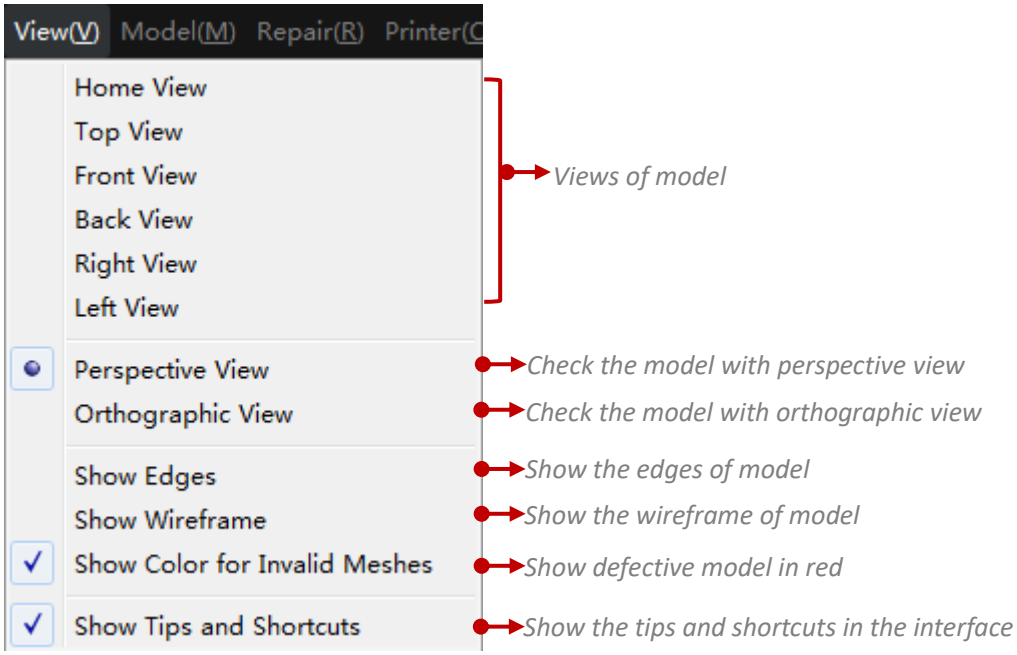


Figure 4.18: Compare Templates.

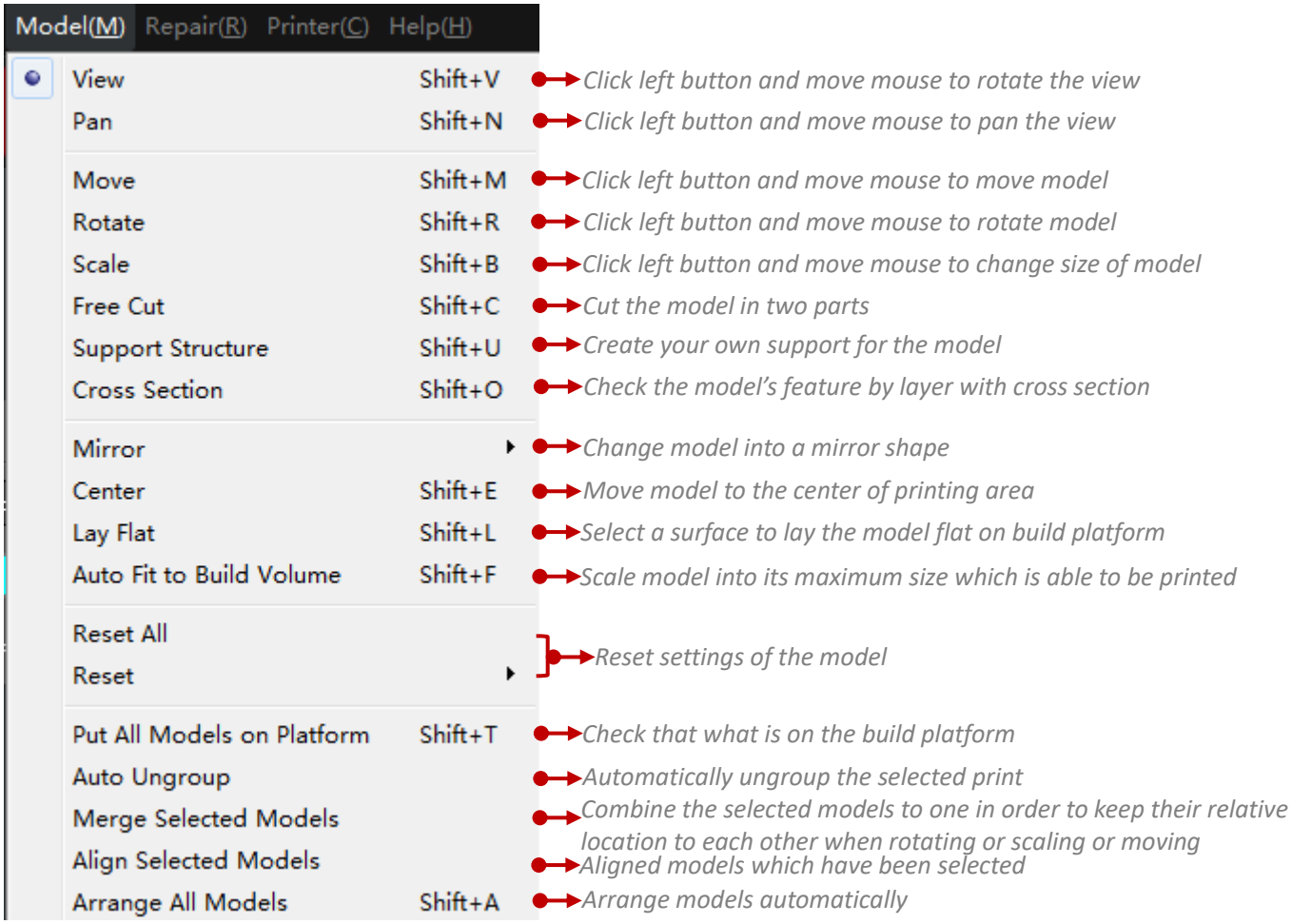
4>View



Note:

You can also change the model view freely by right clicking and dragging your mouse.

5>Model



Note:

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

Cross Section refers to check the model's features by layer.

Cross Section will divide the model into two parts. Click on ***Reverse*** to remain the upper part of the model.

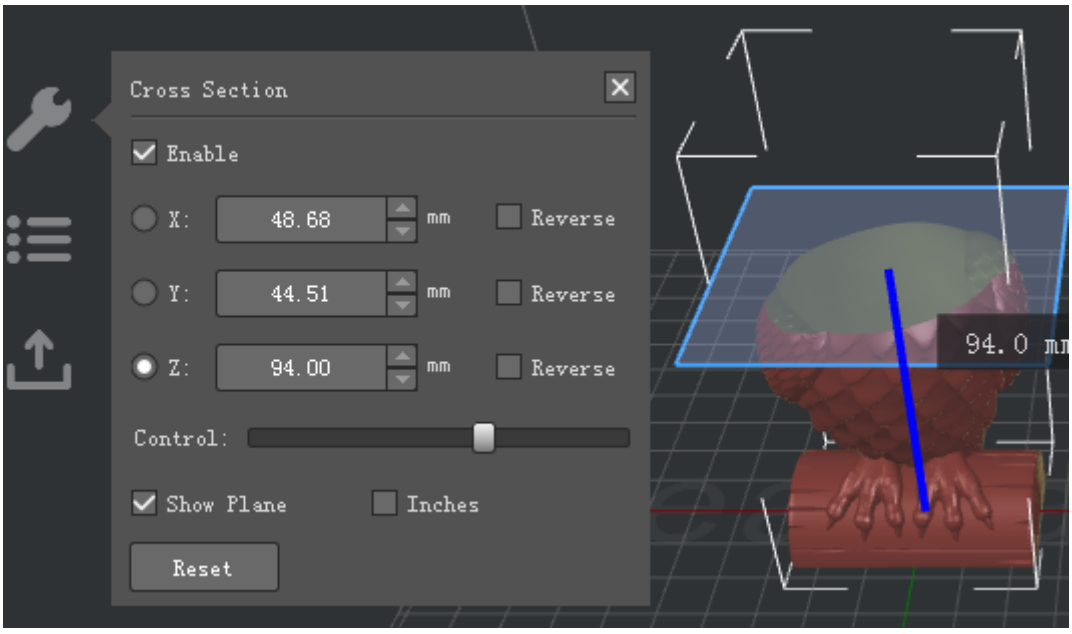


Figure 4.19: Cross Section in the Z axis direction.

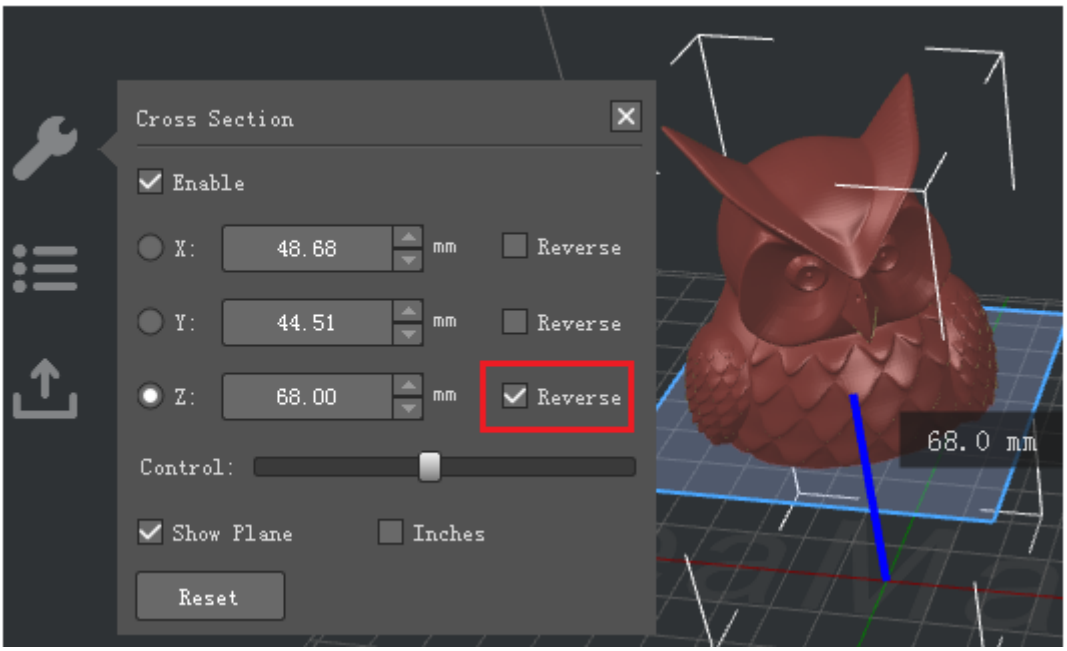


Figure 4.20: Cross Section in "Reverse" of Z axis direction.

Show Plane will toggle the visibility of the cross-section plane.

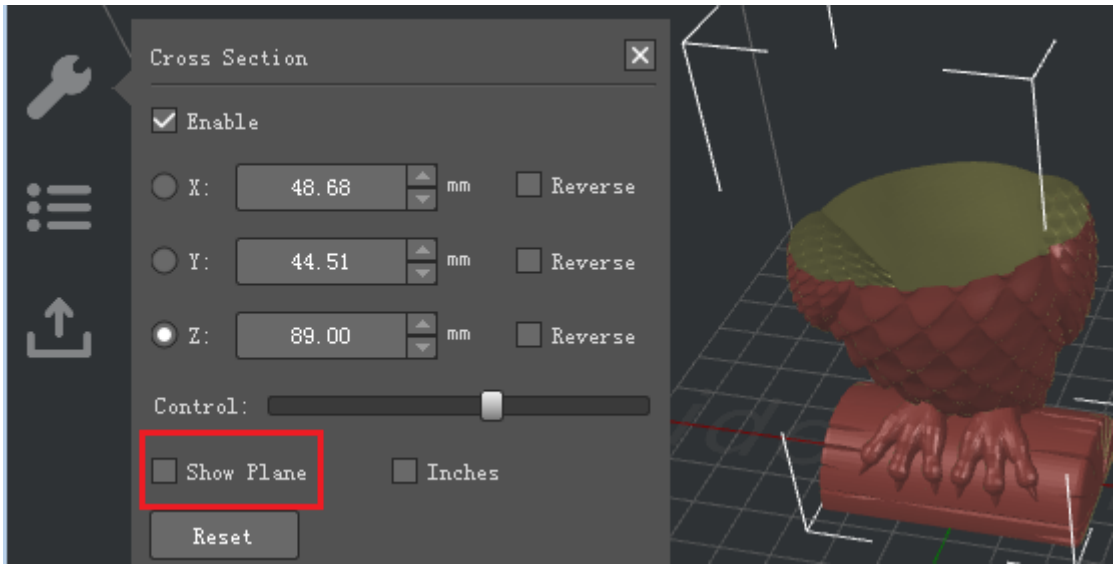


Figure 4.21: Cross Section without "Show Plane".

Inches will select the unit between inches or mm.

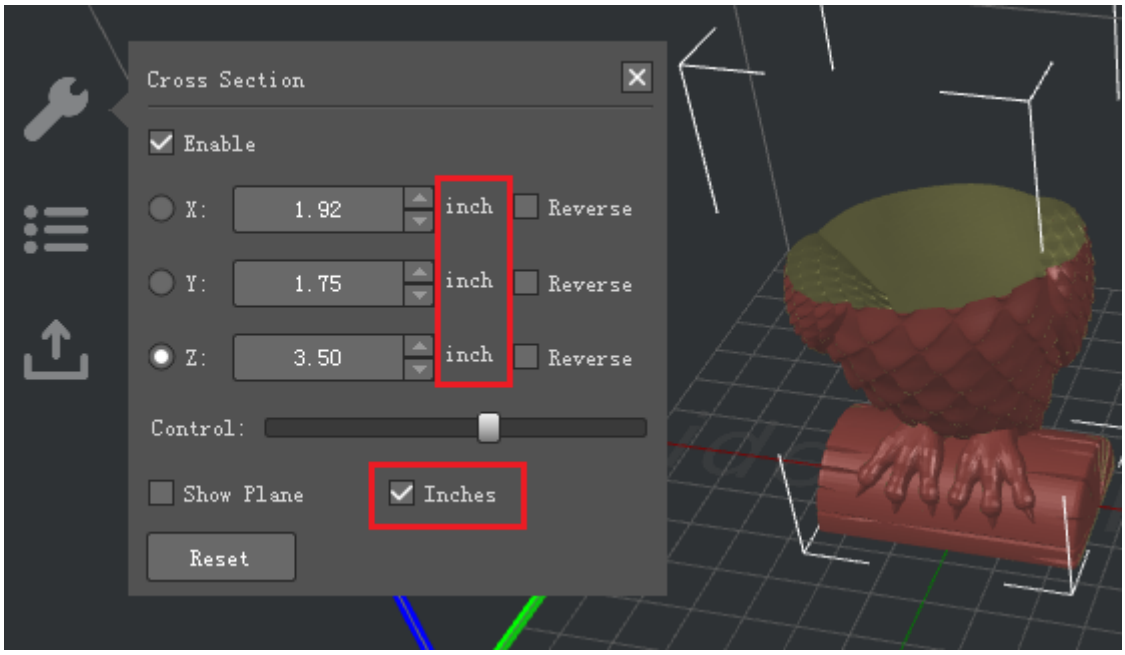


Figure 4.22: Cross Section with "Inches".

Note:

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

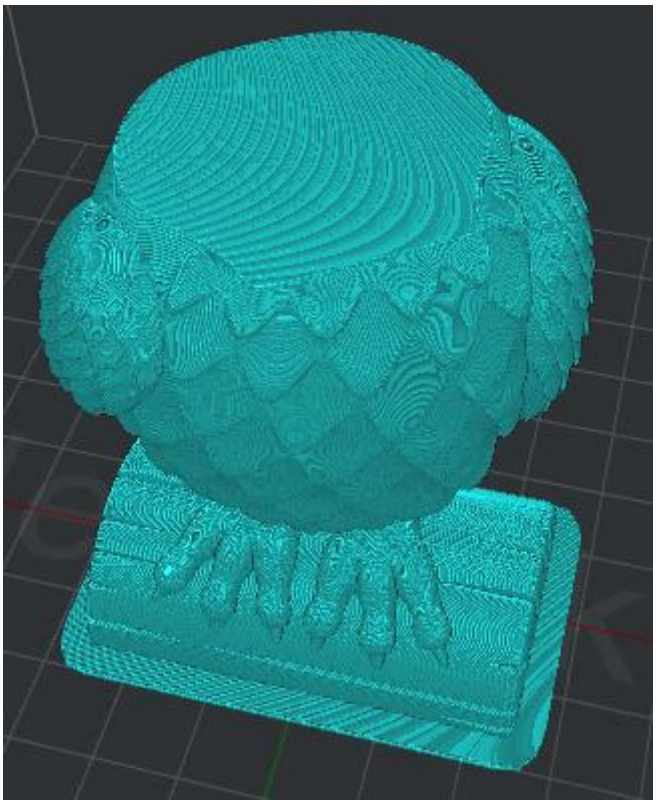


Figure 4.23: The slice preview with "Cross Section".

Lay Flat refers to selecting a surface to lay the model flat on the build platform.

Face Index refers to the face you selected.

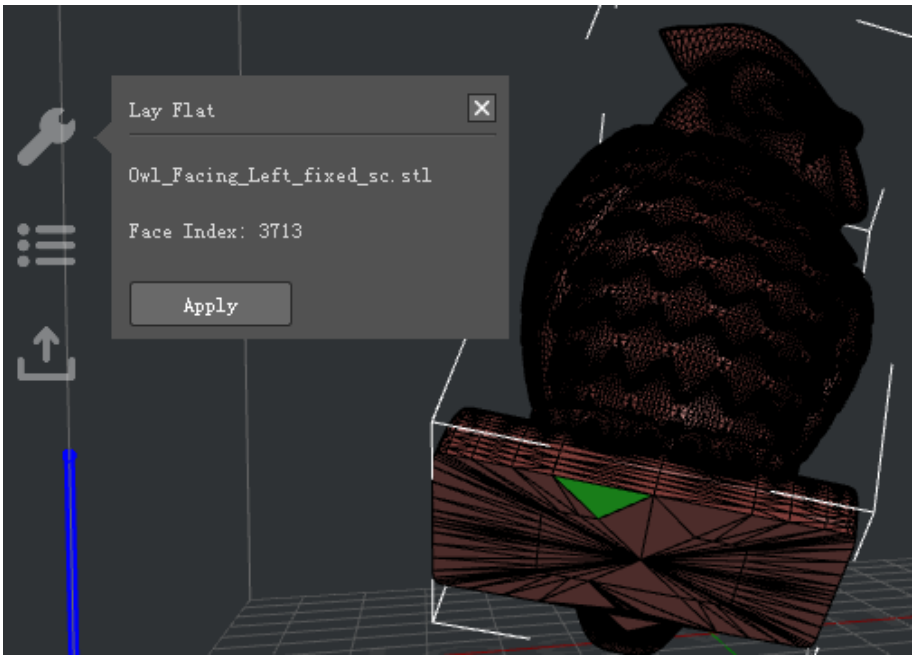


Figure 4.24: Click "Apply" to lay the model flat on the platform.

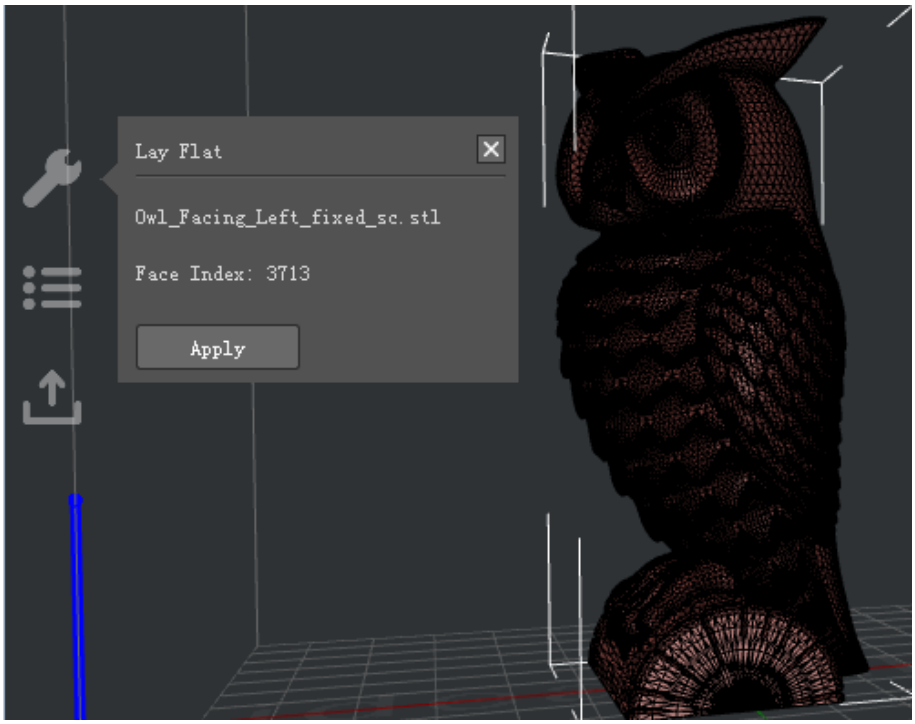
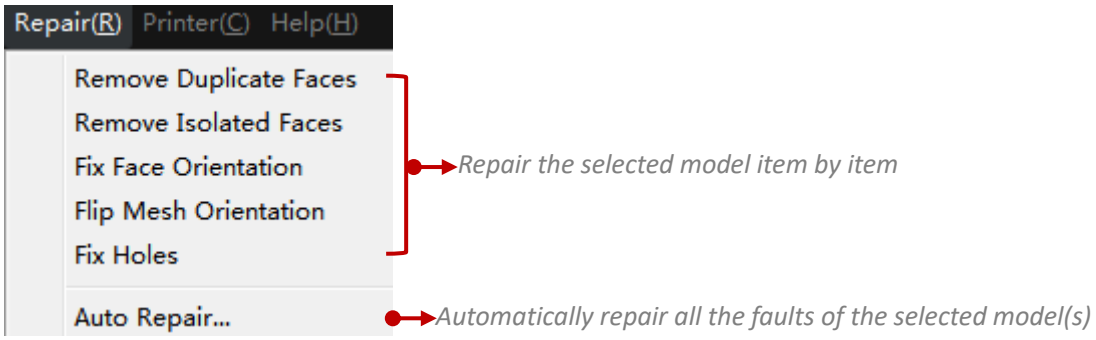


Figure 4.25: The model after "Lay Flat".

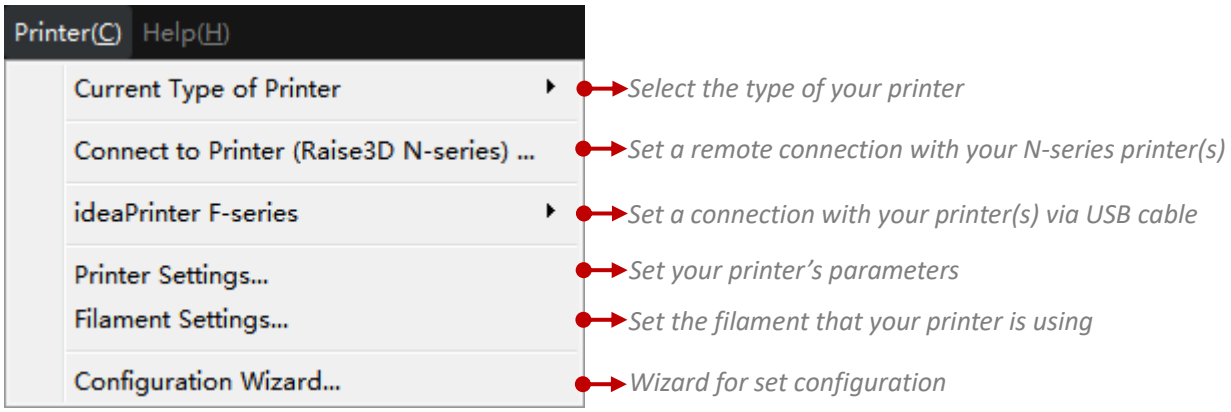
6> Repair



Note:

You can also find auto repair icon in tool bar.

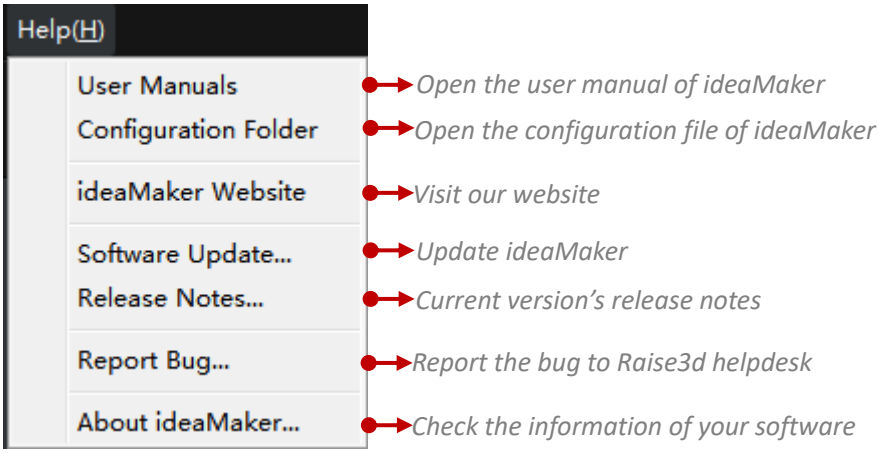
7>Printer



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*

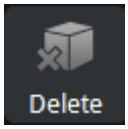


4.2 Tool Bar

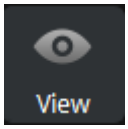
Tool buttons for slicing models, these buttons are shortcut of menu bar.

Figure 4.26: The tool bar.

: Add new .stl, .obj or .3mf 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 set your printer as a dual-extruder printer).

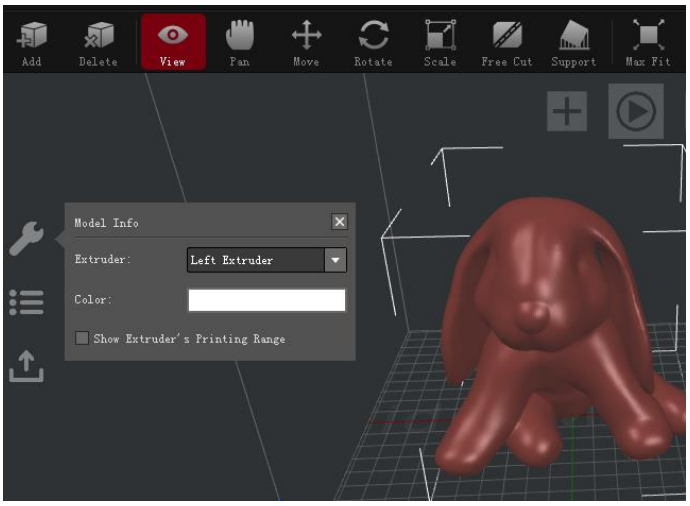


Figure 4.27: Check models with different extruders in different colors.

(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 holding the left button and dragging your mouse.

: When you enable this button, you can move the selected model in Z direction by left-dragging the blue arrow above the model with your mouse. And move the model freely in X and Y directions by holding the left button and dragging 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.

Arrange All Models refers to arranging the models automatically when you print several models in one time.

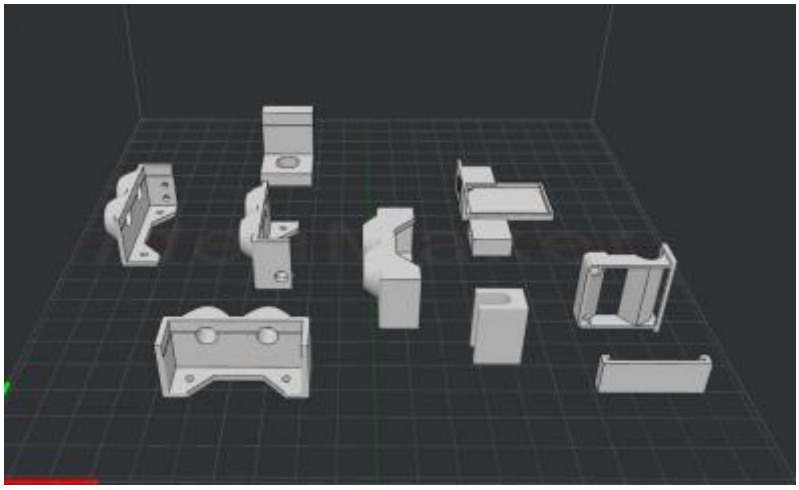


Figure 4.28: Unable the function of the "Arrange All Models".

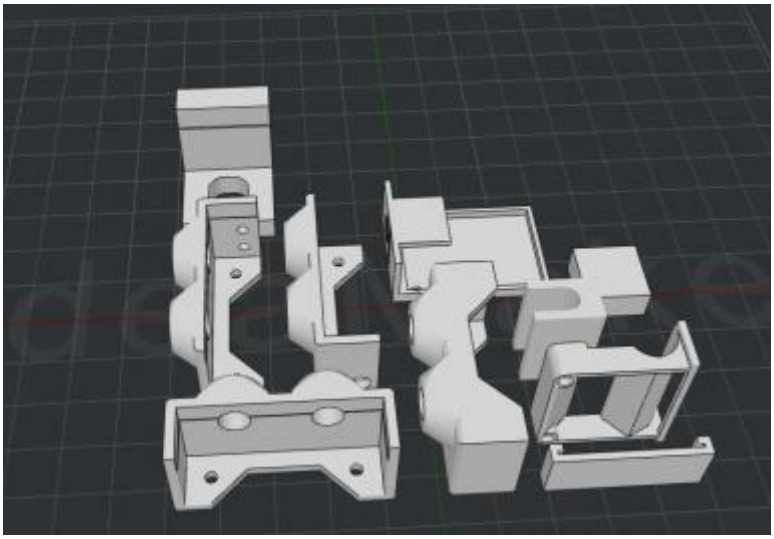


Figure 4.29: Enable the function of the "Arrange All Models".

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

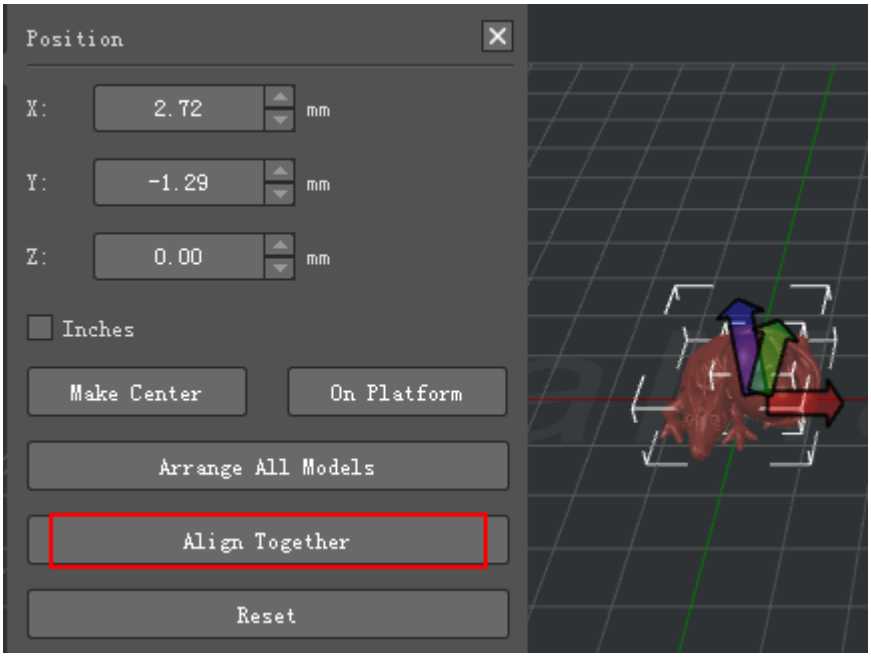
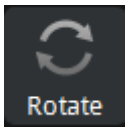


Figure 4.30: You can drag the selected model freely in X & Y directions with left-click. And click the blue arrow to drag the model in Z direction.



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

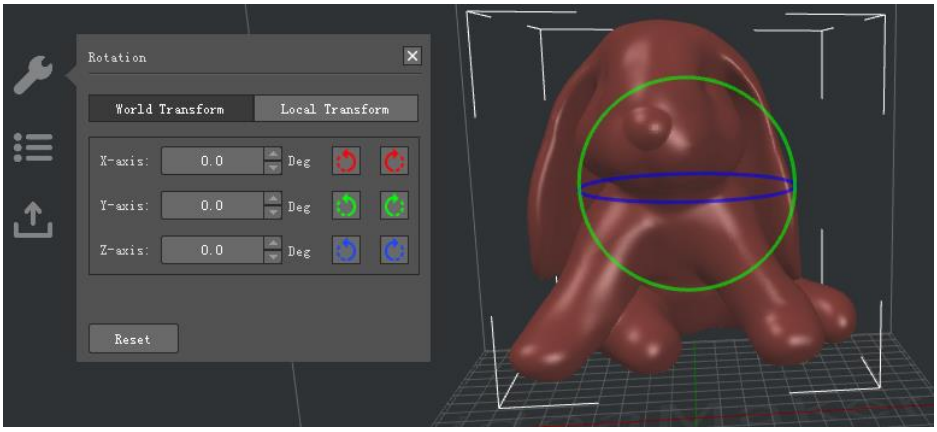


Figure 4.31: You can use both the color rings and left values to rotate the model.

: When you enable this button, you can amplify model's size by holding the left button and dragging 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 scaling the width, depth and height in the same scale.

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

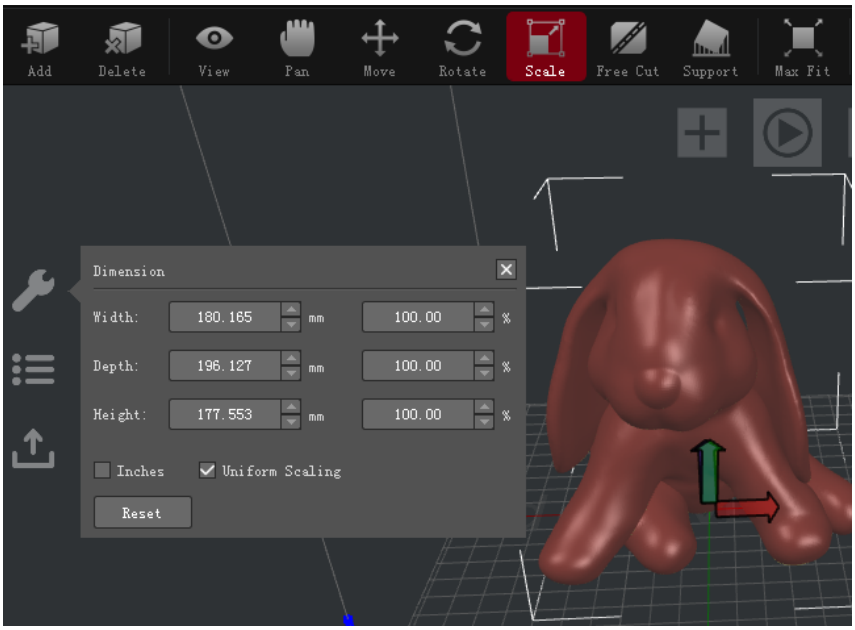


Figure 4.32: You can use both the color arrows and the spin boxes in the Dimension panel to scale the model.

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

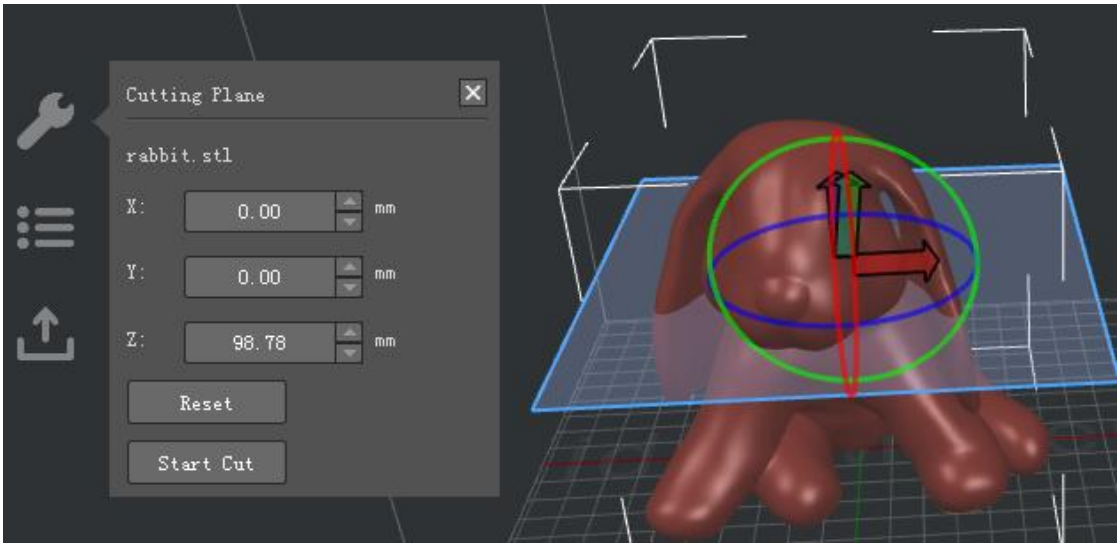


Figure 4.33: Click "Start Cut" to cut the model into individual pieces.

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

Option 1: Move the red, blue, green arrow.

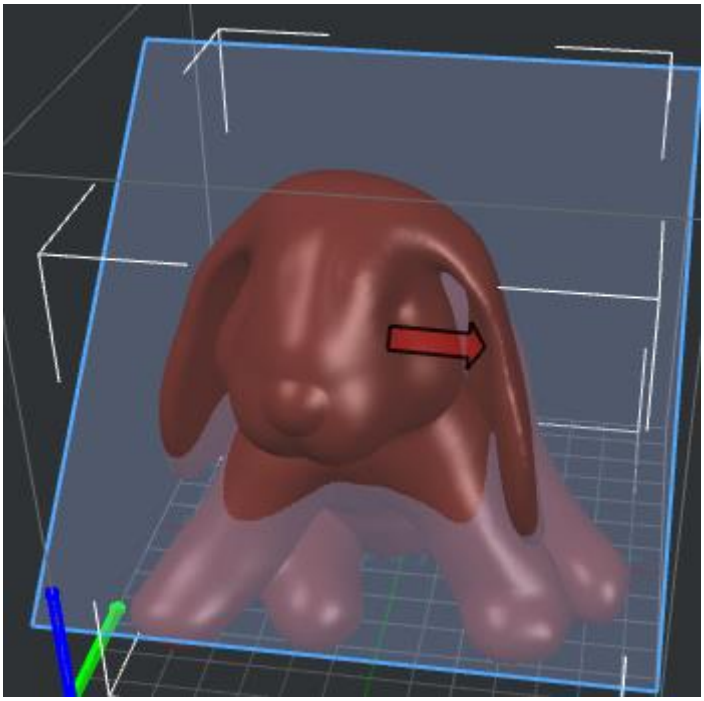


Figure 4.34: Click red arrow to move the "Cutting Plane".

Option 2: Rotate the red, blue, green circle.

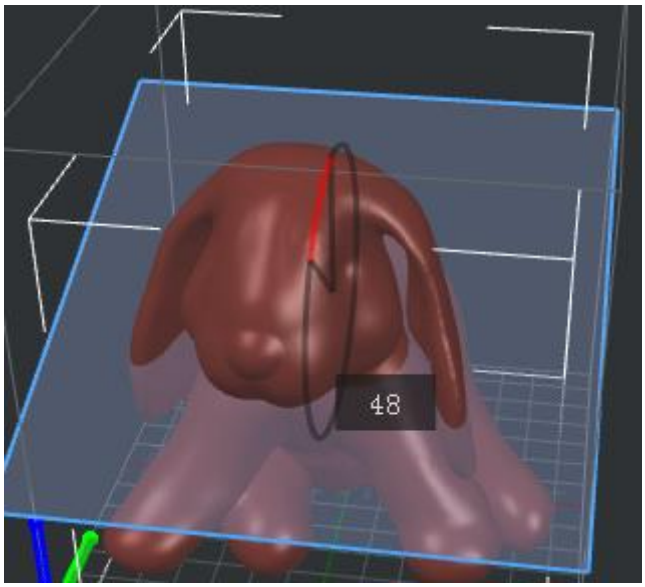


Figure 4.35: Click red circle to rotate the "Cutting Plane".

Option 3: Input the exact value.

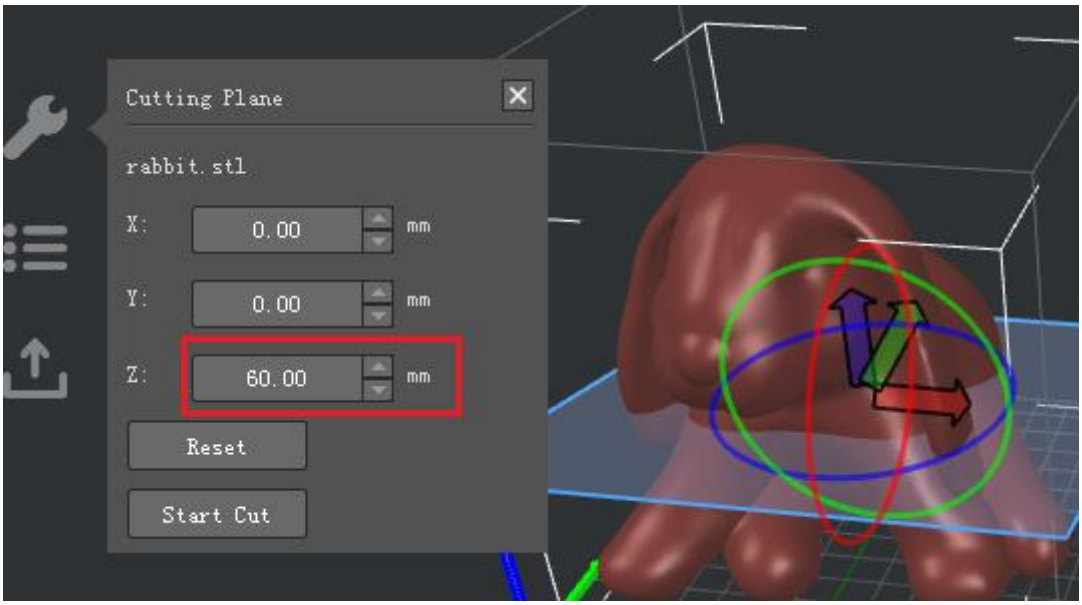


Figure 4.36: Input values to move the "Cutting Plane".

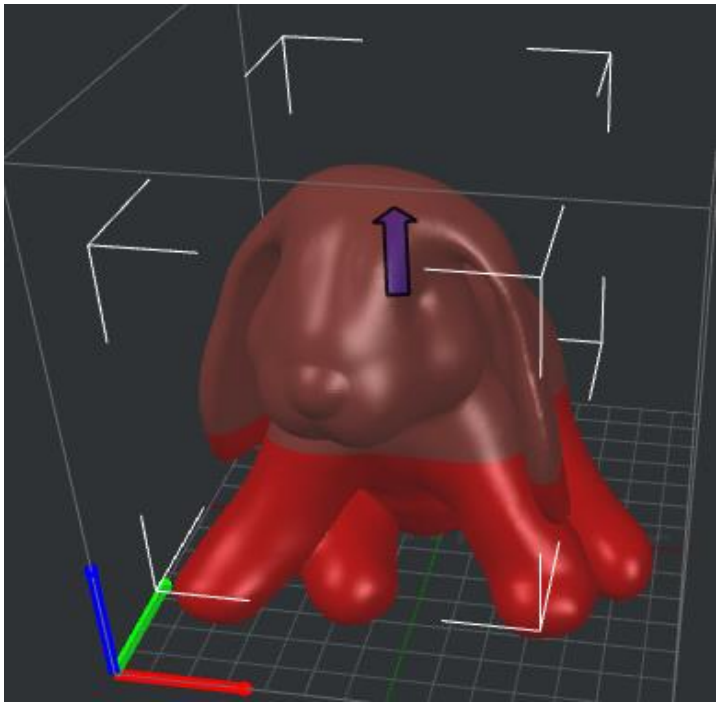


Figure 4.37: The selected model has been cut into two parts.

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

Auto Supports:

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

Note:

Pillar Size only works for the support type of **Pillar**.

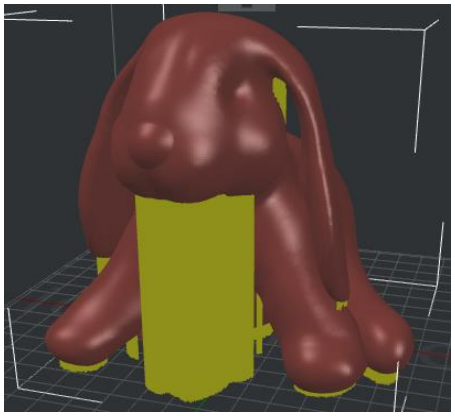


Figure 4.38: Set "Pillar Size" to be 1mm.

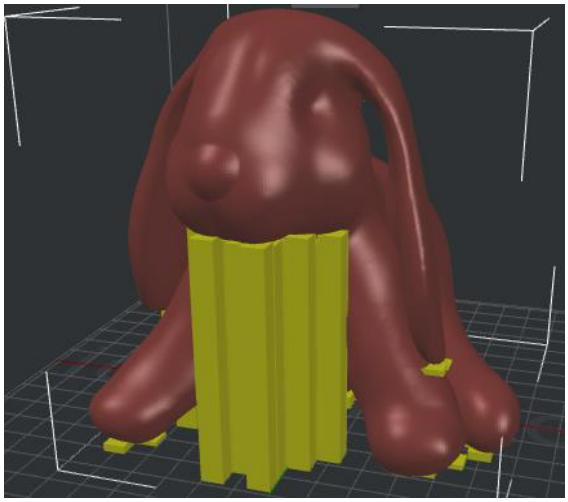


Figure 4.39: Set "Pillar Size" to be 10mm.

Overhang Angle refers to the value which decides whether the model needs support or not. If the real overhang angle is smaller than this value, support will not be generated.

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

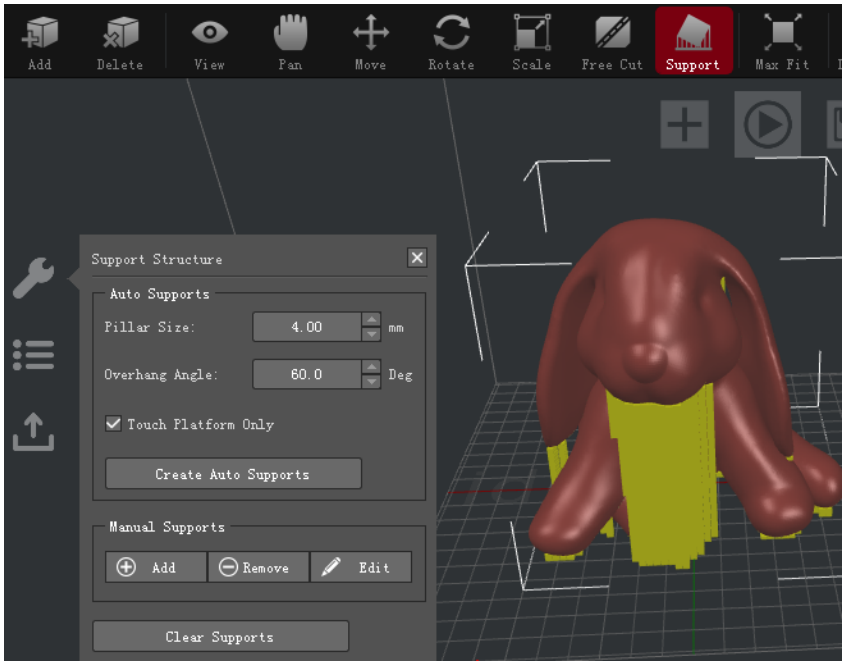


Figure 4.40: You can set the "Pillar Size" and "Overhang Angle" first. And try "Create Auto Supports". Then adjust the support with "Manual Supports".

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.

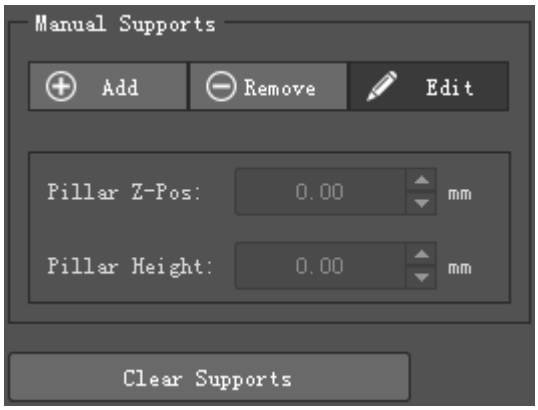
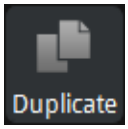


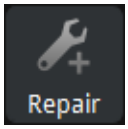
Figure 4.41: These two options will only appear after you click the "Edit" button.

: When you click this button, the selected model will be re-sized to maximum size that your printer can print.



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

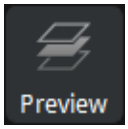
: Revoke all the settings of the selected model.



: 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.

4.3 Operation property

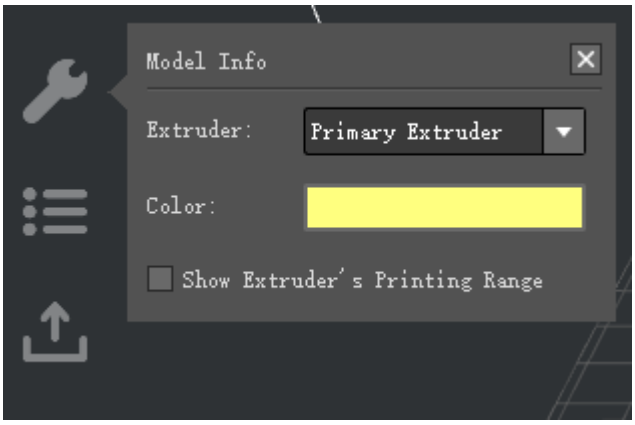


Figure 4.42: The information shown here depends on which function you are using.

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

4.4 Model list

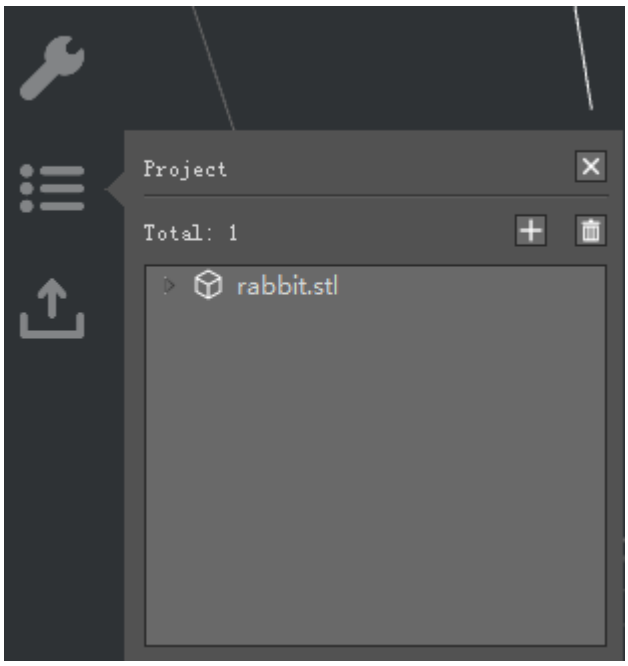


Figure 4.43: You can also add or delete models here.

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.

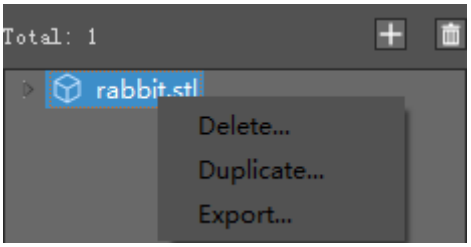


Figure 4.44: Right-click the blue section will display Delete, Duplicate, Export.

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 exported via this button.

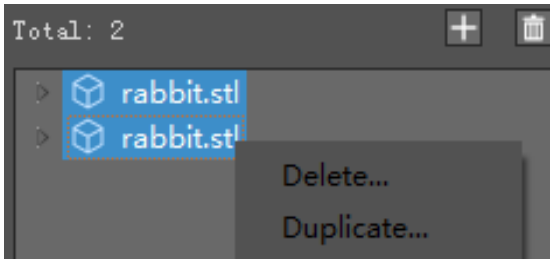


Figure 4.45: Multiple models can be exported at a time.

4.5 Uploading queue

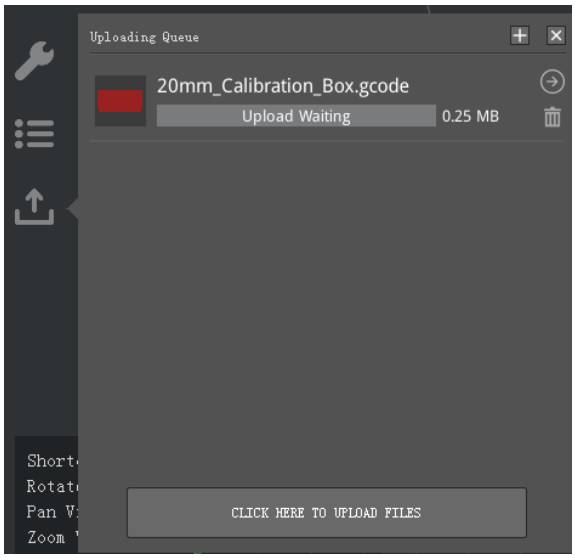


Figure 4.46: ideaMaker also can upload G-Code files sliced by other slicing software. Press "+" button to find the files you want to upload to your N-series printer from your computer.

This section shows the uploading list from ***ideaMaker*** to your printer.

Press " -> " to check more information of the uploading G-code file such as layer height, shell width and so on, and you also can connect to the printer or delete the uploading task as ***figure 4.47***.

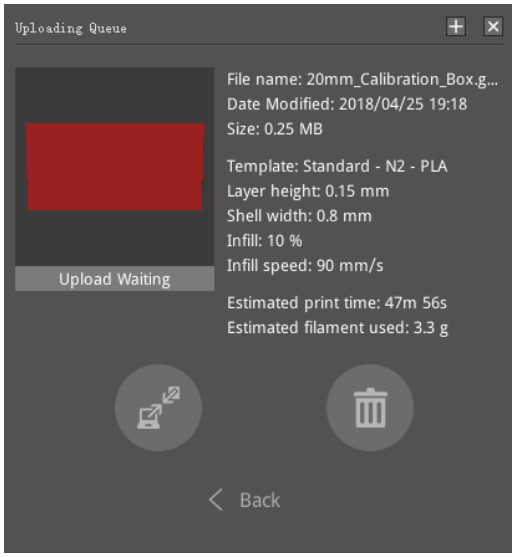


Figure 4.47: Detail information of the uploading task

4.6 Instant operation bar

: Add models.

: Start slicing directly.

: Export the gcode file to local disk or upload to printer.

4.7 Model preview

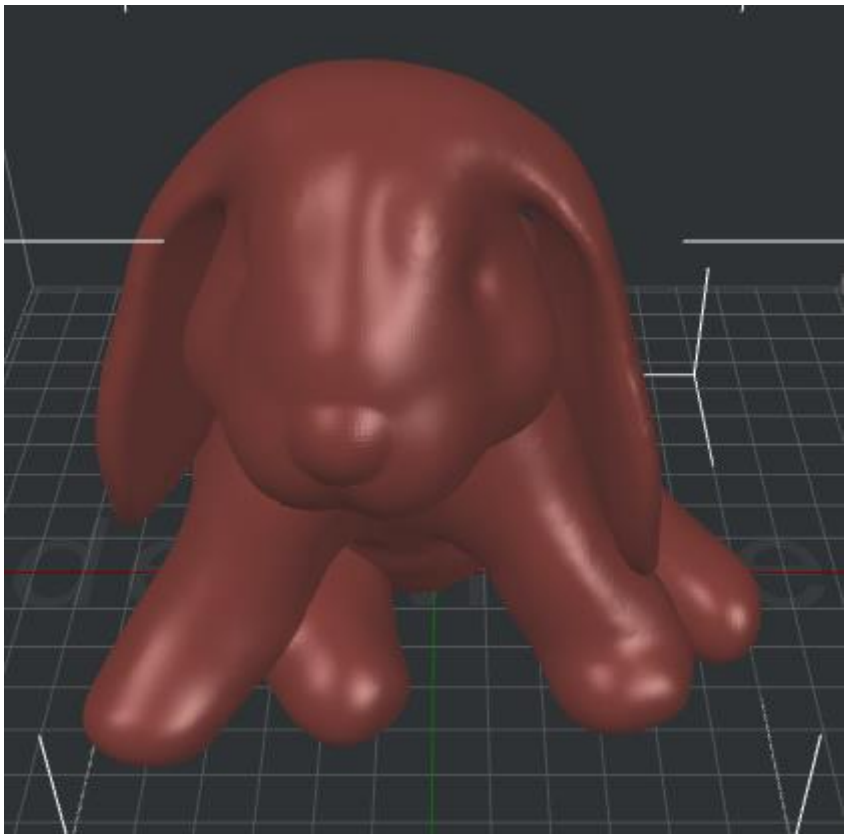


Figure 4.48: Preview your model.

This section is designed for previewing the current model.

4.8 Perspective transformation

Figure 4.49: 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 model with the default views.

4.9 Detecting Information

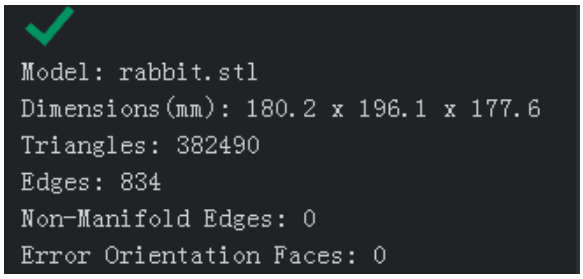


Figure 4.50: If here appears an error or a warning, try using the Auto-Repair function.

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. A ***Triangle*** is a polygon with three edges and three vertices.

Edges refers to the number of edges in the model.

Non-Manifold Edges refers to the number of non-manifold edges in the model. Generally, there are two types of non-manifold errors, one is an open object which means the model has holes or loose edges, another one is extra faces such as internal faces, overlapping faces and so on.

Error Orientation Faces refers to the number of error orientation faces in the model. In geometry the orientation of an object such as a line, plane or rigid body is part of the description of how it is placed in the space it is in. If your model shows in green which means you need to flip mesh orientation.

5 Slicing settings

5.1. Main Template

Main Template includes the printing settings inside and the default one can be selected to print your model directly.

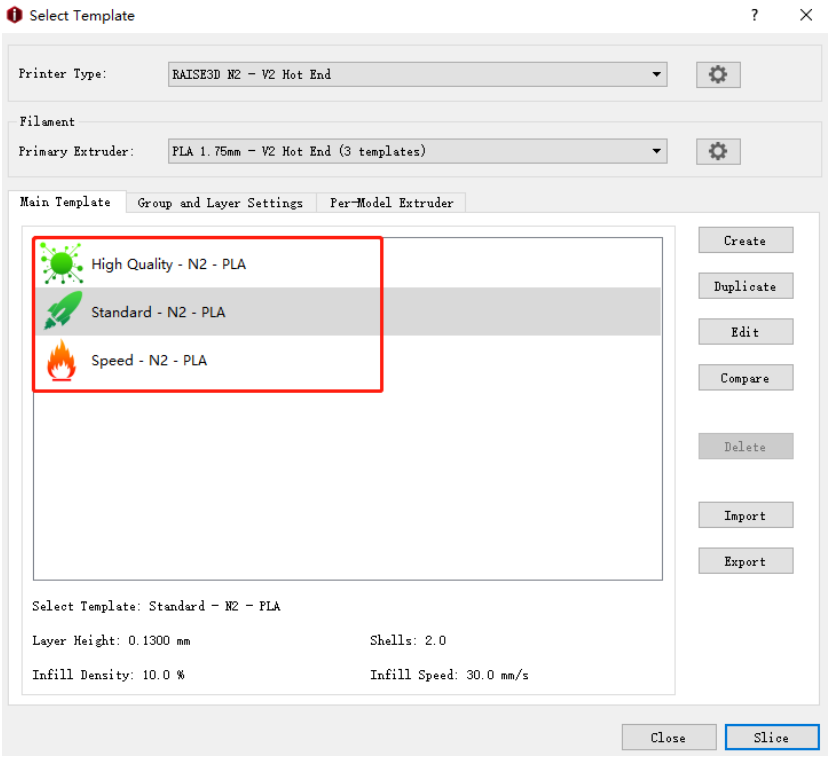


Figure 5.1: The Main Template tab.

5.1.1 Create Template

Create refers to create a new template and in this page you can edit your template's name, printer type, filament type and template.

For example: Create a new template as ***Figure 5.2*** and named it PLA then press ok button, and the Select Template tab will show interface as ***Figure 5.3***.

Template Name refers to the new template's name which you create, and you also can rename it by yourself.

Copy Template From refers to from which current template this template will copy.

Printer Type refers to the printer type of the copied template.

Filament Type refers to the filament type of the copied template.

Template refers to the name of the copied template.

Note: the initial Printer type, filament type and template are same as the template which you selected in ***Select Template*** interface.

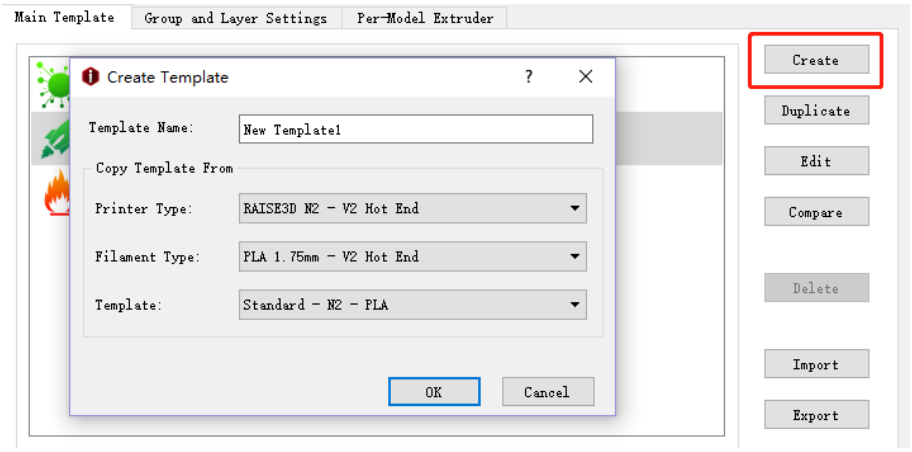


Figure 5.2: Create a new template.

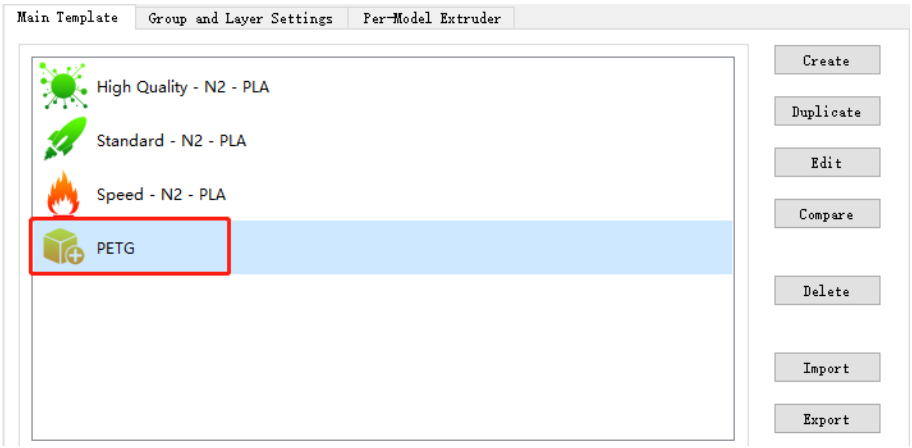


Figure 5.3: The interface of creating a new template.

5.1.2 Duplicate Template

Duplicate refers to creating a duplicated template from the selected slicing template.

Note: ideaMaker will name the duplicate template as ***New Template 1*** automatically, and you also can rename it by yourself.

For example: Duplicate a new template as ***Figure 5.4***.

Template Name refers to the name of your new template, and you also can rename it by yourself.

Copy Template From refers to from which current template this template will copy.

Printer Type refers to the printer type of the copied template.

Filament Type refers to the filament type of the copied template.

Template refers to the name of the copied template.

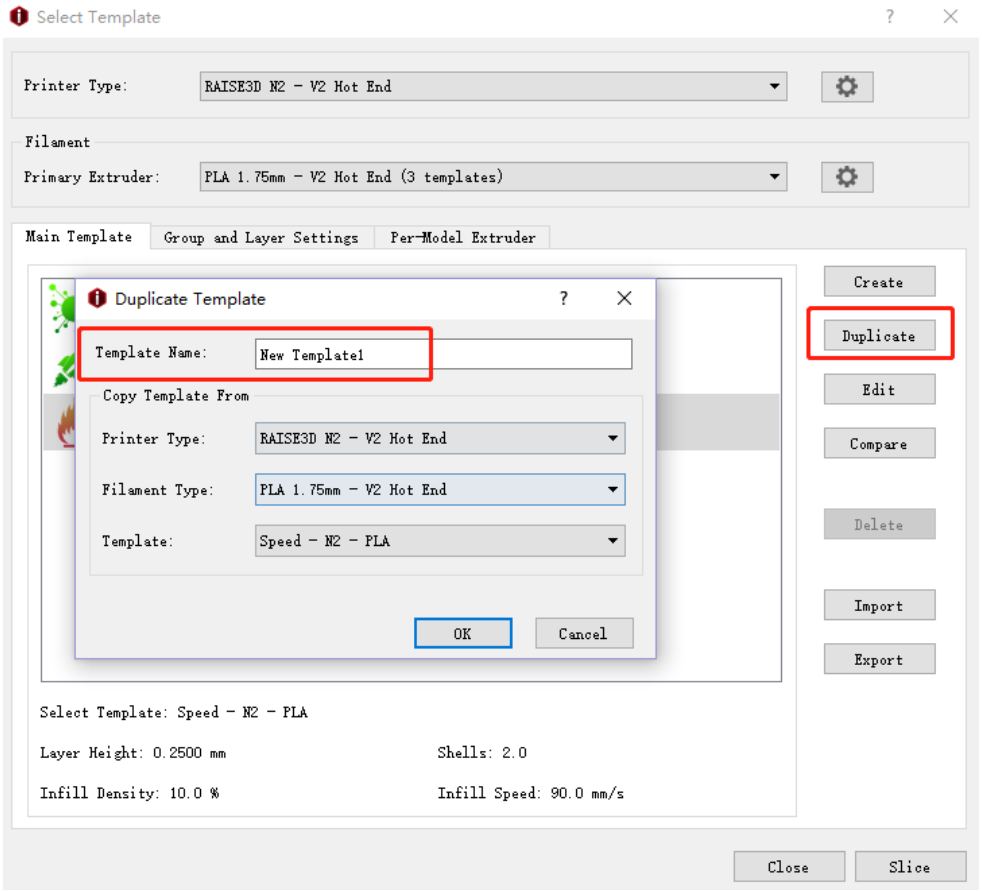


Figure 5.4: Duplicate a new template.

5.1.3 Edit Template

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

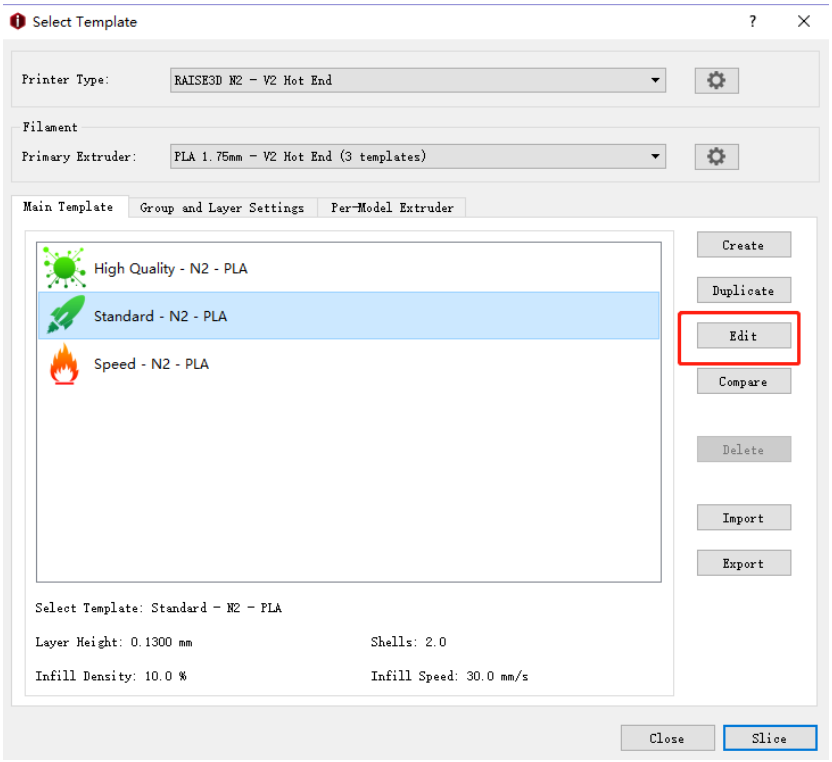


Figure 5.5: Select a template and press "Edit" to enter the setting page.

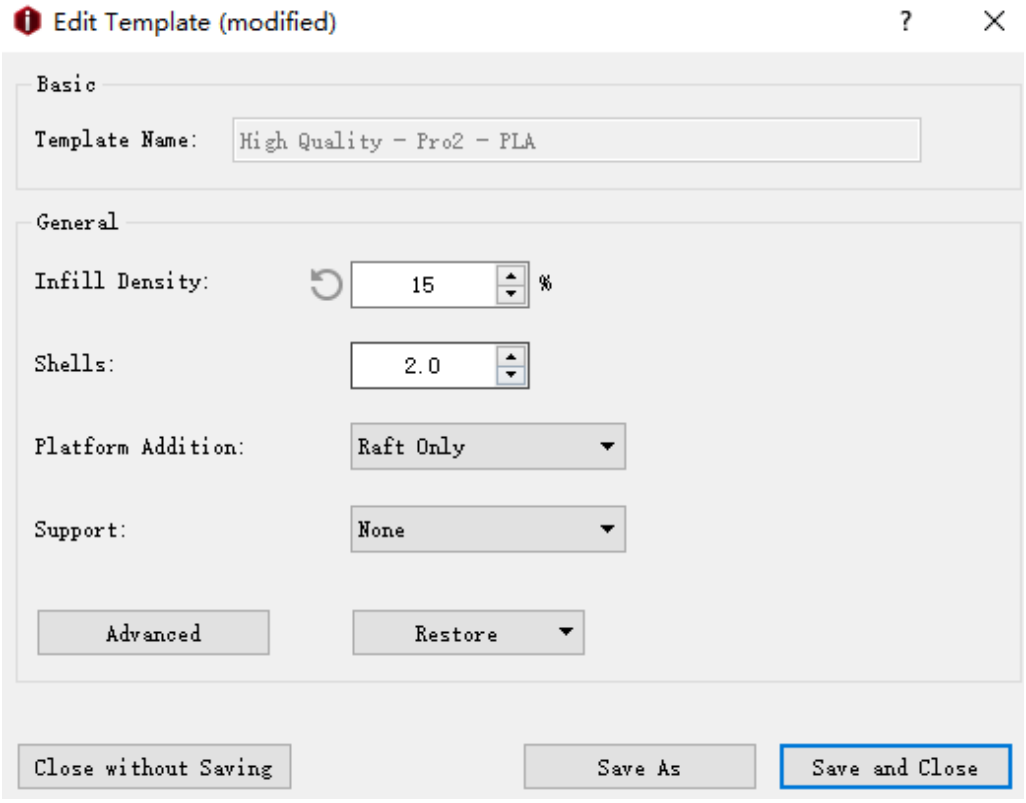


Figure 5.6: Common parameters.

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

: Press this button to back to default parameter, this function only works for default profile.

Shells refer to the thickness of model's wall.

Platform Addition refers to whether using ***Raft***/***Brim***/***Skirt*** or not.

The **Skirt** is loop(s) of plastic extruded around the first layer of the model to ensure a steady flow for model.

The ***Raft*** is composed by a few layers placed on the printing surface before the model is printed to help a model stick to the printing surface.

The ***Brim*** layer will only print a single layer placed on the first layer of the model which allow for better adhesion to the printing surface.

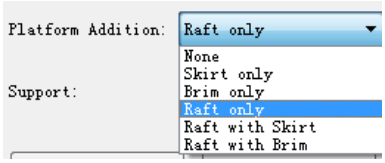


Figure 5.7: There are 6 types of "Platform Addition".

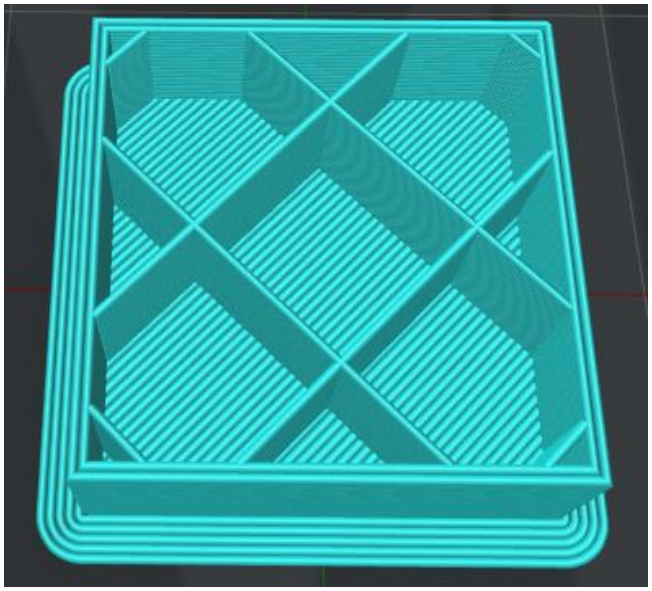


Figure 5.8: Printed with 2 shells, 10% fill density and Brim.

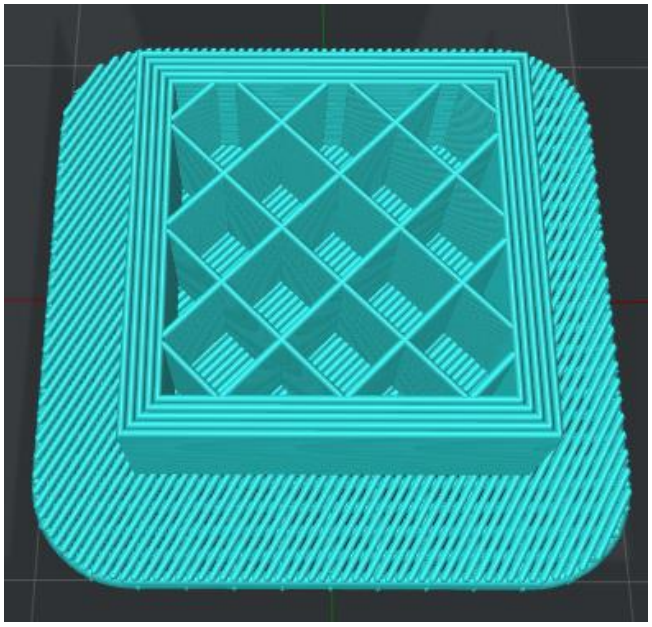


Figure 5.9: Printed with 5 shells, 20% fill density and Raft.

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.

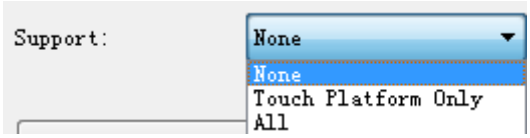


Figure 5.10: There are 3 types of supports.

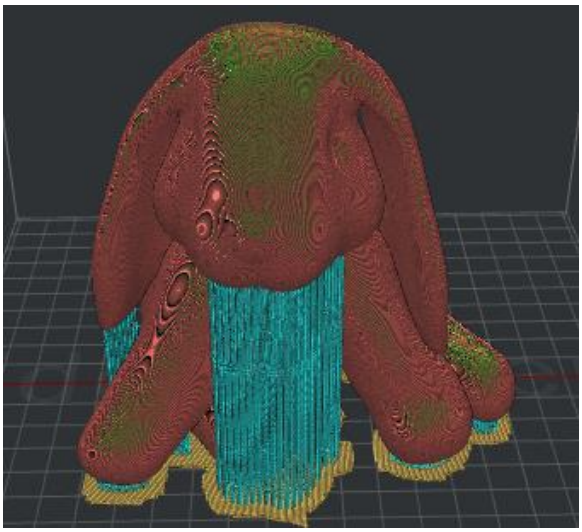


Figure 5.11: Printed with supports of "Touch Platform Only".

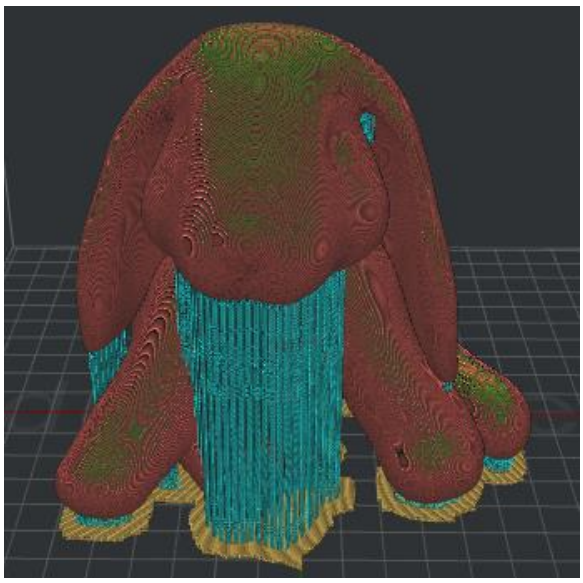


Figure 5.12: Printed with supports of "All".

Note: Advance Setting.

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

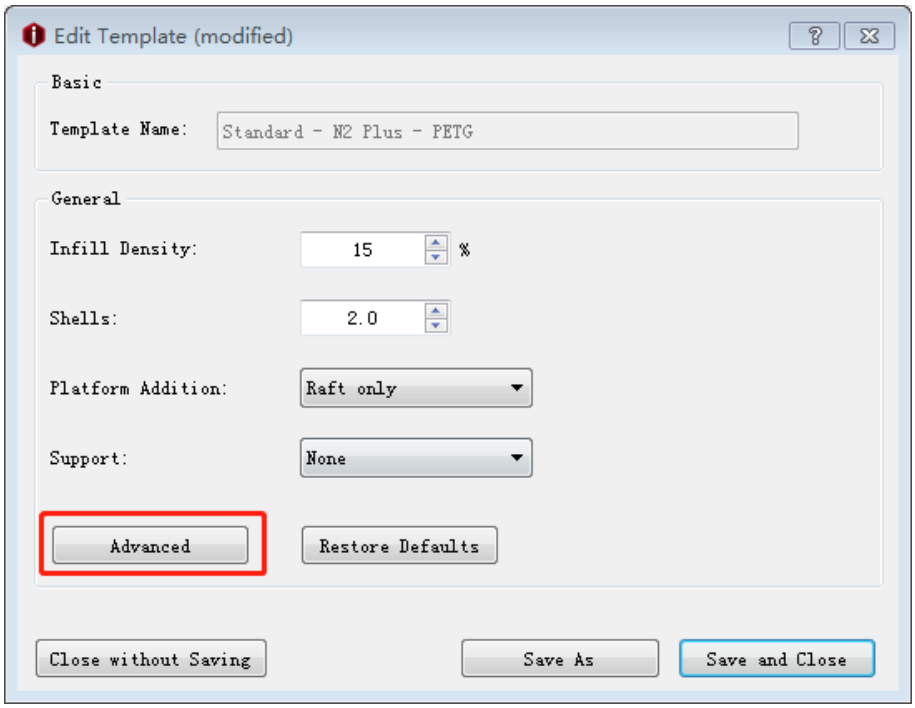


Figure 5.13: There are more settings in Advanced.

5.1.3.1 Layer

General:

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

Shells refer to the thickness of model's wall.

Maximum Shells overlap percentage refers to the max percentage of overlap between shells. When the overlap percentage of the lines from one single shell loop is larger than the set value, the shell will be replaced to be solid fill structure.

Print Shells in Optimal Order refers that with this function enabled the shells would be printed with optimizing order for reducing the travels paths and retractions.

Speed:

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

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

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

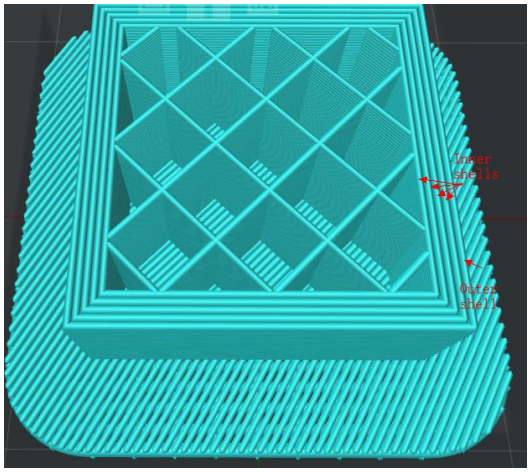


Figure 5.14: Only the shell located at outer edge is the outer shell. No matter how many shells you have set, except the outer one, the remaining shells are inner shells.

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

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

First Layer Settings:

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

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

First Layer Flowrate refers to setting the flow rate at the first layer. ***Flowrate*** refers to how much plastic the printer is told to extruder.

Layer Start Point refers to the position where the nozzle will start move from at each layer.
Layer Start Point Type has three options can be set as.
Nearest refers that the layer start point will be placed at an optimal position to save the time of movement.
Fixed refers that the layer start point will be placed as close as possible to the specified position.
Random refers that the layer start point will be placed randomly over the model.

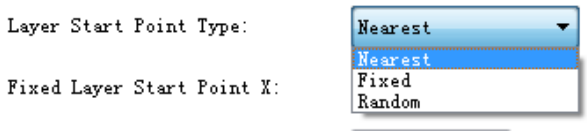


Figure 5.15: There are 3 types of Layer Start Point. If you select Fixed as your start point type, you can set X & Y's value to define a position. ideaMaker will set a closest position based on the position you set.

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

Seam Hiding
Place Seam on refers to placing the seam on the specified corner of the shells. There are four types in all: ***None, Reflex or Convex Corner, Reflex Corner*** and ***Convex Corner.***
Place Seam on the nearest corner as possible refers that with this function enable the slicer will take the nearest corner to place the seam on as possible.

Other:
Spiral Vase Mode means that the model will be printed out to be single outline corkscrew vase structure with no retraction. 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 by spiraling upwards.
Print External Shells First refers that with this function enabled software will print external shells first then print internal shells.

Print Infill First refers to with this function enabled the slicer will print infill first then print shells.

Print Parts in Same order for each layer refers that the models will be printed under a specific order in each layer. And the specific orders depend on models import order.

Note: this function only works for multiple models.

Minimal Segment Length refers that the minimal length of extruded line segment.

Merge Nearby Lines refers that the distance of two nearby lines less than the value you set, they would be merged.

Dimensional Compensation:

XY Size compensation for Contours refers to the compensation for the measured error of contours, especial for the expanded or shrunken object.

XY Size compensation for Holes refers to the compensation for the measured error of holes.

Note: Negative value will shrink the objects in each layer.

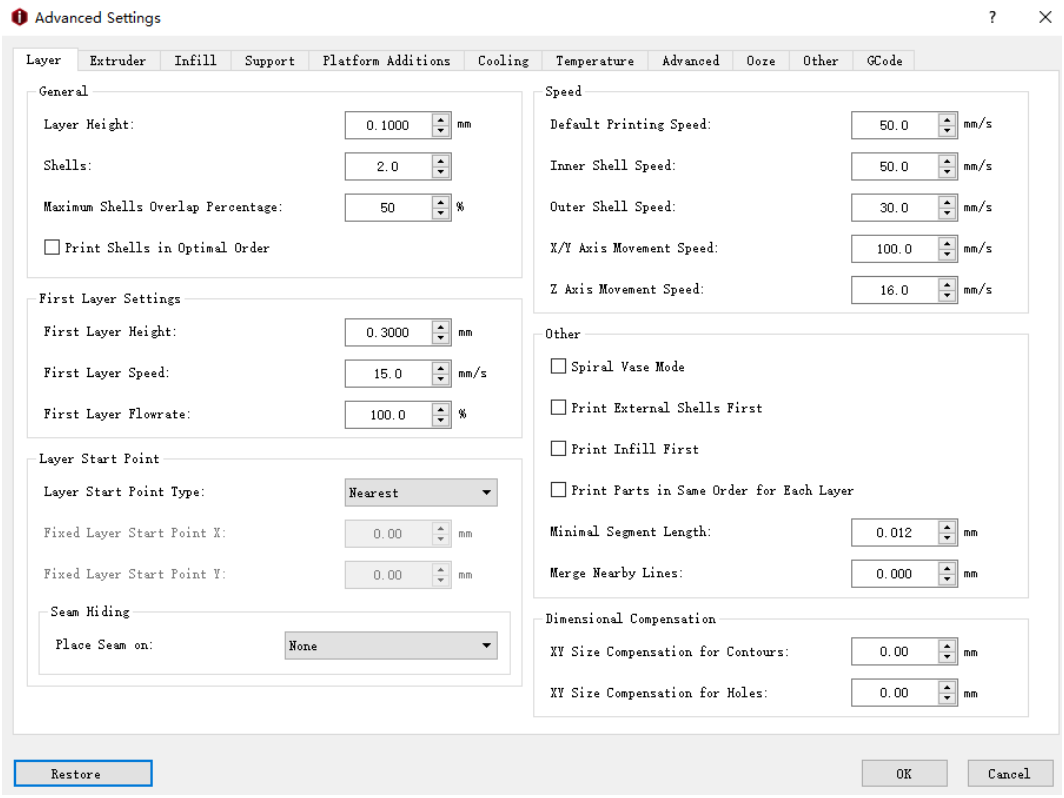


Figure 5.16: The Layer tab.

5.1.3.2 Extruder

Note: If you have set the ***Extruder Count*** to be 2, the Extruder tab will show ***Left Extruder*** and ***Right Extruder***. If you have set the ***Extruder Count*** to be 1, the Extruder tab will show ***Primary Extruder*** only. And we set the primary extruder as left extruder by default.

All the ***Settings*** are independent which means the settings in Left Extruder only work for left extruder and the settings in right extruder interface only work for right extruder such as ***Extrusion Width, Enable Retraction, Retraction speed, Retract Material Amount, Minimal Travel of Retraction, Minimal Amount of Retraction, Extra Restart Amount, Restart Speed*** and ***Z hop at Retraction.***

Left Extruder:

General:

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

Retraction:

Enable Retraction refers to enabling filament retraction to prevent stringing.

Retraction speed refers to the extruder speed for the retraction. A higher retraction speed works better. But it can lead to filament grinding when the speed is too high.

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

Minimal Travel of Retraction refers to the minimum distance of extruder motor reserves that defines if the filament needs to be retracted. Set this item to make sure you do not get a lot of retractions in a small area.

Minimal Amount of Retraction refers to the minimal amount of extrusion that defines if the filament needs to be retracted. If the amount of extrusion for printing is less than the minimal amount of retraction, the retraction will be ignored by system automatically. This avoids retracting a lot on the same piece of filament which flattens the filament and causes grinding issues.

Extra Restart Amount refers to the amount of extrusion compensation after the retraction.

Restart Speed refers to the extruder speed when the filament is extruded out after the retraction.

Z Hop at Retraction refers to the nozzle will lift from the surface of the model during retraction. Normally retraction is set before long-way movement which normally has faster moving speed than normal printing. So Z Hop can reduce the effects of nozzle scratching during fast move.

Z Hop Only at Solid Parts refers that with this function enabled Z Hop will be performed at top and bottom solid fill layers only.

Coasting Distance refers that with this function enabled the last part of the extrusion path will be replaced with a travel path only with no extrusion as ***Figure 5.17***. The distance of the extrusion path's last part is the **Coasting Distance**.

Enabling the function of ***Coasting Distance*** can decrease the filament leaking when nozzle travels from one point to another point.

Apply Coasting on Gird Infill and Lines Solid Fill refers to that with this function enabled the last part of the infill extrusion path will be replaced with a travel path only with no extrusion as ***Figure 5.19***.

Note: Please don't set the ***Coasting Distance*** too large. Or it may cause gaps between layers.

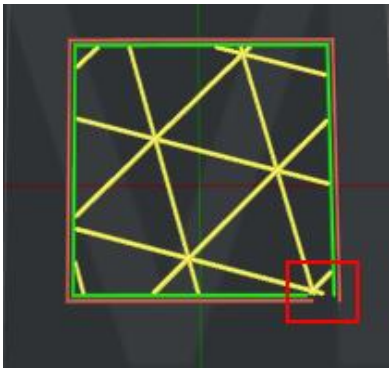


Figure 5.17: Set Coasting Distance to be 2mm.

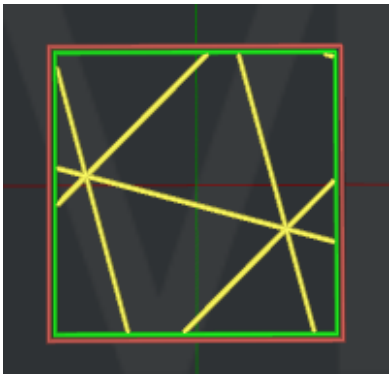


Figure 5.18: Set Coasting Distance to be 0.

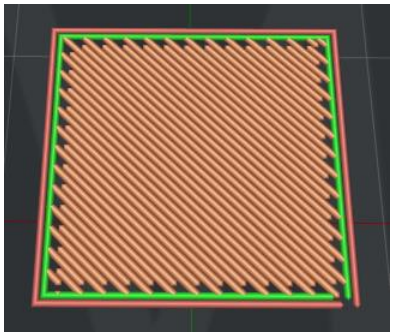


Figure 5.19: Enable the function of "Apply Coasting on Gird Pattern Infill and Lines Pattern Solid Fill".

Wipe:

Outer Shell Wipe Distance refers to the nozzle wipe distance at the end of the outer shell as ***Figure 5.20***.

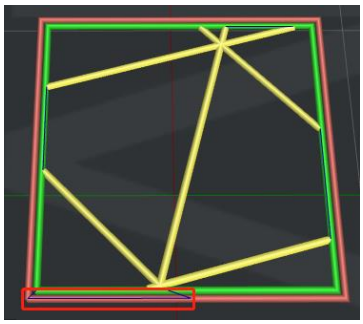


Figure 5.20: Set Outer Shell Wipe Distance to be 10mm.

Outer Shell Wipe Speed refers to the printer head movement speed at the end of the outer shell.

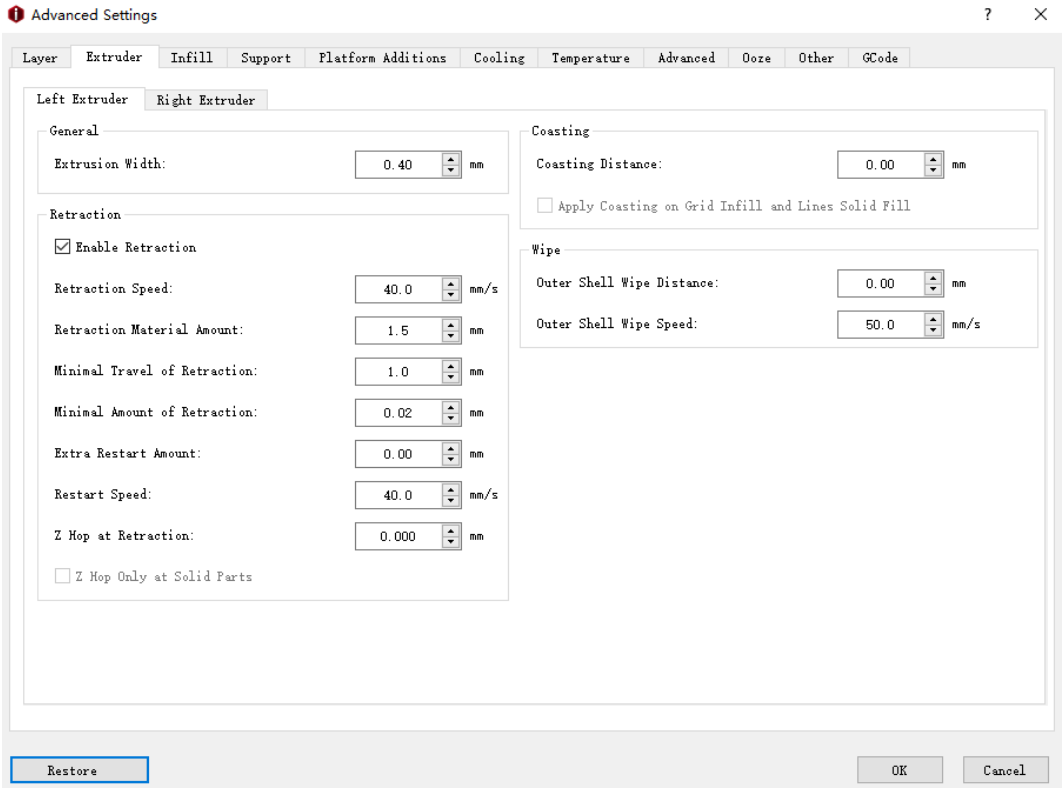


Figure 5.21: The Left Extruder tab.

Right Extruder:

Enable Right Extruder Configuration refers to enabling ***Right Extruder's Configuration*** function the below settings will be activated such as ***Retraction***. If not, the right extruder will continue to use the settings of left extruder.

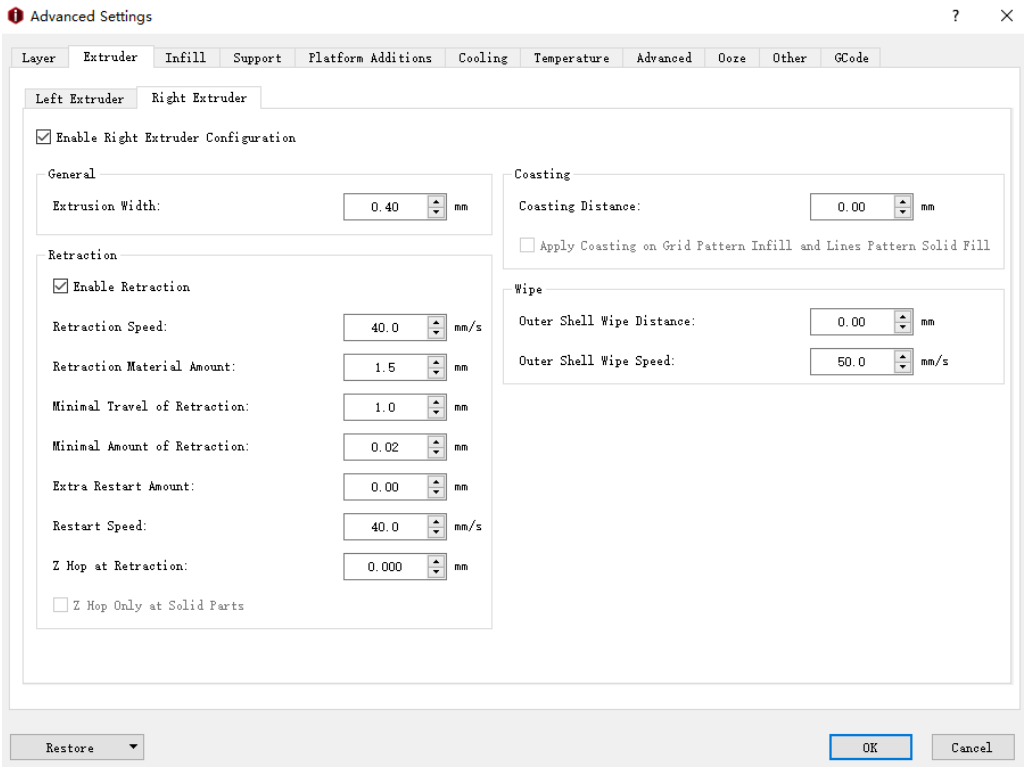


Figure 5.22: The Right Extruder tab.

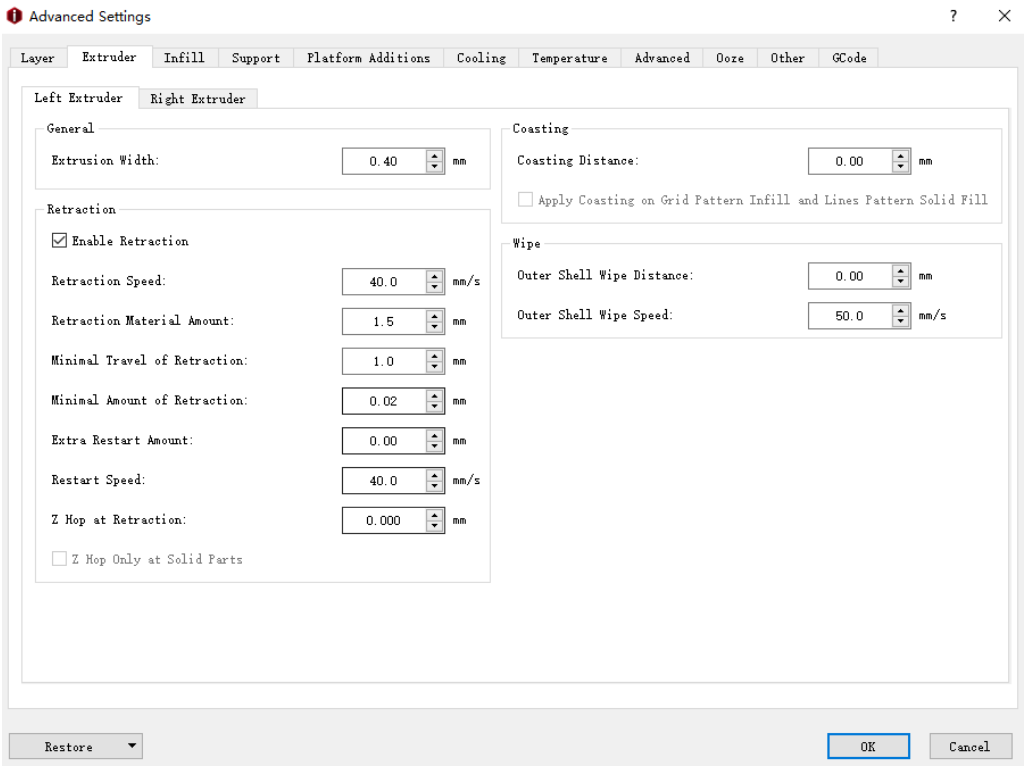


Figure 5.23: The Extruder tab.

5.1.3.3 Infill

Infill:

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

Figure 5.24: Select an "Infill Extruder" for your model.

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

Infill Speed refers to the printing speed for the model's infill structure.
Infill Overlap refers to the percentage of overlap between the infill and the shell.
Infill Flowrate refers to the flow rate of printing the infill structure. ***Flowrate*** refers to how much plastic the printer is told to extruder. 100% equals to default amount.
Infill Pattern Type refers to selecting the infill pattern for the interior of the model.

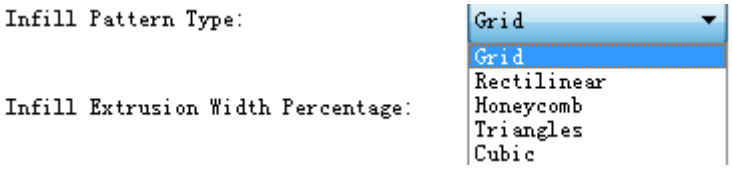


Figure 5.25: Select an "Infill Pattern Type" depending on the model type.

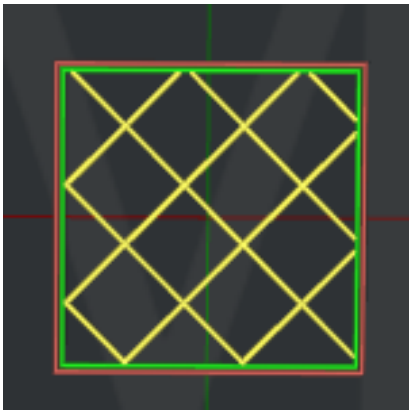


Figure 5.26: Set "Infill Pattern Type" to be "Grid".

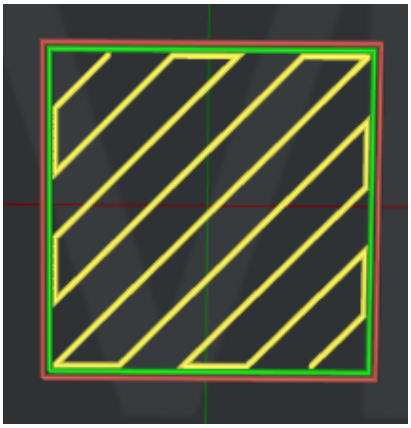


Figure 5.27: Set "Infill Pattern Type" to be "Rectilinear".

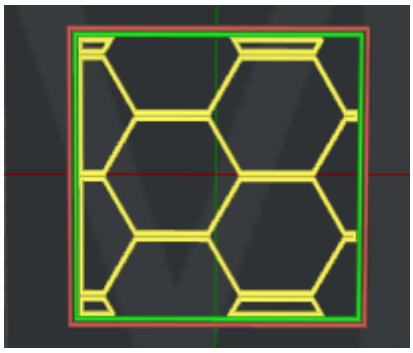


Figure 5.28: Set "Infill Pattern Type" to be "Honeycomb".

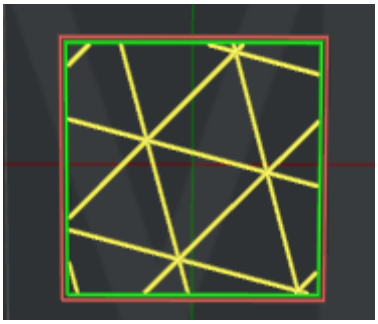


Figure 5.29: Set "Infill Pattern Type" to be "Triangles".

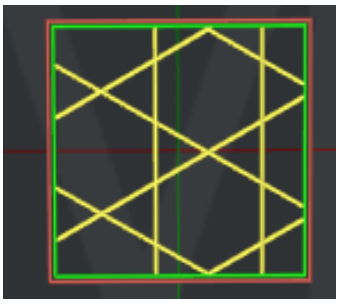


Figure 5.30: Set "Infill Pattern Type" to be "Cubic".

Infill Extrusion Width Percentage refers to the percentage of infill extrusion width. For example: If your extrusion width is 0.4mm, and you set this value to 120% then the infill extrusion width is 0.48mm.
Infill Offset X refers to with this function enabled the slicer will apply X offset to infill structure.
Infill Offset Y refers to with this function enabled the slicer will apply Y offset to infill structure.
Combine Infill Layer refers that multiple layers of infill structure will be combined and printed with thicker layer thickness to reduce print time. For example: If you set Combine Infill layer to be 5, the first four layers would no print infill, then the fifth layer will print a thicker layer thickness infill structure.

Infill Angle refers that you can define the direction of each layer of the infill structure. If you add 30, 60, 90deg, the angle of the first layer of infill structure will be 30deg.The second layer will be 60deg. The third layer will be 90deg. The forth layer will be back to 30deg and the angle of the following layers changes in the order.
Angle refers to the directional angle of infill structure.
Add Infill Angle refers to adding a value.
Remove Infill Angle refers to removing the selected value.

Top and Down Solid Part:
Bottom Solid Fill Layers refers to the number of solid layers at the bottom of the model.
Top Solid Fill Layers refers to the number of solid layers at the top of the model.
Bottom Solid Fill Speed refers to the printing speed for solid bottom layers.
Top Solid Fill Speed refers to the printing speed for solid top layers.

Bottom Solid Fill Flowrate refers to the flow rate of printing the interior of the solid part at the bottom of the model.

Top Solid Fill Flowrate refers to the flow rate of printing the interior of the solid part at the top of the model.

Top Solid Fill Pattern Type refers to the infill pattern for the solid layers at the top of the model.

Bottom Solid Fill Pattern Type refers to the infill pattern for the solid layers at the bottom of the model.

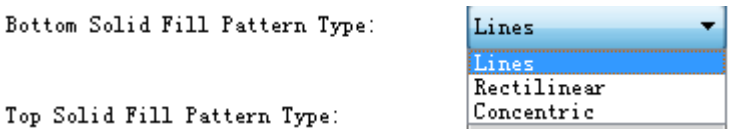


Figure 5.31: Select a Solid Fill Pattern Type depending on the model type.

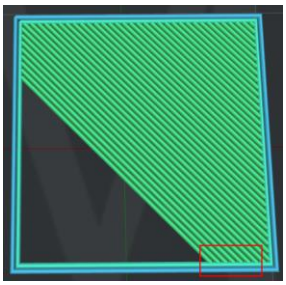


Figure 5.32: Set "Top Solid Fill Patter Type" to be "Lines".

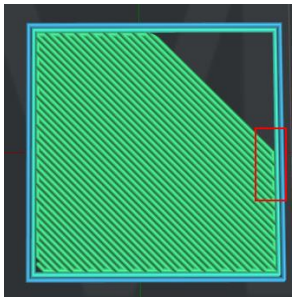


Figure 5.33: Set "Top Solid Fill Pattern Type" to be "Rectilinear".

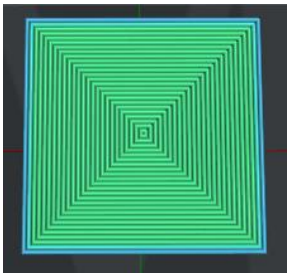


Figure 5.34: Set "Top Solid Fill Pattern Type" to be "Concentric".

Use Gap Filling for Failing Solid Part refers that with this function enabled the failed solid part will be filled with the selected solid fill pattern type.

Top and down Solid Fill Angle refers that you can define the direction of each layer of the top and down solid fill structure. If you add 30, 60, 90deg, the angle of the first layer of top and down solid fill structure will be 30deg.The second layer will be 60deg. The third layer will be 90deg. The forth layer will be back to 30deg and the angle of the following layers changes in the order.

Angle refers to the directional angle of top and down solid fill structure.

Add Infill Angle refers to adding a value.

Remove Infill Angle refers to removing the selected value.

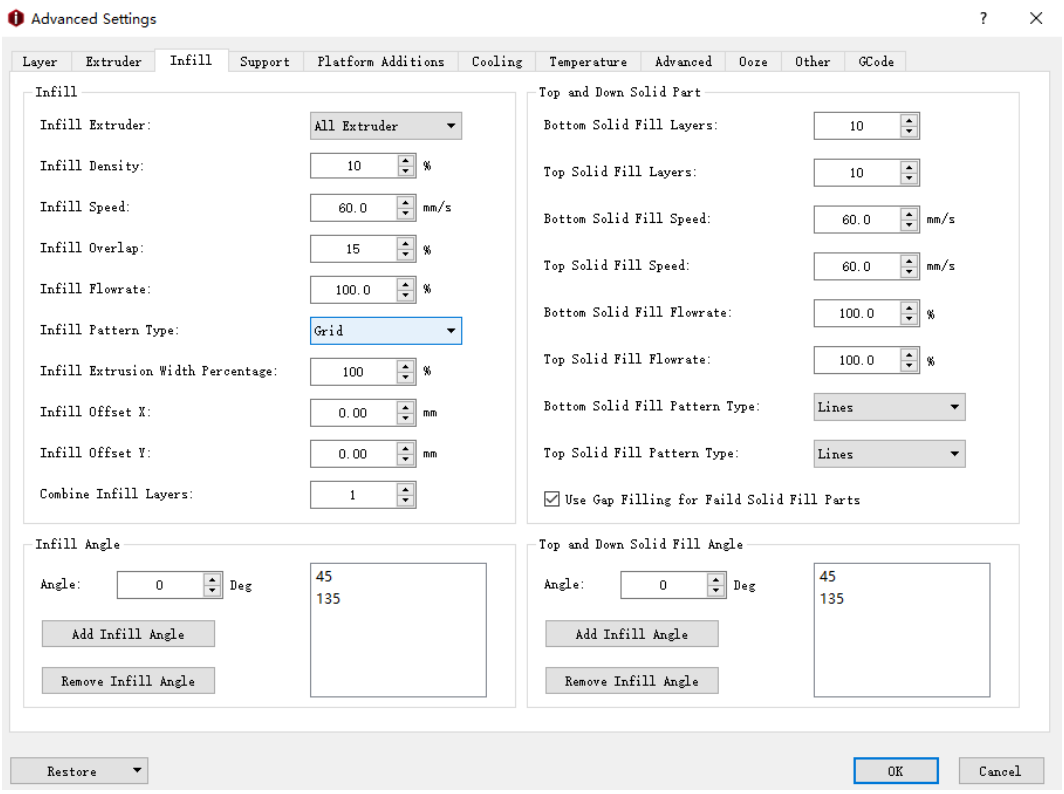


Figure 5.35: The Infill tab.

5.1.3.4 Support

Generate Support refers to selecting which structure for overhang part.

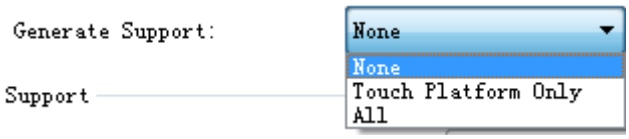


Figure 5.36: Selecting support structure for overhang part.

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.

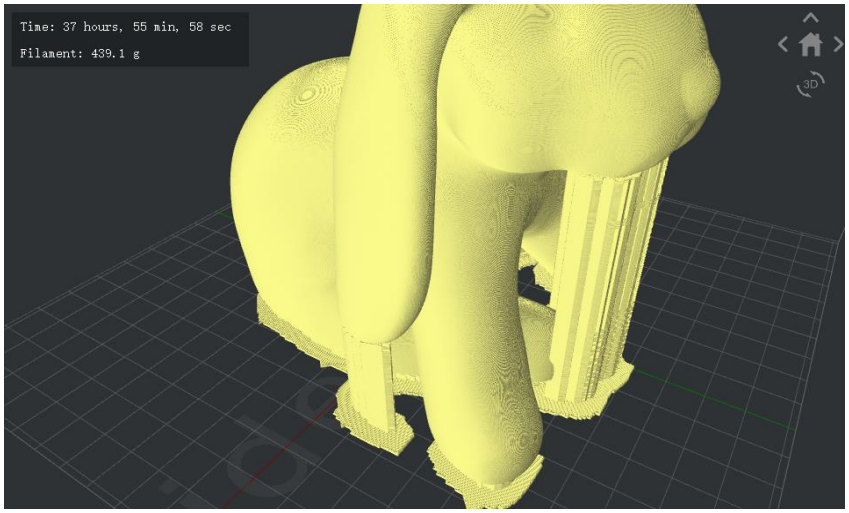


Figure 5.37: Printed with supports of "Touch Platform Only".

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

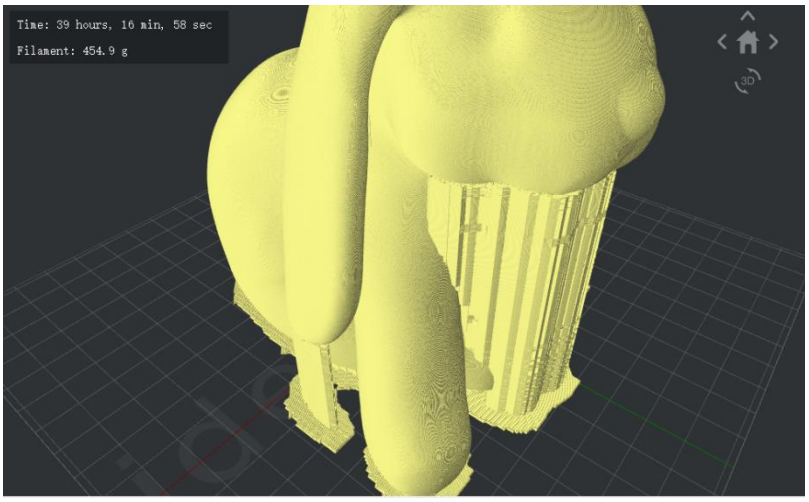


Figure 5.38: Printed with supports of "All".

Support:

Support Extruder refers to selecting which extruder to print support. (If you have set number of extruders to be 1, then it will only show ***Primary Extruder***; if you have set it to be 2, then it will show then it will show two functions: ***Left Extruder and Right Extruder***.).

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 entirety based on and computed based only pillars.

Figure 5.39: Two types of support structure.

Support Infill Type refers to the infill pattern for the support structure.

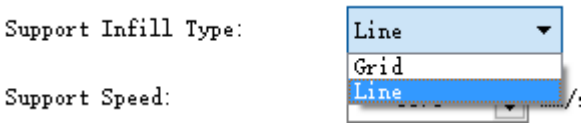


Figure 5.40: Two types of Support Infill pattern.

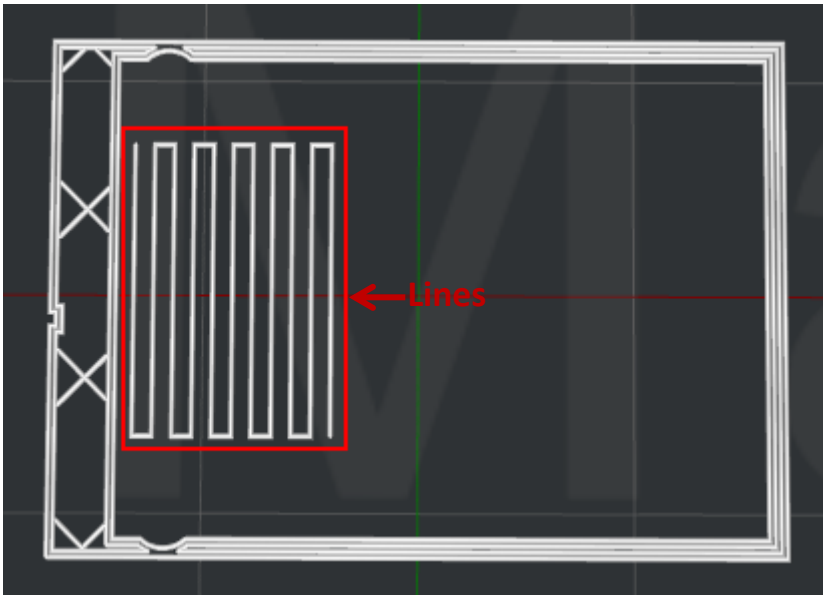


Figure 5.41: Set "Support Infill Type" to be "Lines".

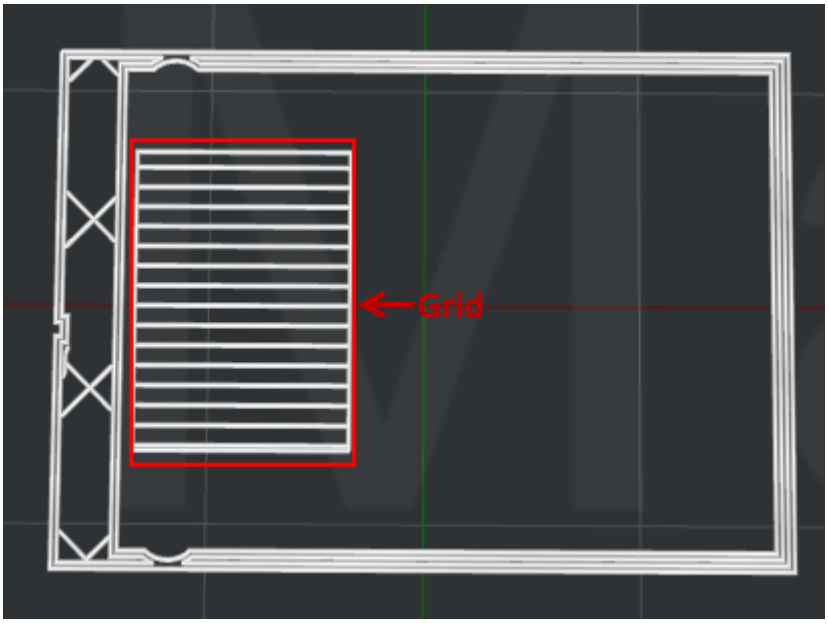


Figure 5.42: Set "Support Infill Type" to be "Grid".

Support Speed refers to the printing speed for the support structure.

Infill Ratio refers to the density of support structure.

Max Overhang Angle defines which parts of the model will be added support onto. When the real overhang angle on model is greater than this set value, support structure will be generated. Overhang Angle refers to the angle between the overhang surface and Z axis as ***Figure 5.43***. (Set 0 means all overhang parts of the model will be added with support. Set 90 means no support will be added to the model).

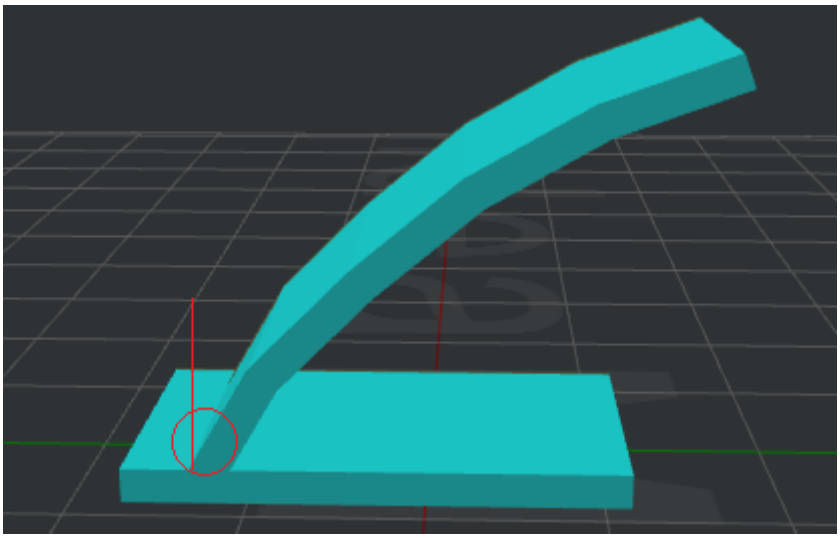


Figure 5.43: Overhang angle.

Horizontal Offset refers to the distance between the support structure and the model parts in X/Y direction.

Vertical offset Top Layers refers to the offset between the top of support structure and the model parts in Z direction.

Vertical offset Down Layers refers to the offset between the bottom of support structure and the model parts in Z direction.

Support Flowrate refers to the flow rate of printing the support structure.

Horizontal Expansion refers that with this function enabled the support would be larger in the Horizontal direction and it can be removed easier.

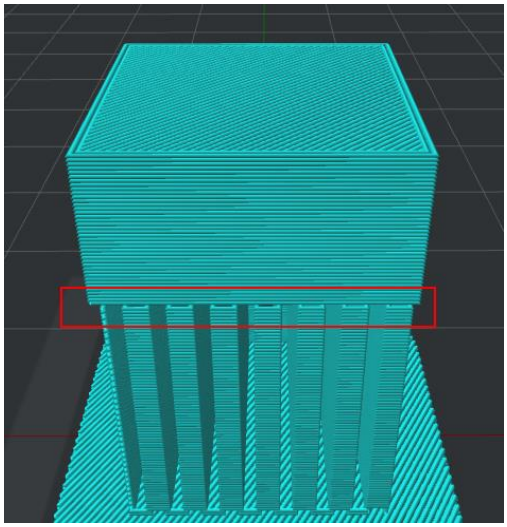


Figure 5.44: Set "Horizontal Expansion" to be 0.

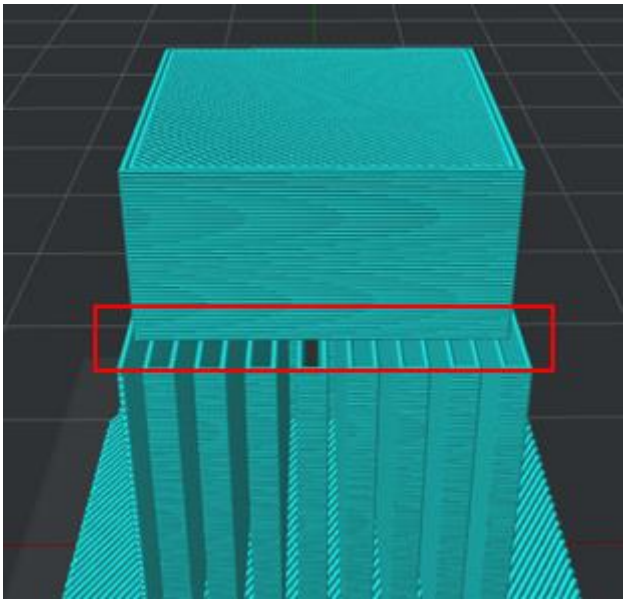


Figure 5.45: Set "Horizontal Expansion" to be 2.

Solid Base Layer refers to increasing adhesion of support structure of below layers as ***Figure 5.46.***

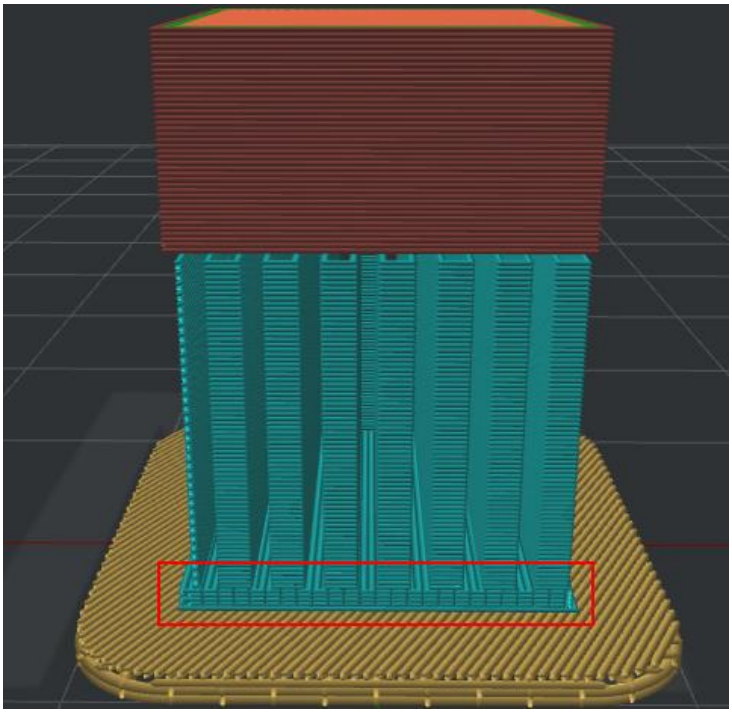


Figure 5.46: Set "Support Base Layer" to be 5.

Dense Support:

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

Dense Support Infill Ratio refers to the infill density of the dense support layers.

Dense Support Infill Type refers to selecting the infill pattern for the dense support layers. ***Grid*** type is more solid to be the basement. ***Line*** type is easier to peel.

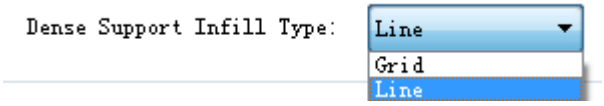


Figure 5.47: Two types of dense support infill pattern.

Dense Support Extruder selecting which extruder to print dense 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***).

Figure 5.48: Selecting an extruder for dense support.

Figure 5.49: Set "Dense Support Infill Type" to be "Lines".

Figure 5.50: Set "Dense Support Infill Type" to be "Grid".

Interlaced Support Infill Angle: You can define the direction of each layer of the infill structure which can decrease the potential of hangs in corners due to all support at the same direction. If you add 30, 60, 90deg, the angle of the first layer of support will be 30deg.The second layer will be 60deg. The third layer will be 90deg. The forth layer will be back to 30deg and the angle of the following layers changes in the order.

Angle refers to the directional angle of infill structure.

Add Infill Angle refers to adding a value.

Remove Infill Angle refers to removing the selected value.

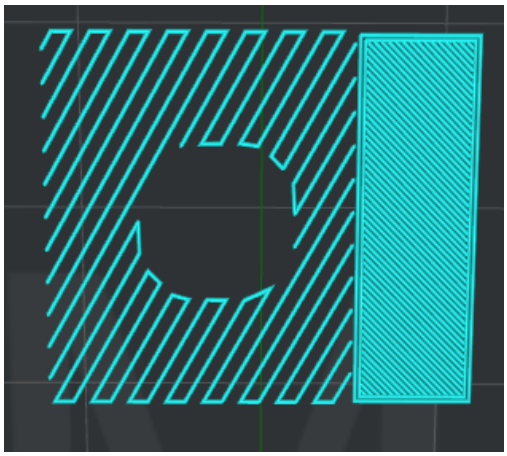


Figure 5.51: Set "Interlaced Support Infill Angle" to be 30 degrees.

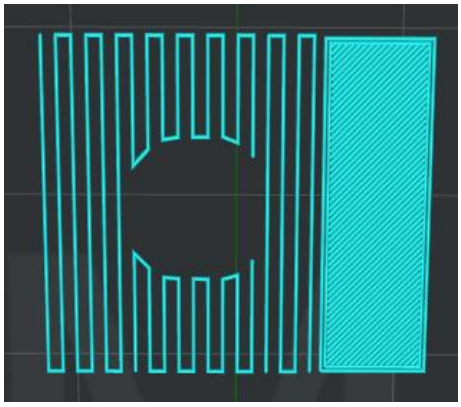


Figure 5.52: Set "Interlaced Support Infill Angle" to be 0 degree.

Other:

Pillar Size refers to the size of each support pillar. This value only works with Pillar type support.

Sparse Connection refers that disable retraction between support structures, which will cause some oozing strings, look like sparse connection.

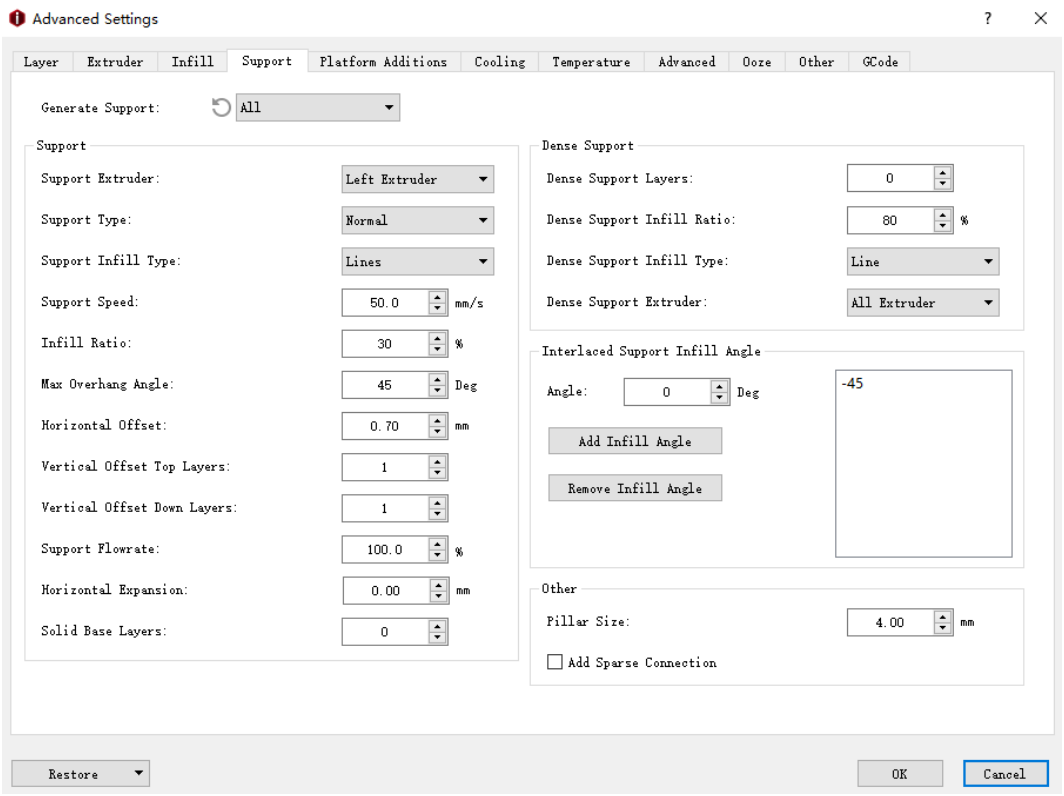


Figure 5.53: The Support tab.

5.1.3.5 Platform Additions

Platform Addition refers to whether the ***Raft***/***Brim***/***Skirt*** needs to be added or not, there are six options in all.

The ***None*** refers to not printing ***Raft/Brim/Skirt*** for model.

The **Skirt** is loop(s) of plastic extruded around the first layer of the model to ensure a steady flow for model.

The ***Raft*** is composed by a few layers placed on the printing surface before the model is printed to help a model stick to the printing surface.

The ***Brim*** layer will only print a single layer placed on the first layer of the model which allow for better adhesion to the printing surface.

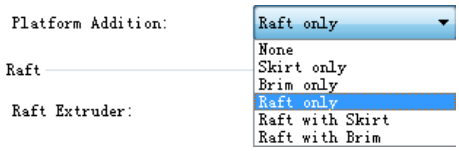


Figure 5.54: Select a structure for "Platform Addition".

Raft:

Note: It will only exist when the platform addition is set as ***Raft only*** or ***Raft and Skirt*** or ***Raft and Brim***.

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

Raft Offset refers to the distance between the model and extra raft area around it.

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

Raft Lines Type refers to infill pattern for the raft.

Lines refer to the segments at the same direction.

Rectilinear refers to the fill structure printed with a continuous moving path.

Figure 5.55: Two types of Raft Lines.

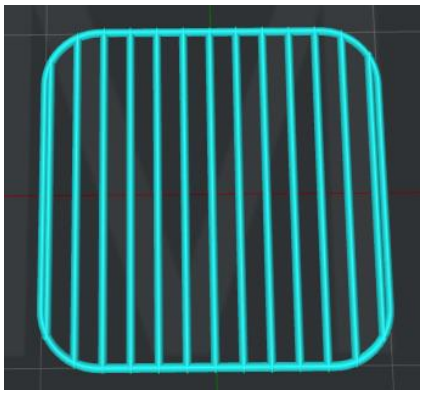


Figure 5.56: Set "Raft Lines Type" to be "Lines".

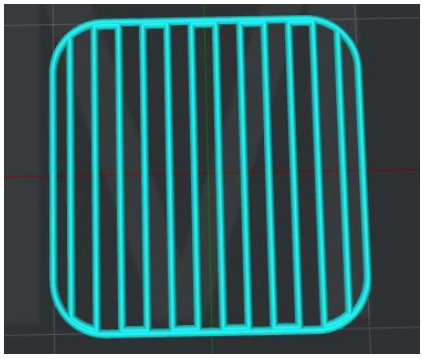


Figure 5.57: Set "Raft Lines Type" to be "Rectilinear".

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

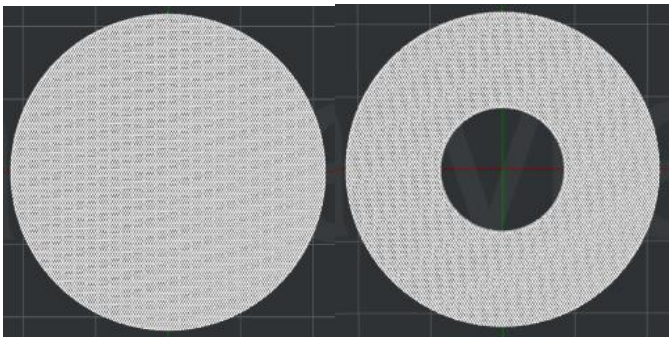


Figure 5.58: Normal Raft. ***5.59: Keep Holes in raft structure.***

First Layer refers to the number of layers as the base layers of the raft.

First Layer Speed refers to the printing speed for the first layers of the raft.

Extrusion Width Percentage refers to the percentage of extrusion width in raft first layers. For Example, if the extrusion width is 0.4 mm, and you set this value to be 120% then the raft first layer extrusion width will be 0.48mm.

Layer Height refers to the thickness of raft first layer.

First Layer Infill Ratio refers to the infill density for the first layers of the raft.

First Layer Infill Angle refers to the infill angle for the first layers of the raft.

Middle Layer refers to the number of middle layers at the middle of the raft.

Middle Layer Speed refers to the printing speed for the middle layers of the raft.

Extrusion Width Percentage refers to the percentage of extrusion width in raft surface layers. For Example, if the extrusion width is 0.4 mm and you set this value to be 120% then the raft middle layer extrusion width will be 0.48mm.

Layer Height refers to the thickness of raft middle layer.

Middle Layer Infill Ratio refers to the infill density for the middle layers of the raft.

Middle Layer Infill Angle refers to the infill angle for the middle layers of the raft.

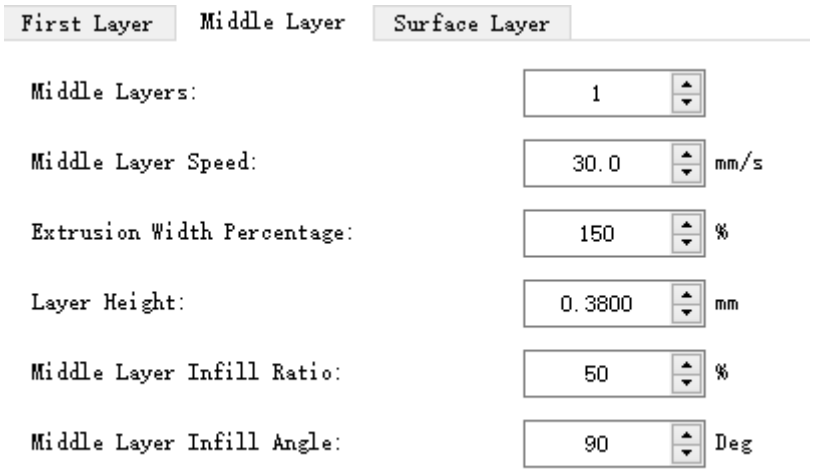


Figure 5.60: Middle Layer.

Surface Layers refers to the number of surface layers at the top of the raft.

Surface Layer Speed refers to the printing speed for the surface layers of the raft.

Extrusion Width Percentage refers to the percentage of extrusion width in raft surface layers. For Example, if the extrusion width is 0.4 mm and you set this value to be 120% then the raft surface layer extrusion width will be 0.48mm.

Layer Height refers to the thickness of raft surface layer.

Surface Layer Infill Ratio refers to the infill density for the surface layers of the raft.

Surface Layer Infill Angle refers to the infill angle for the surface layers of the raft.

Surface Layer Flowrate refers to the setting the Flowrate at the raft surface layer. *Flowrate* refers to how much plastic the printer is told to extruder.

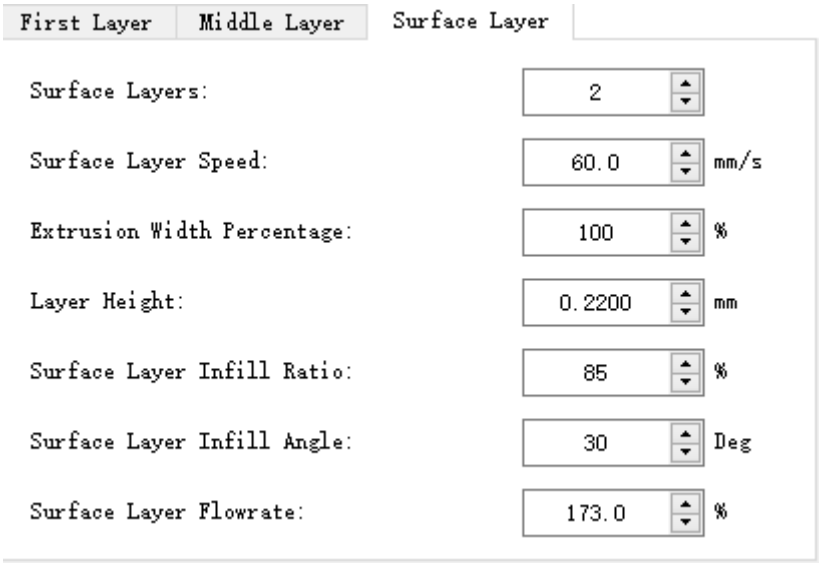


Figure 5.61: Surface Layer.

Skirt:

Note: It will only exist when the platform addition is set to be ***Skirt only*** or ***Raft and Brim***.

Skirt /Brim Extruder refers to selecting an extruder for skirt or brim part. (If you have set number of extruders to be 1, then it will only show ***Primary Extruder***; if you have set it to be 2, then it will show ***Left Extruder*** *and you can switch to* ***Right Extruder***).

Skirt Loop Lines refers to the number of skirt circle.

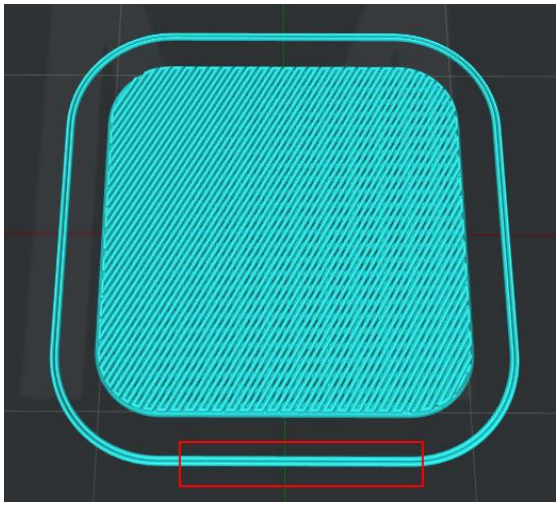


Figure 5.62: Set "Skirt Loop Lines" to be 2.

Skirt Offset Distance refers to the distance between skirt and model.

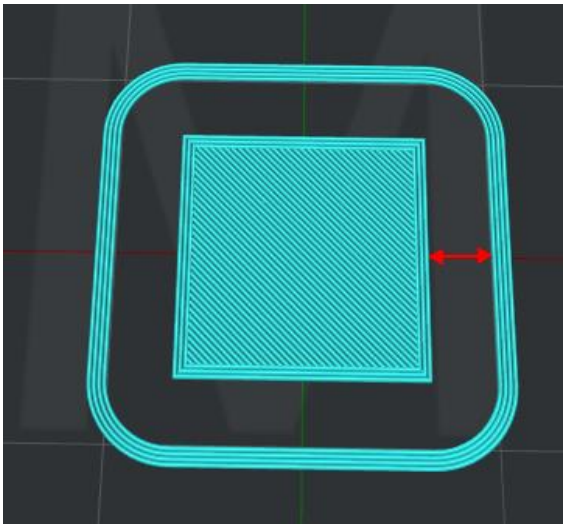


Figure 5.63: Set "Skirt Offset Distance" to be 5mm.

Brim:

Note: It will only exist when the platform addition is set to be ***Brim only*** or ***Raft and Brim***.

Brim Loop Lines refers to the number of brim circle.

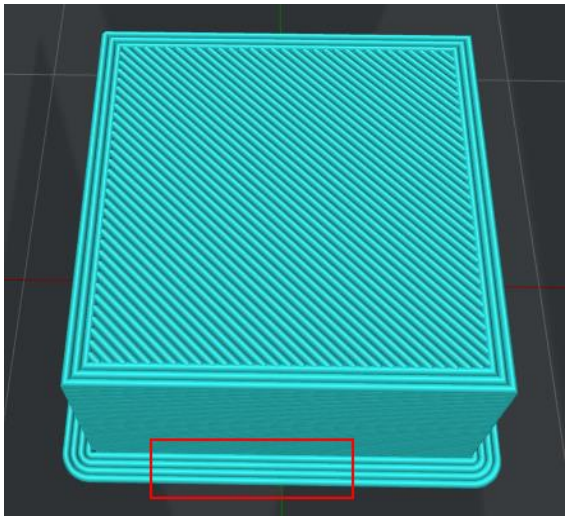


Figure 5.64: Set "Brim Loop Lines" to be 4.

Print Outer Shell Before Inner Shell in the First Layer refers to that with this function enabled software will print outer shell first if you set ***Platform Additions*** to be ***Brim only***.

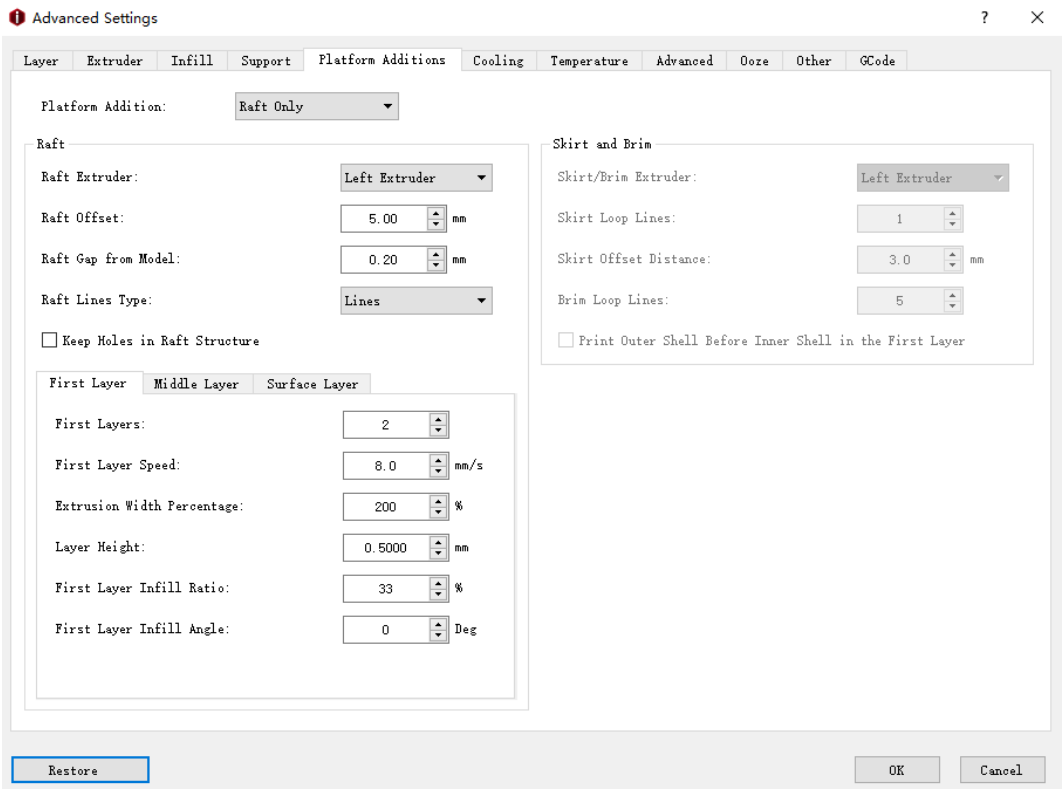


Figure 5.65: The Platform Additional tab.

5.1.3.6 Cooling

Cooling:

Minimal Layer Print Time refers to the minimum time spending in a layer to provide enough cooling time for the just printed out features in this layer before the next layer starts.

Slow Down Printing Speed refers that with this function enabled printing speed would be reduced to the value between current printing speed and the minimal printing speed. Note: This function only works for when single layer used time is less than minimal layer print time.
Minimal Printing Speed refers that we need to reduce print speed to increase print time when layer print time less than ***Minimal Layer Print Time***.

Increase Fan Speed refers that with this function enabled fan speed would be increased between the max and current fan speed. Note: This function only works for when single layer used time is less than minimal layer print time.
Maximum Fan Speed refers to the maximum speed of the extra cooling fan. If the cooling setting slows down the printing speed, the fan could be adjusted between the max and current fan speed.

Fan Control:

Fan Control: only the downward model fan on single extruder printer of N series and the two side fans of Pro2 series are controllable. The side extruder cooling fan is constant on. We don't suggest you switch the extruder cooling fan to the controllable slot which may cause filament in throat tube be melted before getting into the hot zone due to improper cooling for cold end.

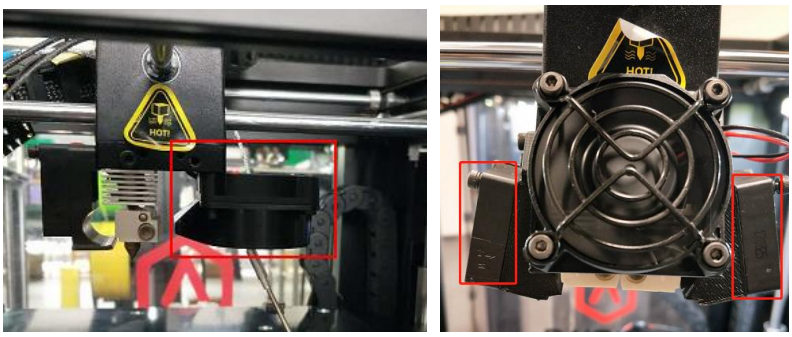


Figure 5.66: Controllable cooling fan.

Layer refers to selecting at which layer to change the fan speed.

Fan Speed refers control the speed of the downward model fan on single extruder printer. 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 selecting the value you want to remove in right box and clicking the ***Remove Fan Point*** button.

Note: If you want to change the fan's speed, enable the function of ***Use PWM-Controllable fans*** which is under the ***Printer Settings*** as ***Figure 5.67***. If not, this tab will be disabled and cannot be edited as ***Figure 5.68***.

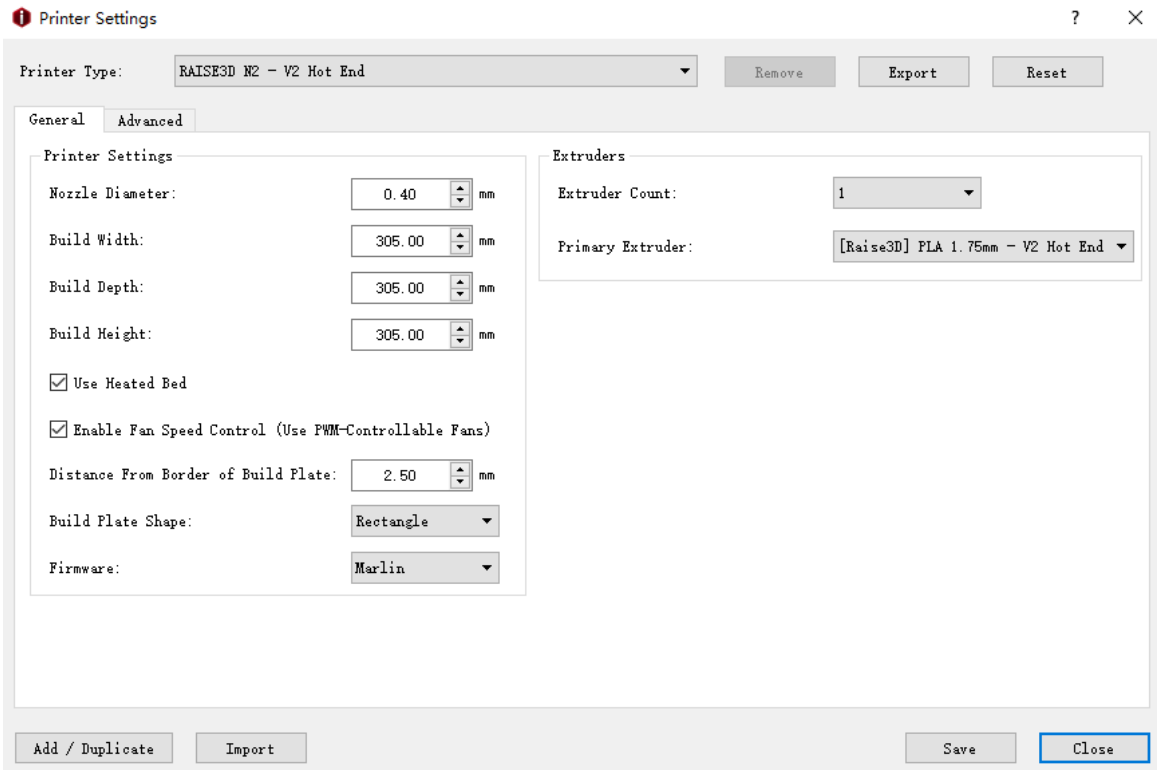


Figure 5.67: Use PWM-Controllable fans

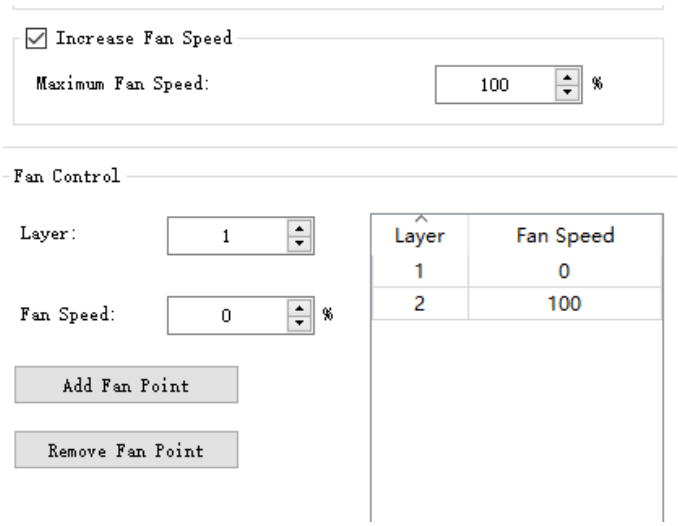


Figure 5.68: Fan Speed.

Blip fan speed to 100% when starting from low speed

Low Fan Speed Threshold refers to that fan speed will be increased to be 100% automatically if current fan speed is lower than threshold.

Blip Fan Speed Pause Duration refers to that pause the printer for a period in milliseconds after fan speed blipped to 100% and before increased to a higher speed.

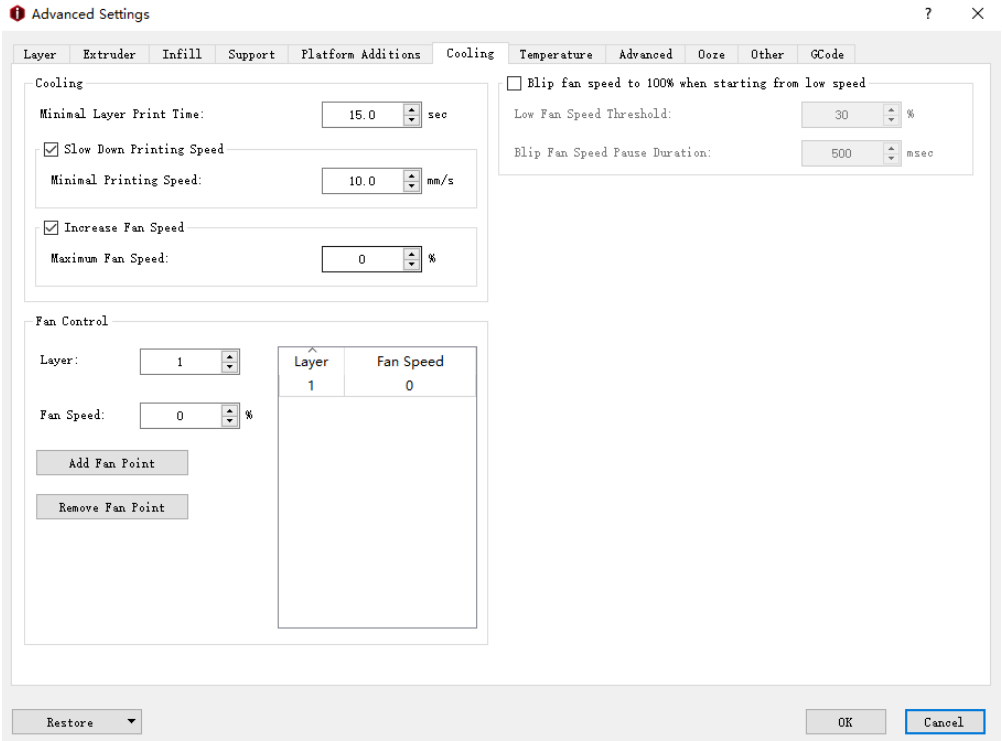


Figure 5.69: The Cooling tab.

5.1.3.7 Temperature

Temperature:

Heated Bed Temperature refers temperature of heated 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 select ***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*** settings can only be seen after selecting ***Extruder Count*** as 2 in ***Printer Settings***.

Use Temperature Control List refers that you can set different temperature values for specific layers with this option enabled.

Changing the temperature by clicking the ***Add Temperature*** button and input the temperature in right box. And delete it by selecting the value in right box and clicking the ***Remove Temperature*** button.

Cool Down Inactive Extruder refers to which this function enabled the inactive extruder will be cooled down during printing.

Park Position X refers that the park position in X direction is for heating up the inactive extruder before inactive extruder switched to active.

Park Position Y refers that the park position in Y direction is for heating up the inactive extruder before inactive extruder switched to active.

Inactive Cooling Temperature (Left) refers to cooling down the inactive left extruder to the target temperature.

Inactive Cooling Temperature (Right) refers to cooling down the inactive right extruder to the target temperature.

Heat up Inactive Extruder in Advance refers to the inactive extruder will be heated up before moving to park position.

Heat up Ahead of Time refers to the inactive extruder will start to heat up in seconds ahead of time before moving to park position.

Inactive Heating Temperature (Left) refers to the inactive left extruder will be heated up to the target temperature before moving to park position. Note: do not set it lower than printing temperature.

Inactive Heating Temperature (Right) refers to the inactive right extruder will be heated up to the target temperature before moving to park position. Note: do not set it lower than printing temperature.

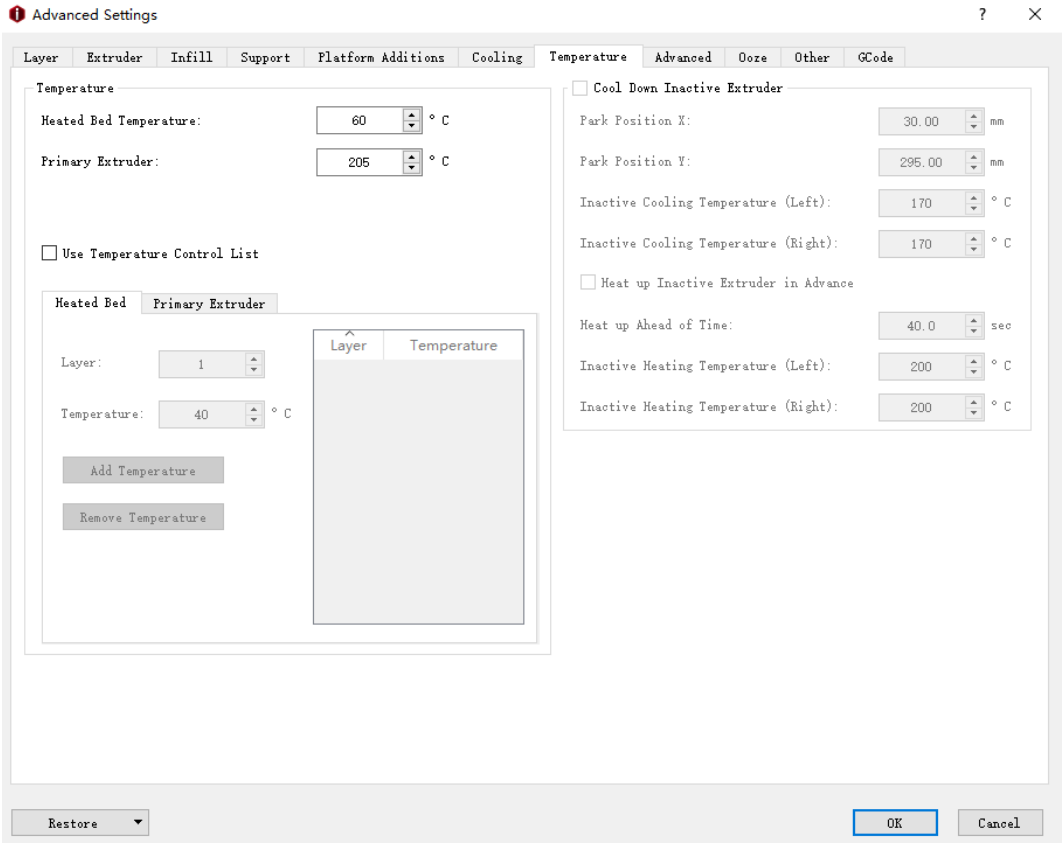


Figure 5.70: The Temperature tab.

5.1.3.8 Advanced

Enable Acceleration Control refers to enabling ***Acceleration Control*** function. The below settings will be activated such as ***Printing Acceleration.***

Acceleration is the rate of velocity changing of an object with respect to time.

Printing Acceleration refers to the printing acceleration speed for the non-specified area of the model.

Inner Shell Acceleration refers to the printing acceleration speed for the inner shell of the model.

Outer Shell Acceleration refers to the printing acceleration speed for the outer shell of the model.

Infill Acceleration refers to the printing acceleration speed for the infill structure of the model.

Bottom Solid Fill Acceleration refers to the printing acceleration speed for the bottom solid fill structure of the model.

Top Solid Fill Acceleration refers to the printing acceleration speed for the top solid fill structure of the model.

Travel Acceleration refers to the printing acceleration speed that the nozzle moves at from one place to another place without printing in X and Y directions.

Enable Jerk Control refers to enabling ***Jerk Control*** function. The below settings will be activated such as ***Printing Jerk.***

Jerk is the rate of acceleration changing.

Printing Jerk refers to the printing jerk speed for the non-specified area of the model.

Inner Shell Jerk refers to the printing jerk speed for the inner shell of the model.

Outer Shell Jerk refers to the printing jerk speed for the outer shell of the model.

Infill Jerk refers to the printing jerk speed for the infill structure of the model.

Bottom Solid Fill Jerk refers to the printing jerk speed for the bottom solid fill structure of the model.

Top Solid Fill Jerk refers to the printing jerk speed for the top solid fill structure of the model.

Travel Jerk refers to the printing jerk speed that the nozzle moves at from one place to another place without printing in X and Y directions.

Override Filament Settings:

Enable this function, you can edit the filament Flowrate. If you set the Flowrate to be 95% in ***Filament Settings***, but you enable this function and set it to be 90%, the Flowrate will change to 90%.

Left Filament Flowrate refers to the flow rate of left filament.

Right Filament Flowrate refers to the flow rate of right filament.

Note: If you want to print the PLA and ABS by Raise3D Pro2 series printer, you may need to change the Flowrate to be 90%.

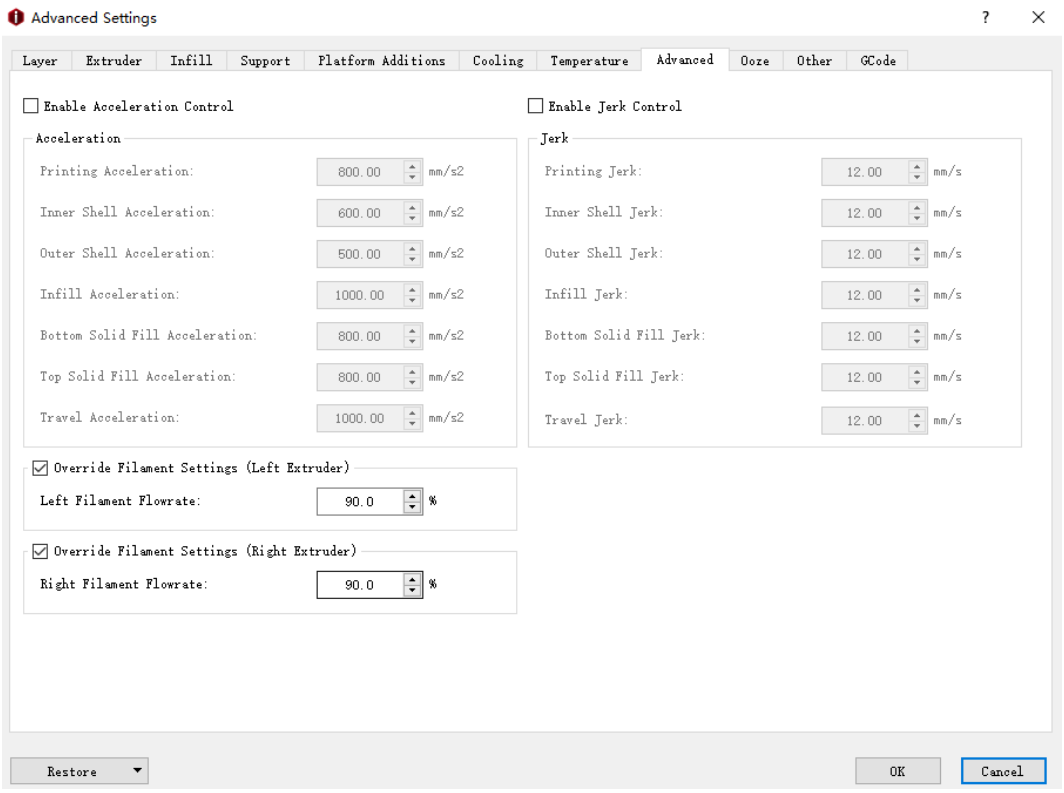


Figure 5.71: The Advanced tab.

5.1.3.9 Ooze

Other:

Force Retraction On Layer Change refers that the retraction will automatically be triggered when the current layer print completes and the next layer starts.

Force Retraction Before Traveling to Outer Shell refers that the reaction will automatically be triggered before the extruder start move to outer shell.

Avoid Retraction Inside Model refers to disabling retraction when printing the inner structures of the model.

Enable Retraction at Bottom and Top of models refers to that with this function enabled the retraction will be triggered at model's top and bottom layers even if the retractions are disabled inside models.

Bottom Layers refers to the number of bottom solid layers in which retraction will be enabled.

Top Layers refers to the number of top solid layers in which retraction will be enabled.

Extruder Switch Ooze Control refers to the retraction settings for extruder when one nozzle completes printing of one layer and the other nozzle starts printing.

Retraction Speed of Extruder-Switch refers to the speed of retraction when extruder switch.

Retraction Amount of Extruder-switch refers to the amount of filament retraction for an extruder change. 0 refers to no retraction at all.

Restart Speed of Extruder-switch refers to the extrusion speed after the retraction for an extruder change.

Extra Restart Amount of extruder-switch refers to the amount of additional filament compensation after the retraction for an extruder change.

Travel:

Avoid Traveling Through Holes refers to the printer will pass around the hole parts automatically to avoid leaving strings inside the holes.

Maximum Travel Path Length refers to that the extruder will move through the hole directly, if the travel path length of move around the hole is longer than the maximum value.

Enable Wipe Wall means when printing a model with dual-extrusion, a nozzle will print a few extra shells around the model. This wipe wall will help to clean the nozzle while printing.
Wipe Wall Offset refers to the distance between the outline of the model and the wipe wall.
Wipe Wall Angle refers to the maximum allowed angle change which the wipe wall will generate (from 0 to 90).

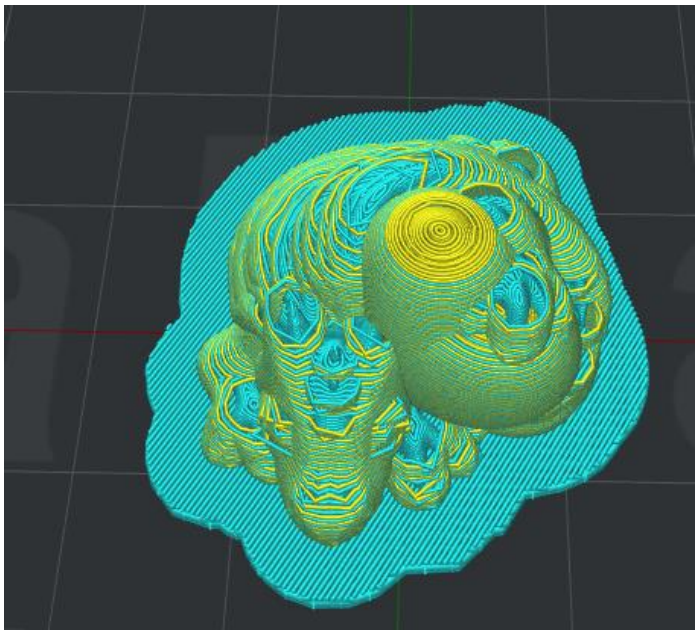


Figure 5.72: Set "Wipe Wall Angle" to be 90.

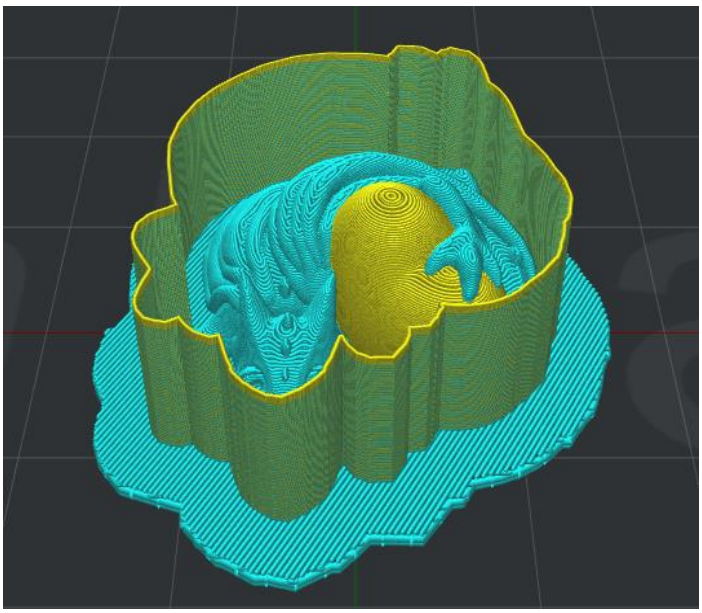


Figure 5.73: Set "Wipe Wall Angle" to be 0.

Wipe Wall Loop Lines refers to the quantity of the shells of wipe wall.

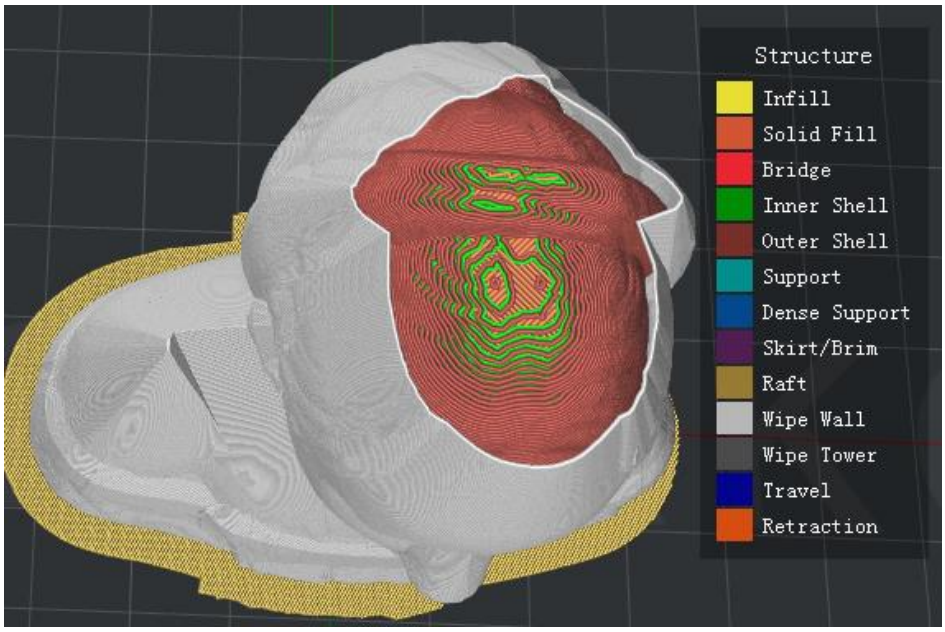


Figure 5.74: Set "Wipe Wall Loop Lines" to be one.

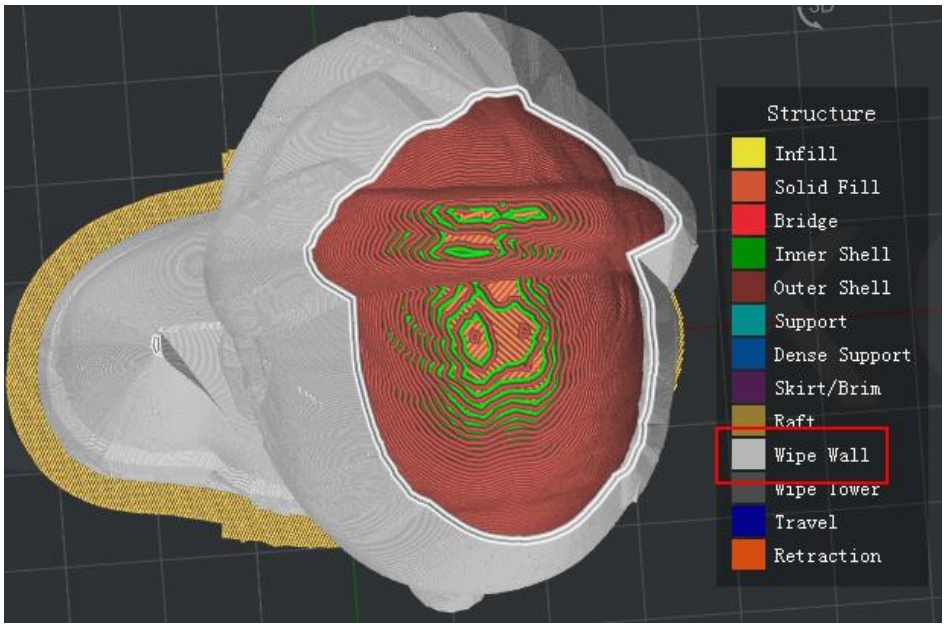


Figure 5.75: Set "Wipe Wall Loop Lines" to be two.

Wipe Wall Type refers to the shape of the wipe wall shells. The primary difference is the distance of model and wipe wall.

Contoured is the one which is closest to the origin model shape. ***Water fall*** is the type which will follow the origin shape, but a vertical drop if the lower structure is narrow that the above one. ***Vertical*** will find the widest part of the model and generate a vertical type wipe wall.

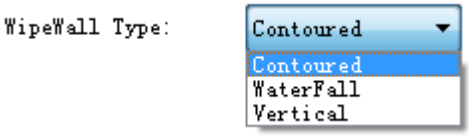


Figure 5.76: Three types of Wipe Wall.

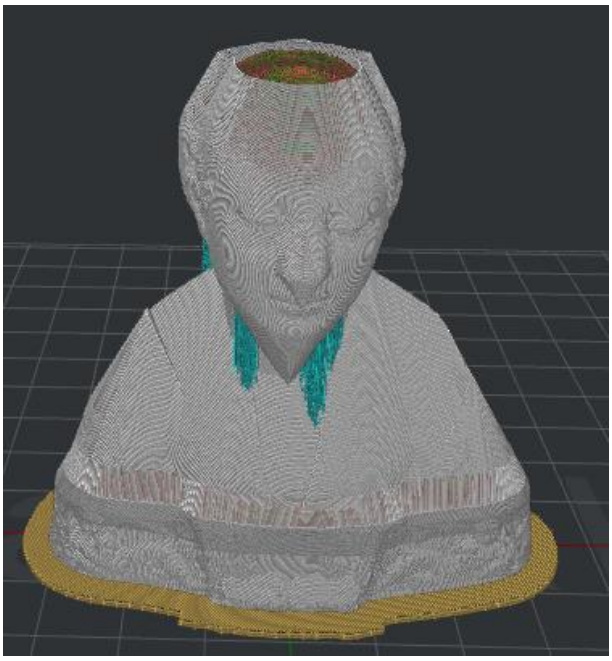


Figure 5.77: Set "Wipe Wall Type" to be "Contoured".

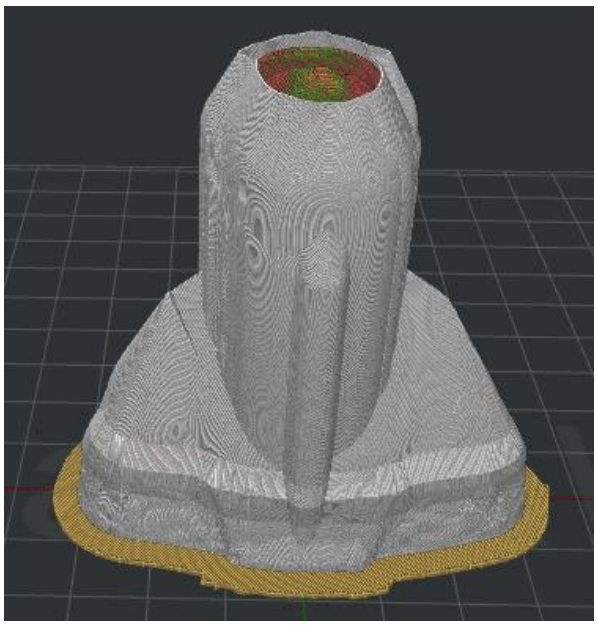


Figure 5.78: Set "Wipe Wall Type" to be "WaterFall".

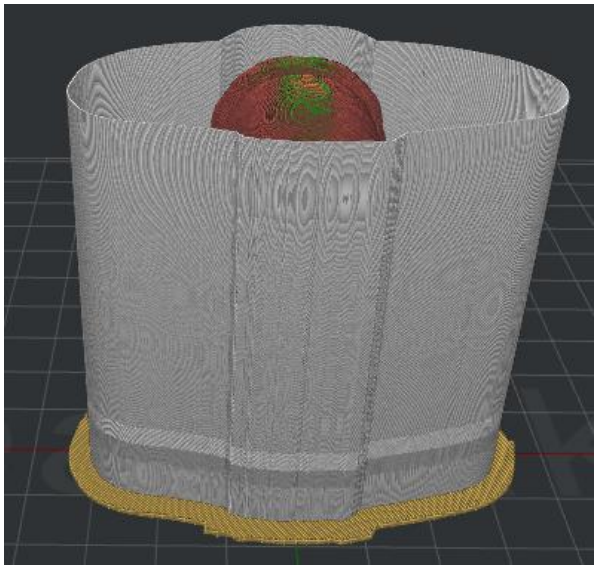


Figure 5.79: Set "Wipe Wall Type" to be "Vertical".

Wipe Wall Speed refers to the printing speed for wipe wall.

Wipe Tower refers to a cube created outside of the model which is used for wiping the nozzle which is going to start print.

Wipe Tower Width refers to width of the wipe tower.

Wipe Tower Infill Ratio refers to the density of wipe tower structure.

Wipe Tower Speed refers to the printing speed for wipe tower.

Placed at the Fixed Position of the plate refers to with this function enabled Wipe Tower's position would be fixed as ***Figure 5. 80***.

Wipe Tower Fixed Position X refers to the position in X direction.

Wipe Tower Fixed Position Y refers to the position in Y direction as ***Figure 5.81***.

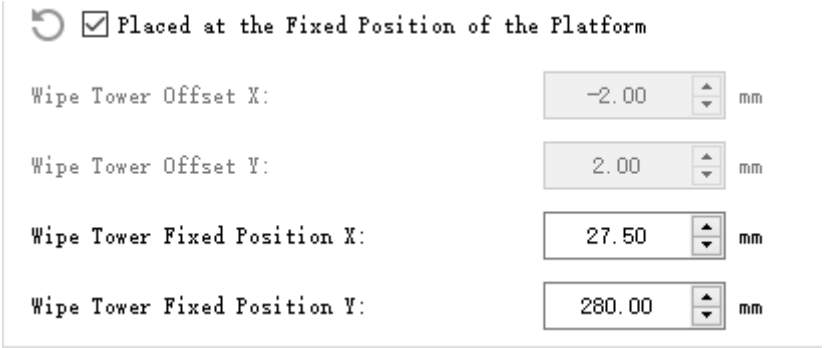


Figure 5.80: Wipe Tower Fixed Position.

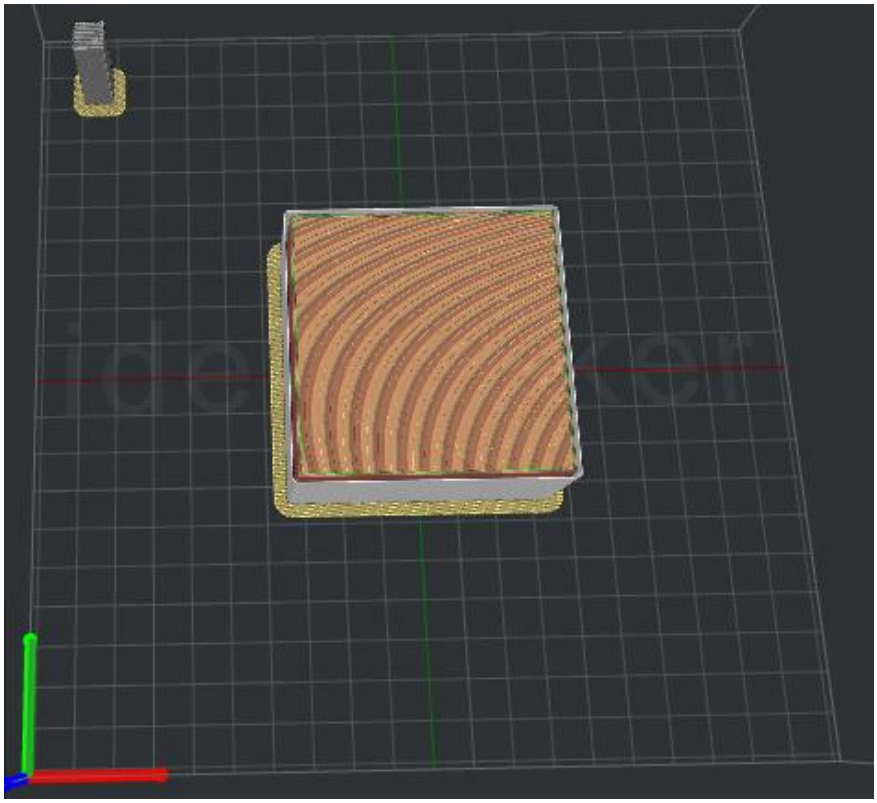


Figure 5.81: Enable the function of Placed at the Fixed of the Platform.

Wipe Tower Offset X refers to the distance between Wipe Tower and the model in the direction of X.

Wipe Tower Offset Y refers to the distance between Wipe Tower and the model in the direction of Y.

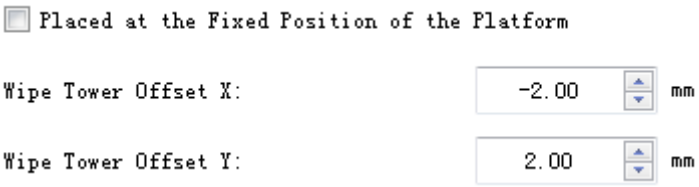


Figure 5.82: Wipe Tower X/Y offset.

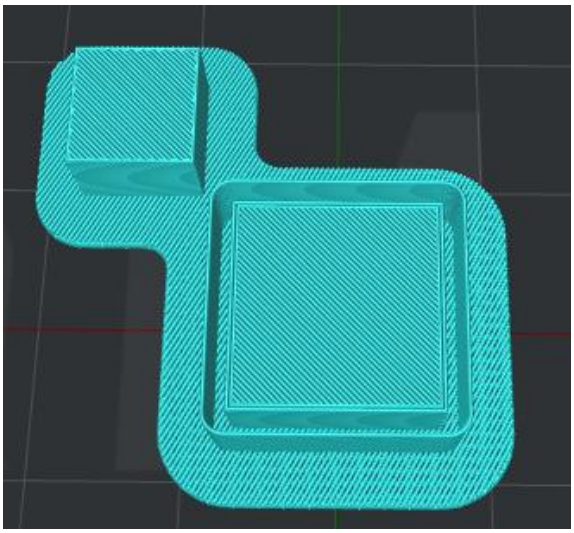


Figure 5.83: Wipe Tower.

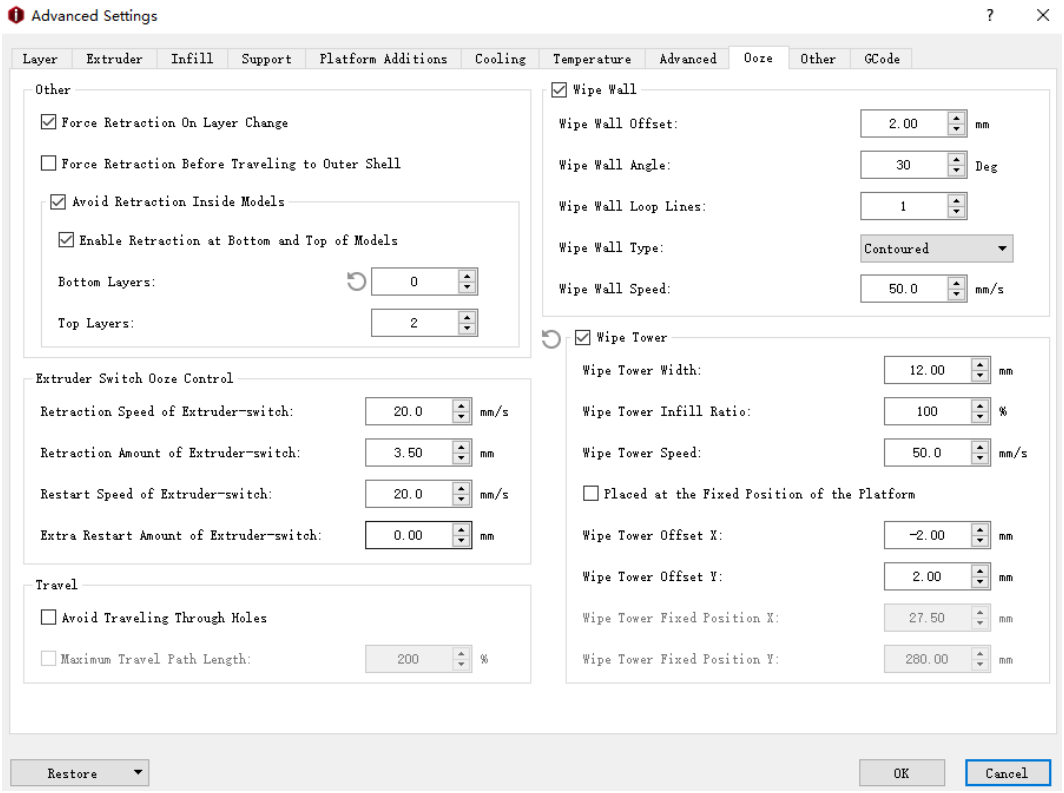


Figure 5.84: The Ooze tab.

5.1.3.9 Other

Repair:

Merge Open Segments of Model Parts means merging the non-closed polygons in the model parts.

Merge Internal Overlapping Parts means merging the parts which are overlapped in interior of the model.

Thin Wall:

Check Thin Wall (Single Extrusion Width) means that check if the width of the model parts is less than the defined thin wall width and extrude one single extrusion width to instead.

Minimal Extrusion Width Percentage refers to the minimal percentage of extrusion width.

Maximum Extrusion Width Percentage refers to the maximum percentage of extrusion width.

For Example, if the extrusion width is 0.4mm, set ***Minimal Extrusion Width Percentage*** to be 25% then the minimal thin wall will be 0.1mm. Set ***Maximum Extrusion Width Percentage*** to be 200%, then the minimal thin wall will be 0.8mm.

If item's width is less than 0.1mm, it will print nothing as the width has less than minimal extrusion width. If item's width is larger than 0.1mm, but less than 0.8mm, the system will extrude one single extrusion path instead. And the width of the single path will be adapted to the size of space dynamically.

Gap Filling:

Fill Gaps in shells refers that with this function enabled, there will no gaps between shells as ***Figure 5.83*** and ***5.84***.

Gap Fill Speed refers to the speed of printing the gap filling.

Allow Filling Gaps using Single Extrusion Filling refers that with this function enabled the gap will be filled by single extrusion filling.

Minimal Single Extrusion Width Percentage refers to the minimal percentage of single extrusion width.

Maximum Single Extrusion Width Percentage refers to the maximum percentage of single extrusion width.

For example: if the extrusion width is 0.4mm, set the minimal singe extrusion width percentage to be 50%, which means the single extrusion width is 0.2mm. Set the maximum singe extrusion width percentage to be 200%, which means the maximum single extrusion width is 0.8mm. If the gap is less than 0.2mm, it will not be filled. If the gap is larger than

0.2mm and less than 0.8mm, it will be printed by single extrusion filling. And the width of the single extrusion filling will be adapted to the size of space dynamically.

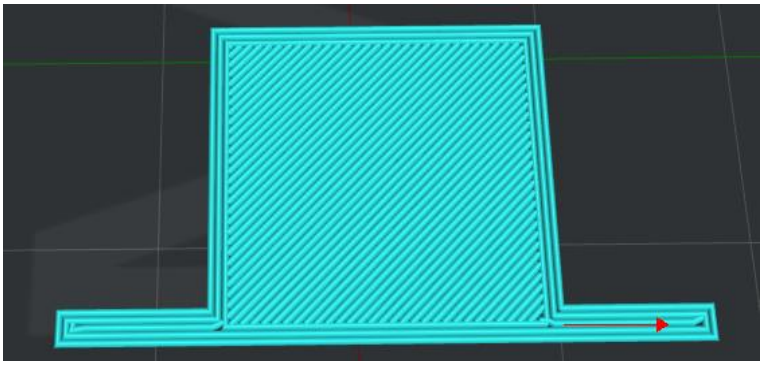


Figure 5.85: Enable the "Fill Gaps in shells".

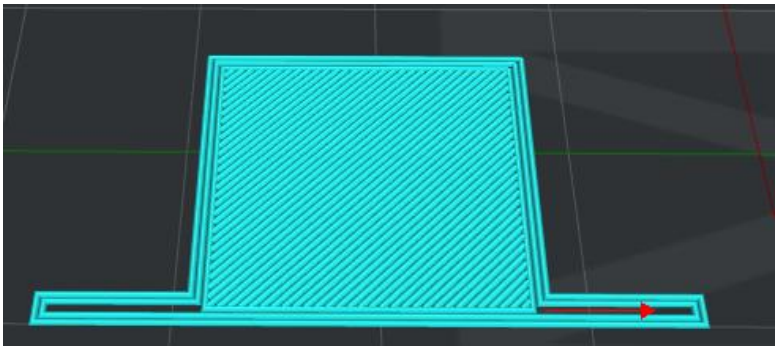


Figure 5.86: Unable the "Fill Gaps in shells".

Pause at Height means that the printer will automatically suspend a print job 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 selecting the value in right box and clicking the ***Remove Pause Point*** button.
Note: This height doesn't include ***Raft***.

Bridging:
Extrusion Width Percentage refers to the percentage of bridging extrusion width. For Example, if the extrusion width is 0.4 mm and you set this value to be 120% then the bridging structures extrusion width will be 0.48 mm.
Bridging Speed refers to the speed of printing the bridging.
Enable Bridging Fan Speed refers that with this function enabled the fan speed of bridging can be modified.

Enable Fixed Bridging Angle refers that with this function enabled the fill lines angle of bridging structures will be fixed as the value you set.

Minimal Allowed Area of unsupported Regain refers that if the overhang area is less than the value you set, the system will recognize that it is not bridging structure.

Global Offset means that means you can apply X/Y/Z offset to all coordinates in the gcodes. You can separately set ***X-Offset***, ***Y-Offset*** and ***Z-Offset***.

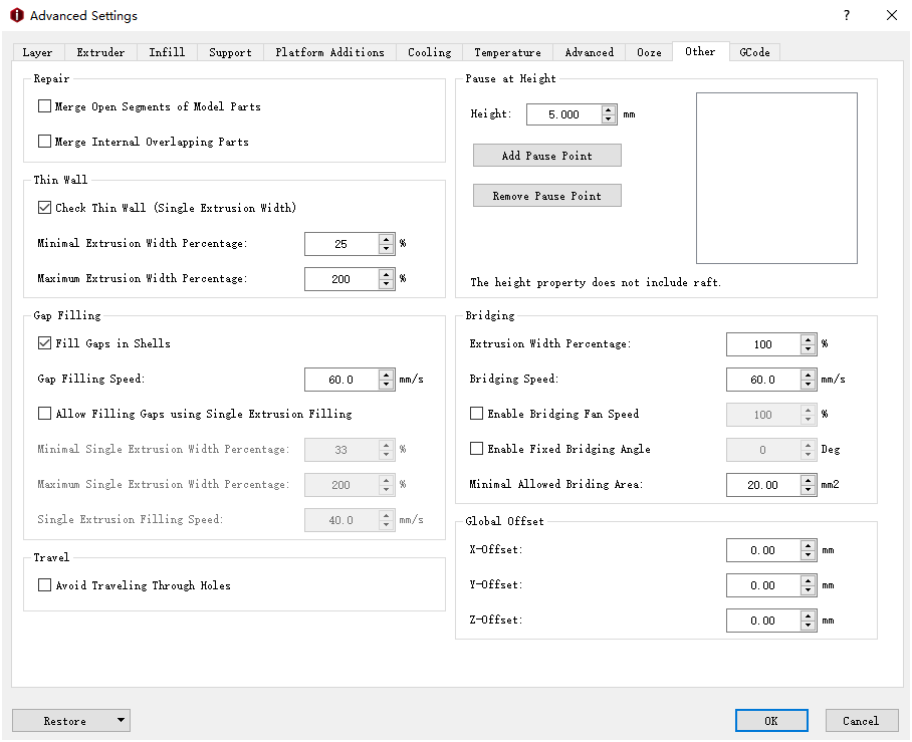


Figure 5.87: The Other tab.

5.1.3.10 GCode

Start GCode refers that custom gcodes which will be executed at the start of a print job (after the preheating extruders and the heat bed).

Start Gcode when the number of extruder is refers to selecting extruder's number to choose its start Gcode. There are two options. ***1*** refers to the start gcode when the number of extruders is set to be 1. ***2*** refers to the start gcode when the number of extruders is set to be 2. Single-extruder print and dual-extruder print have difference in Start Code.

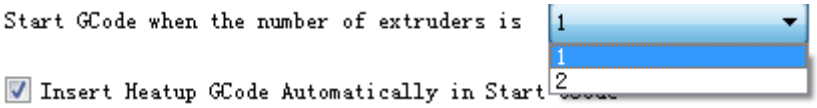


Figure 5.88: Selecting extruder's number for your Start Gcode.

Insert Heatup Gcode Automatically in Start Gcode refers that the heatup gcode will be inserted into start gcode automatically.
Heatup code should be inserted to the start gcode, if not the printer will start print without heating nozzle and heating heated bed. If you want to change the heatup gcode to be customer code, please disable this option and input your codes.

Heatup Sequence refers to the heating order.
Heatup heated bed and nozzle(s) simultaneously refers that with this function enabled the heated bed and nozzle(s) will be heated up at the same time.
Heatup nozzle(s) first refers that with this function enabled the nozzle(s) will be heated up first, then heating up heated bed.
Heatup heated bed first refers that with this function enabled the heated bed will be heated up first, then heating up nozzle(s).

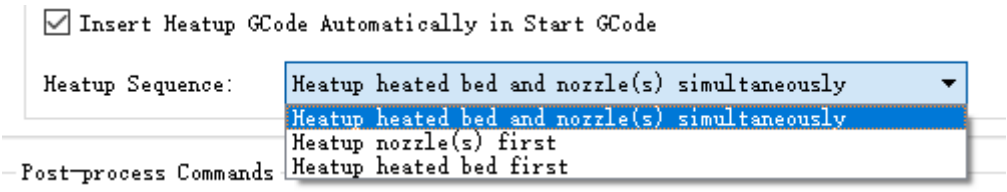


Figure 5.89: Heatup Sequence.

You can add more slice settings by clicking the ***Add More*** button, a dialog will pop-up as ***Figure 5.90*** and click ***ok*** to add the slice setting into right box. And delete it by selecting the value in right box and clicking the ***Remove Selected*** button.

You can add the Placeholder to left code box by double-clicking the slice settings from the right list.

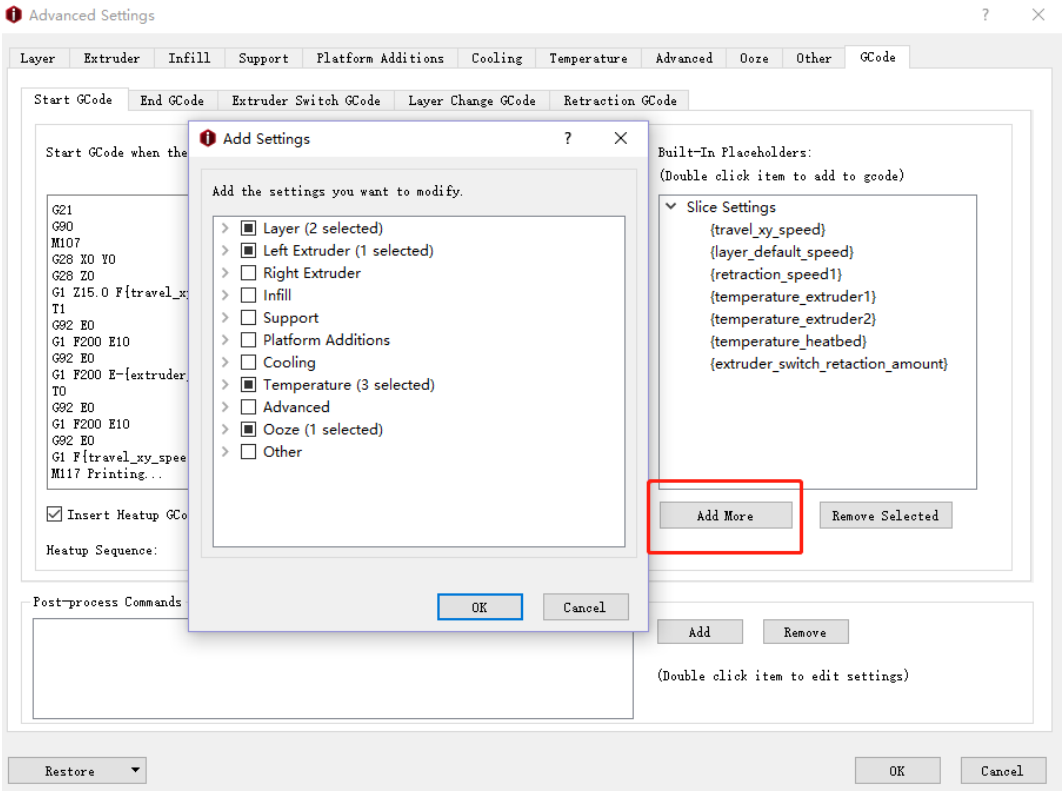


Figure 5.90: Adding more slice settings to placeholders.

Post-process Command refers that the gcode can be delete or replace in batches.

Add refer to inserting the command into the ***Post –process command***.

Replace SOURCE with TARGET refers that the occurrences under the ***TARGET*** tab will be replaced with the occurrences which are under the ***SOURCE*** tab.

For example: Insert "G92 E0" into ***Source*** tab and insert "G1 X20 Y0 F140 E30" into ***Target*** tab as ***Figure 5.91***, then set the Command type to be ***Replace SOURCE with TARGET***. Then press ***OK*** button the Post-process Command will be edit as ***Figure 5.92***.

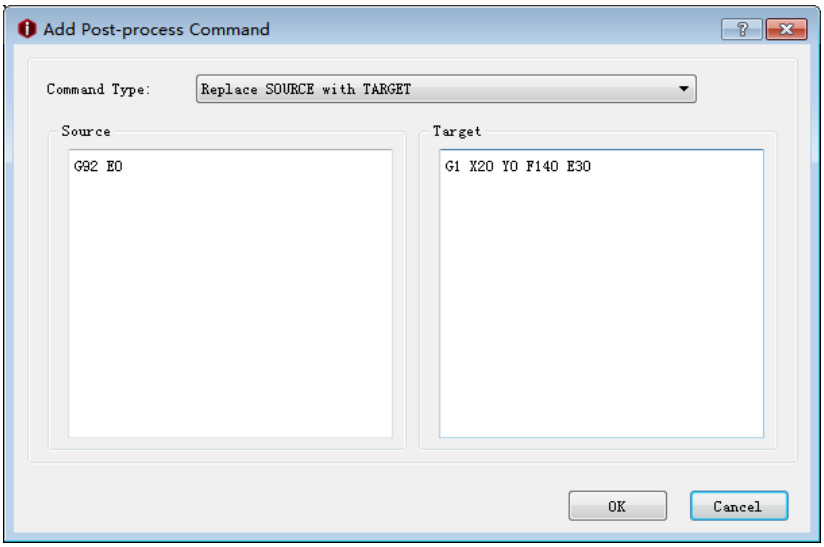


Figure 5.91: Insert the occurrences into Source and Target tab.

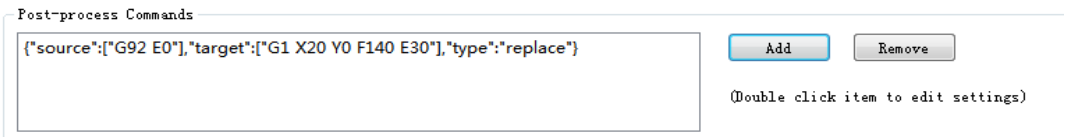


Figure 5.92: The Post-process Command tab.

Remove all occurrences of Source refers that with this function enabled the occurrences will be removed which is under the SOURCE tab.

For example: Insert "G92 E0" into ***Source*** tab and insert "G1 X20 Y0 F140 E30" into ***Target*** tab as ***Figure 5.93***, then set the Command type to be ***Remove all occurrences of Source***. Then press ***OK*** button the Post-process Command will be edit as ***Figure 5.94***.

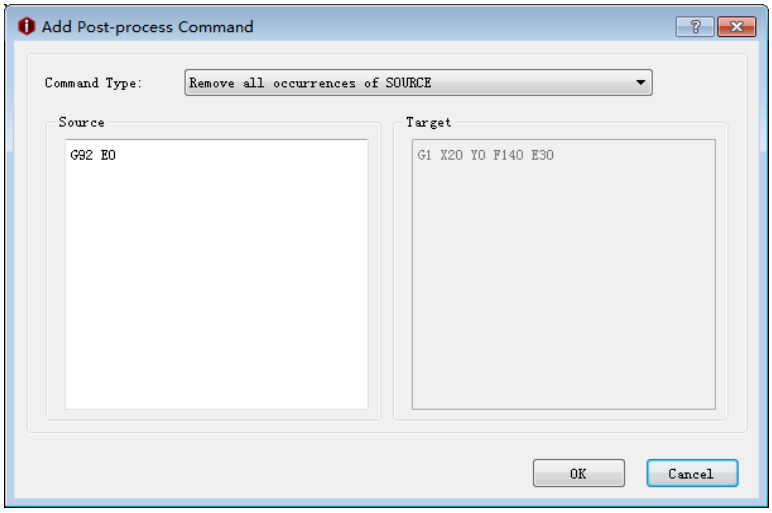


Figure 5.93: Insert the occurrences into Source and Target tab.

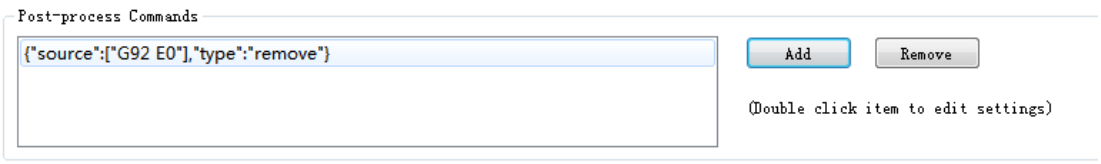


Figure 5.94: The Post-process Command tab.

Remove refers to deleting the command from ***Post-process Command*** tab.

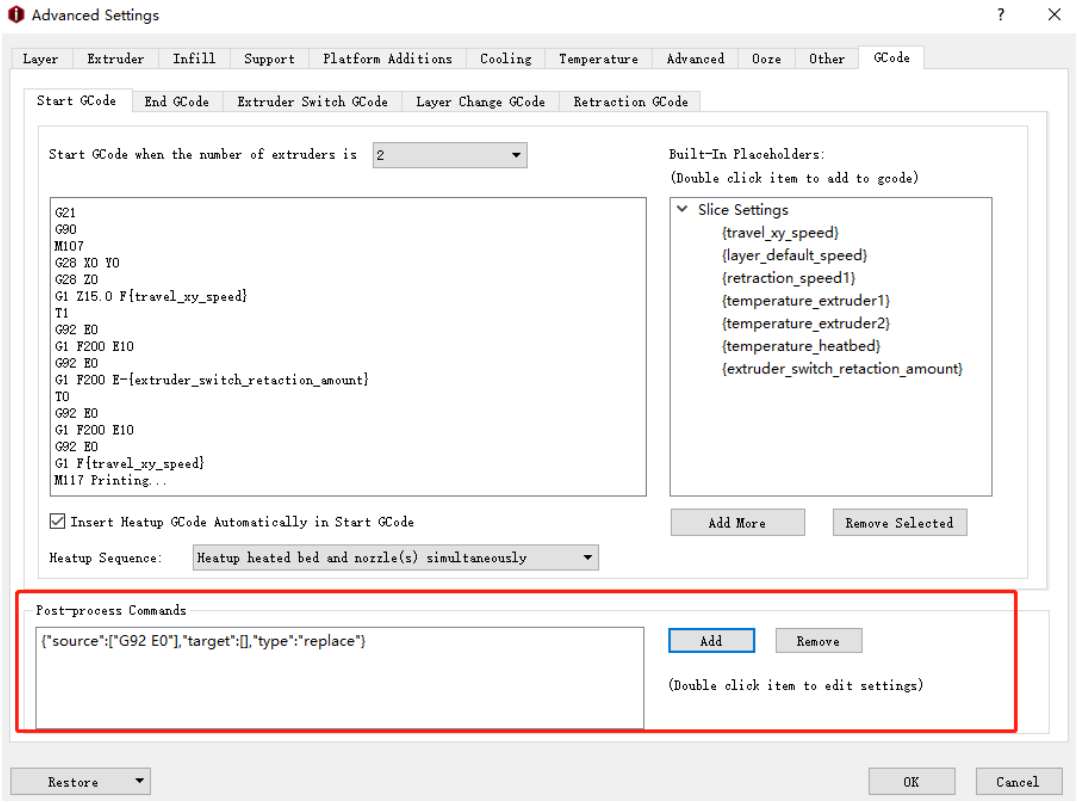


Figure 5.95: The Post-process Command tab.

End Gcode refers to the gcodes being executed at the end of a print job.

Select the extruder number you want to check by selecting ***Number of Extruders***. ***1*** refers to the end gcode when the number of extruders is set to be 1. ***2*** refers to the end gcode when the number of extruders is set to be 2.

Turn off Fan Cooling in End Gcode refers that with this function enabled the controllable cooling fan will be turned off at the end.

Reset Global Flow Rate to 100% in End Gcode refers that with this function enabled the global flow rate will be changed to be 100% at the end.

Extruder Switch Gcode refers that custom gcodes will be executed when extruder change.

Layer Change Gcode refers that custom gcodes will be executed when layer change.

Retraction Gcode refers to the gcodes being executed when filament retracted.

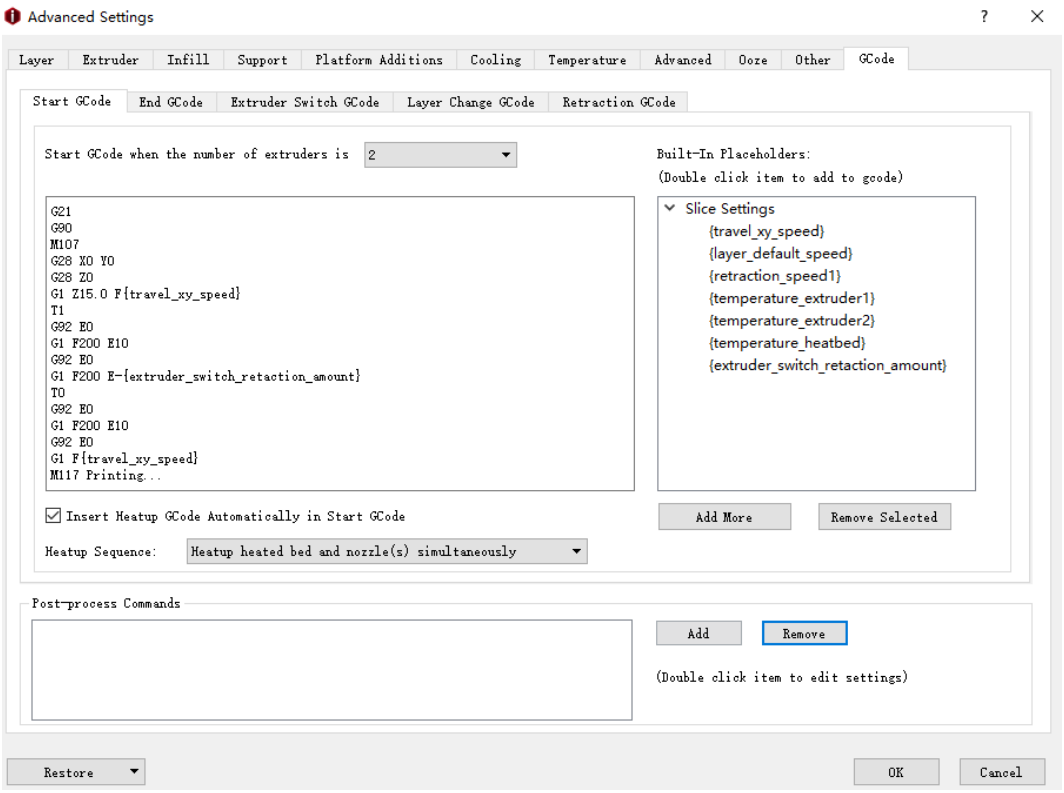


Figure 5.96: The GCode tab.

5.1.4 Compare Template

Compare refers to comparing selected templates. You can choose different printer, filament and template. And you also can import the templates from local disk.

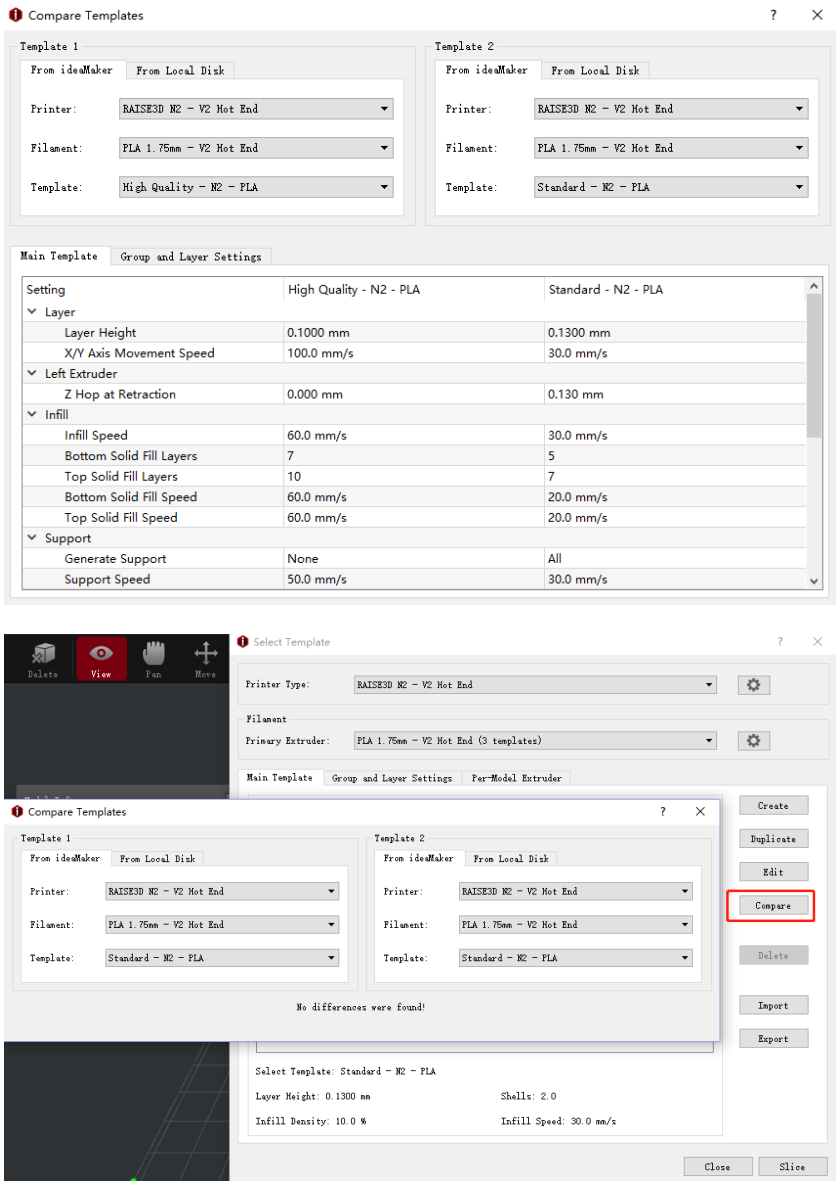


Figure 5.97: Compare slice templates.

5.1.5 Delete Template

Delete refers to removing selected template.

Note: delete function only work for the template which was added by ***Create*** or ***Duplicate*** or ***Import***, the default templates cannot be deleted.

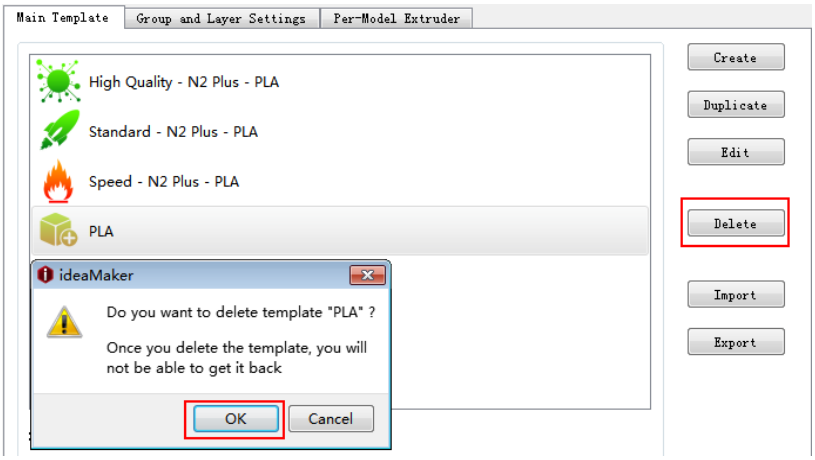


Figure 5.98: Delete slice template.

5.1.6 Import Template

Import refers to importing a slice template (.bin or .date file) from your computer as ***Figure 5.98.***

Template Name refers to the template's name which you import, and you also can rename it by yourself.

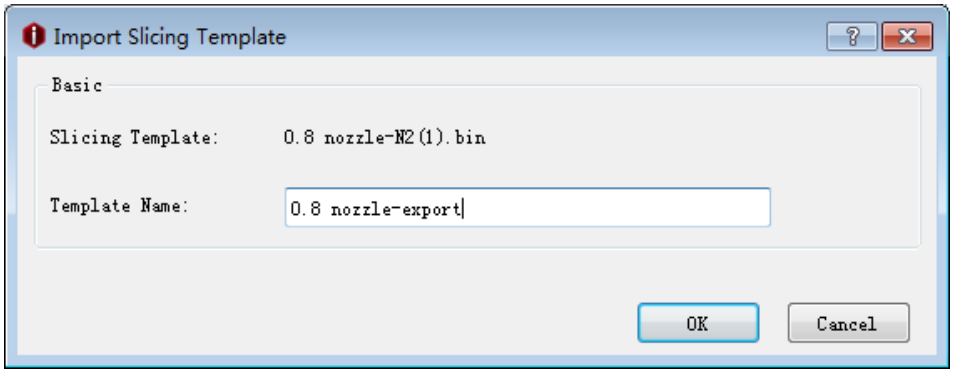


Figure 5.99: Import Slicing Template.

If you want to import a slicing template whose printer type or filament type does not currently exist in ideaMaker, ideaMaker will ask you whether you want to create a new type or not. If yes, it can directly create a new filament or printer type for your template during importing.

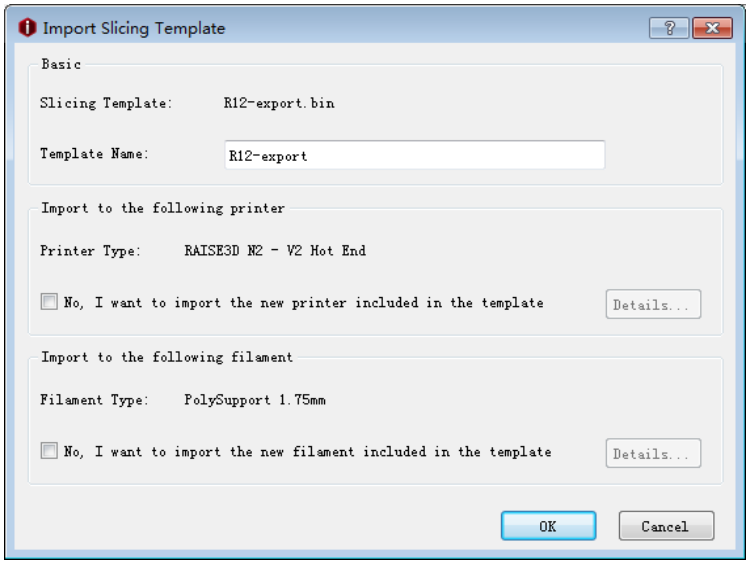


Figure 5.100: Import Slicing Template.

Check ***No, I want to import the new printer included in the template*** to import its Printer Type and Filament Type together as ***Figure 5.101***.

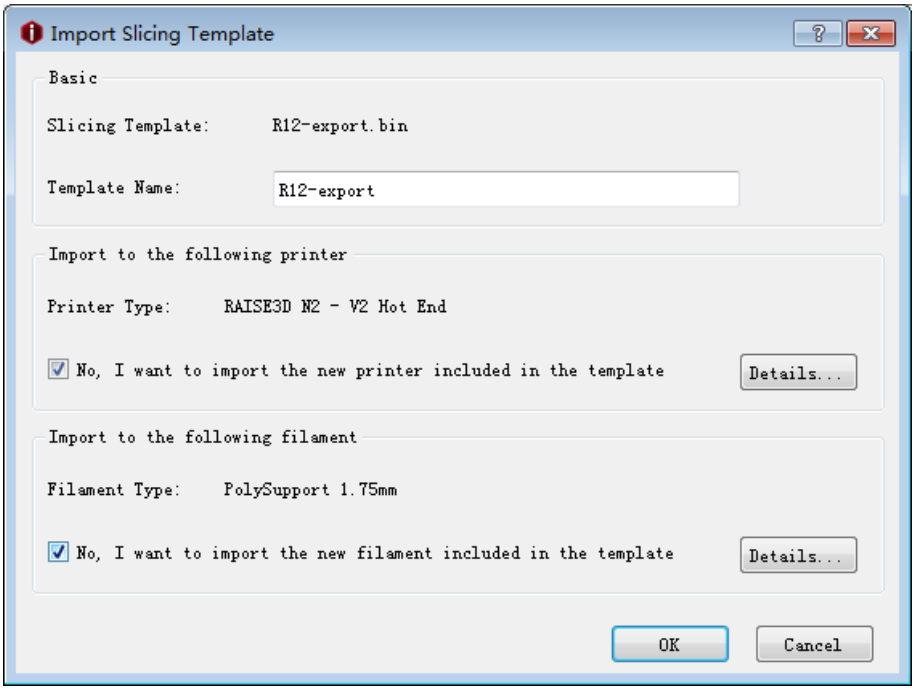


Figure 5.101: Whether to import the information of printer and filament type.

Click ***Details*** button to check or edit the name and setting of the template as ***Figure 5.102***.

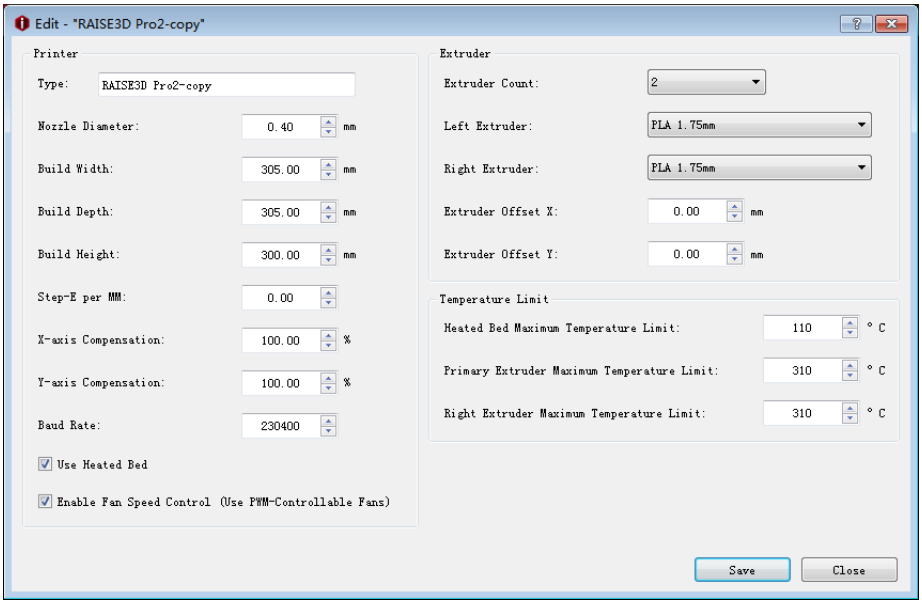


Figure 5.102: Details of "Import Slicing Template".

5.1.7 Export Template

Export refers to export one selected template from ideaMaker to your computer.

When export a slicing template, ideaMaker will ask if you want to export the group and layer settings together as ***Figure 5.103***.

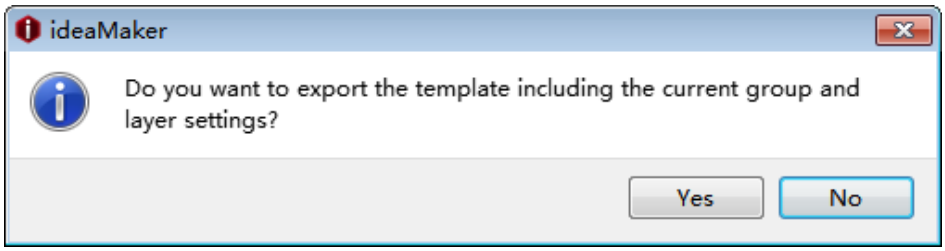


Figure 5.103: Export the slicing template.

5.2. Group and Layer Settings

Group and Layer Settings refers that you can divide models to be several printing groups to carry out different printing settings during one printing or different settings for different height.

5.2.1 Setting Group

Settings Group refers that you can set several setting groups for your model. You can set different settings for each setting group. One group can include several models and one model can only be assigned to one setting group.

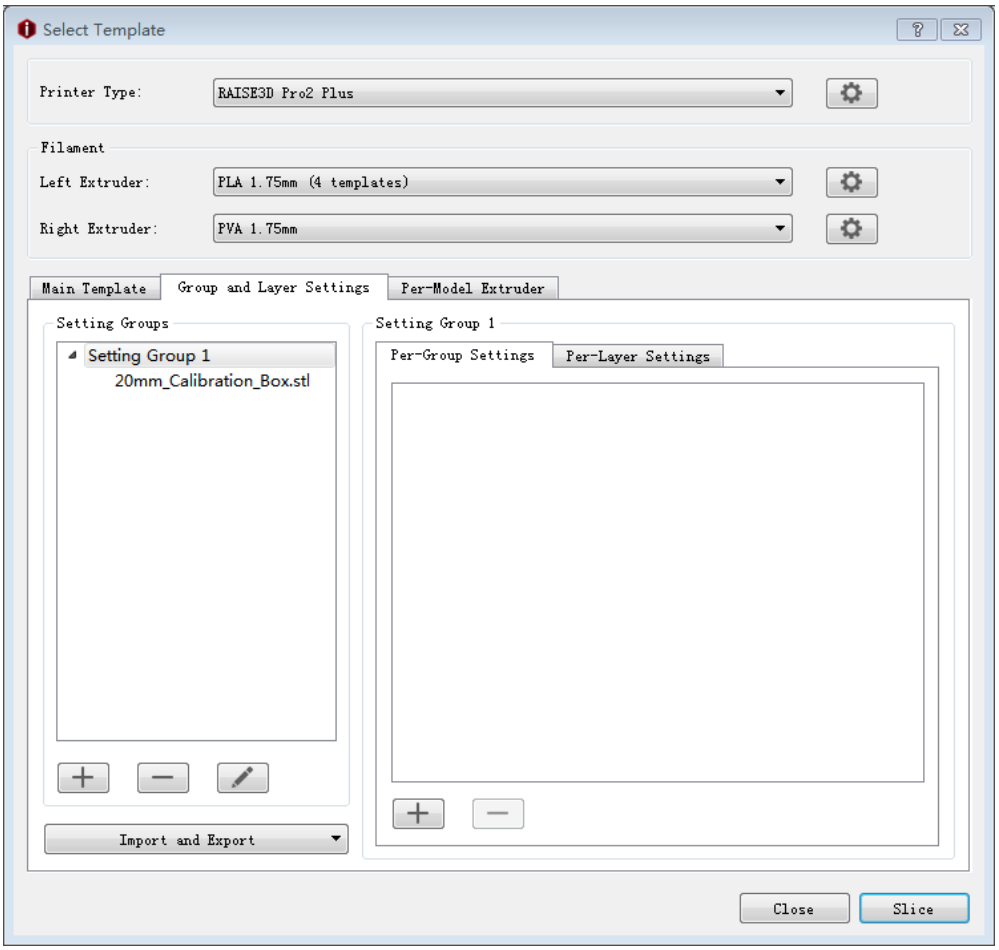


Figure 5.104: The Group and Layer Settings tab.

5.2.1.1 Add New Setting Group

: refers to add a new setting group.

A dialog will pop-up after pressing ***"+"*** button as ***Figure 5.105***, please select models from ***Setting Groups*** list and press right-arrow button to move it to ***New Setting Group*** list or select a model in ***New Setting Group*** and press left-arrow button to move it to ***Setting Groups*** list.

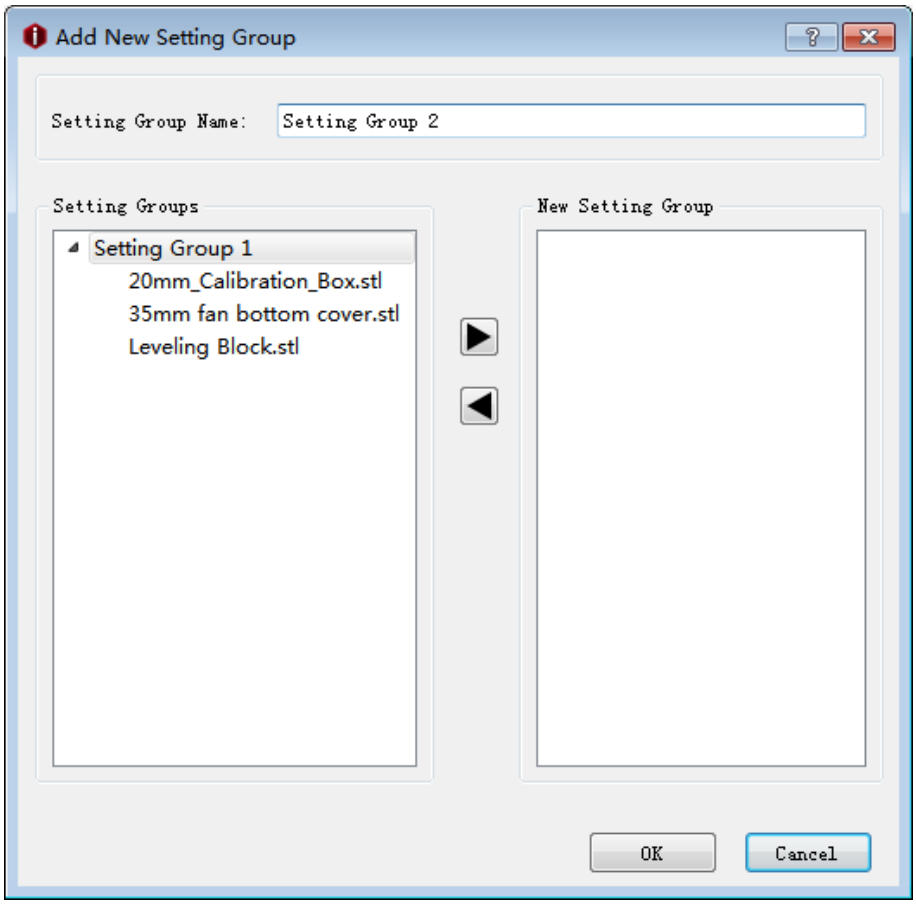


Figure 5.105: Add New Setting Group.

Setting Group Name refers to the new settings group's name, you also can rename it by yourself.

: refers to add selected model to new setting group.

: refers to remove the selected model from new setting groups.

5.2.1.2 Delete setting groups

: refers to deleting setting groups.

Note: this function only works when there are more than two setting groups in ***Setting Groups*** and only works to empty settings group which has no model inside.

5.2.1.3 Edit the selected setting group

: refers to editing the selected setting group.

Select a setting group and press ***Edit*** button, the interface will be shown as ***Figure 5.106***, you can delete (or add) models from (or to) the selected setting group.

Setting Group Name refers to the name of the setting group you are editing, you also can rename it by yourself.

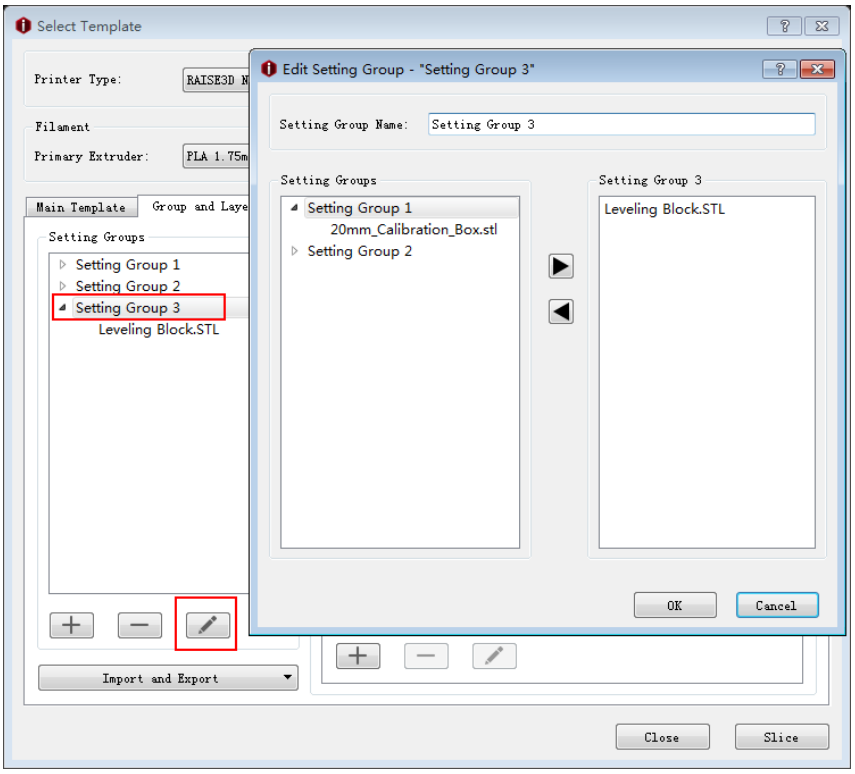


Figure 5.106: Edit Setting Group.

5.2.1.4 Add Settings

: refers to adding settings for the setting group which you selected.

There are several settings you can add into setting group and modify.

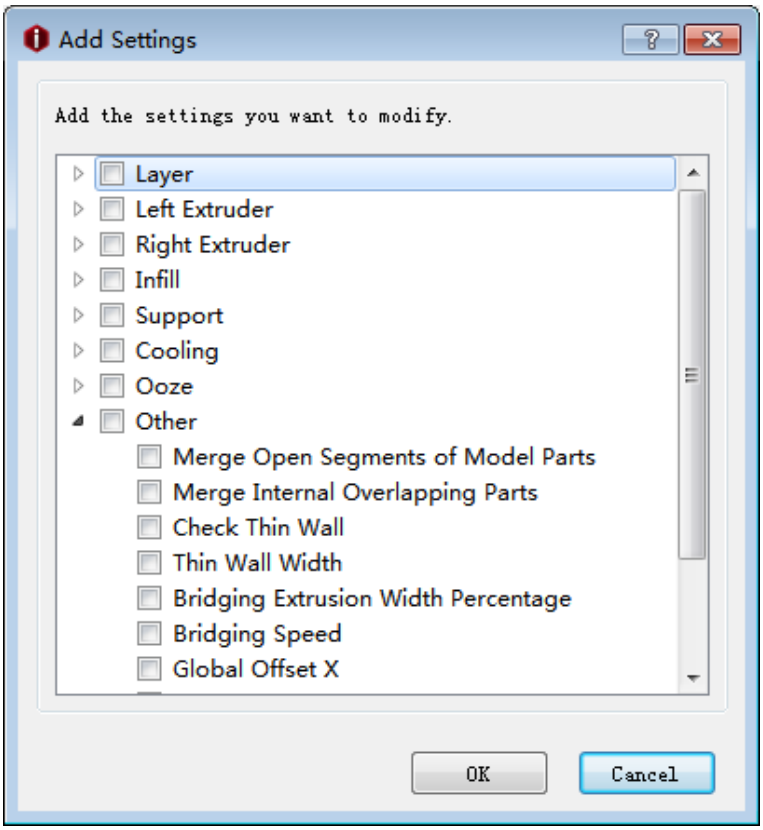


Figure 5.107: Add Settings.

For example: Add Layer settings into setting group 1. There are 15 functions you can edit, such as ***Layer Height, Shells*** and so on.

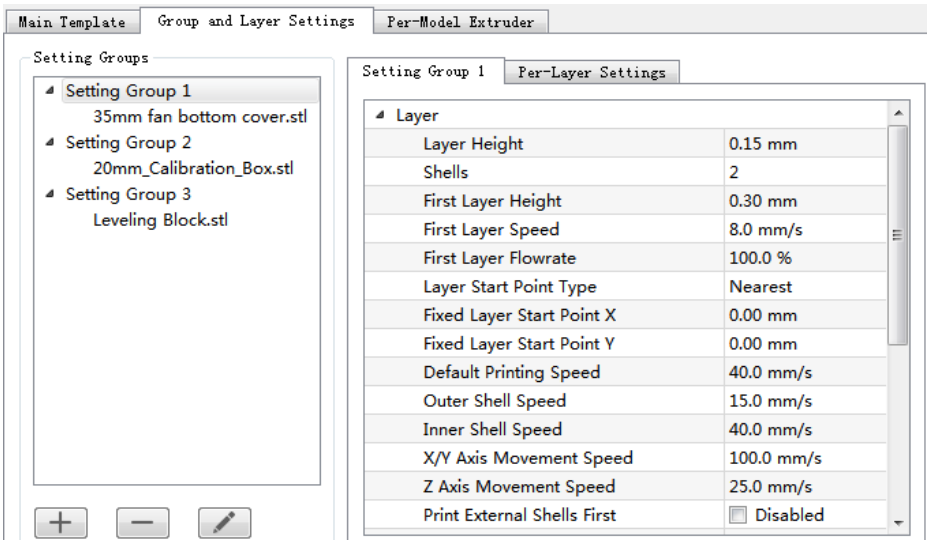


Figure 5.108: The Add Settings tab.

5.2.1.5 Delete Settings

: refers to deleting the selected settings, such as ***Layer*** or ***Layer Height***, ***Shells*** and so on.

5.2.2 Per-Layer Settings

Per-Layer Settings refers that you can apply different slice settings for different height ranges of the model(s) in the setting group.

5.2.2.1 Add Per-Layer Settings

: refers to adding settings for the setting group which you select.

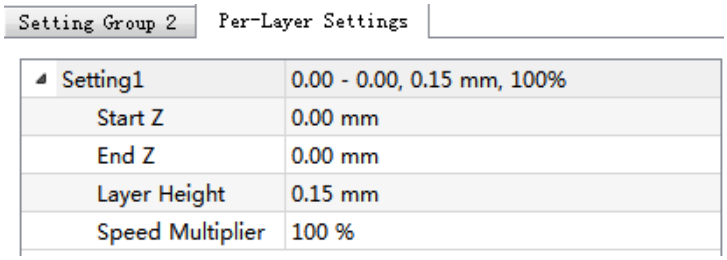


Figure 5.109: Set the layer height for different layers.

5.2.2.2 Delete Per-Layer Settings

: refers to deleting selected Per-Layer settings.

5.2.2.3 Edit Per-Layer Settings

: refers to editing the selected Per-Layer settings. In ***Figure 5.110*** you can edit the layer height for the define height range in setting1.

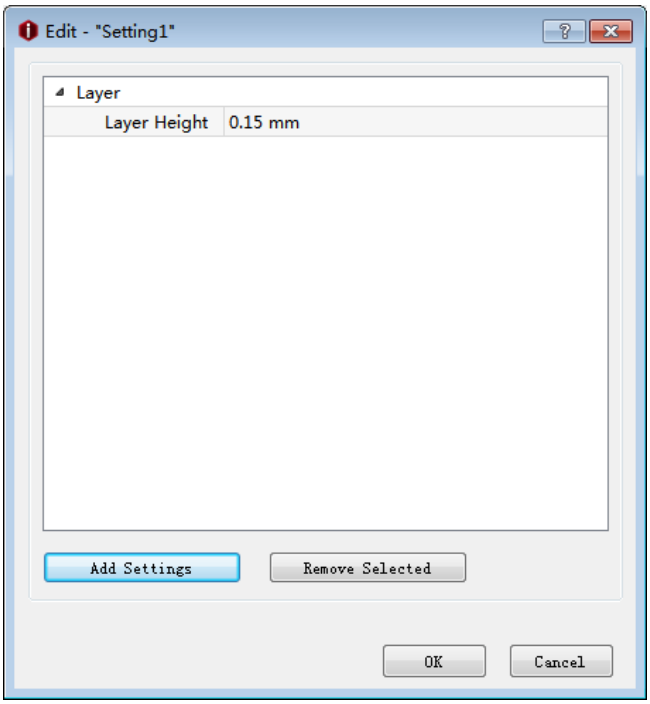


Figure 5.110: Edit the layer height for the "Per-Layer Settings".

Add Settings refers to adding more settings into the ***Per-Layer Settings*** such as ***left Extruder***, ***Right Extruder***, ***Infill*** and so on.

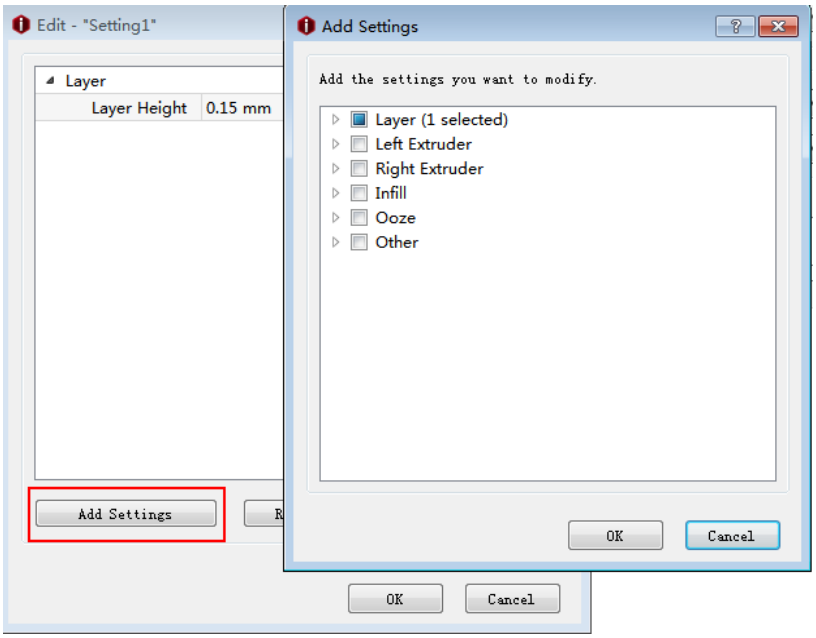


Figure 5.111: Add other settings into the Per-Layer setting.

Remove Selected refers to deleting the selected settings from the ***Per-Layer Settings.***

5.2.3 Import and Export Group and Layer Settings

Import from Last Used Group and Layer Settings refers to resetting the Group and Layer Settings to the ones you have used last time.

Import from .idea refers to importing the group and layer settings from the idea file.

Import from Local Disk refers to importing the group and layer settings from the local disk.

Clear Group and Layer Settings refers to deleting all groups and layer settings which you are importing before.

Export the Local Disk refers to exporting the group and layer settings to the local disk

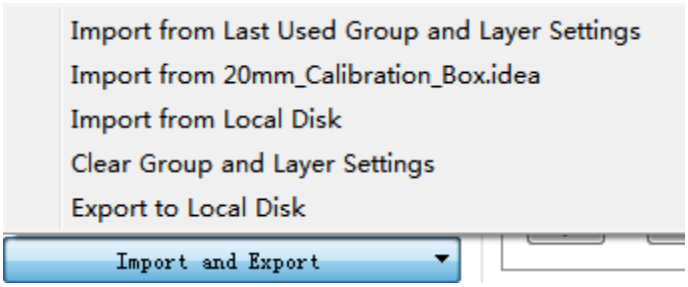


Figure 5.112: Import and export group and layer settings.

5.3 Per-Model Extruder

Per-Model Extruder refers to choosing an extruder for your model. If you have set the ***Extruder Count*** to be 2, this interface will show ***Left Extruder*** and ***Right Extruder***. If you have set the ***Extruder Count*** to be 1, this interface will show ***Primary Extruder*** only. And we define the left extruder as primary extruder by default.

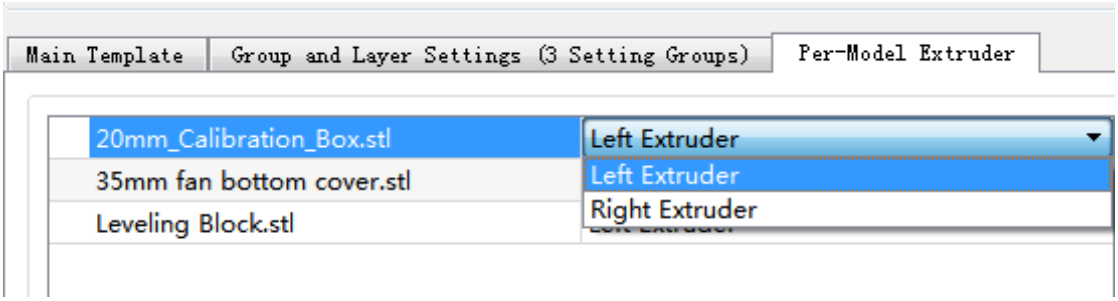


Figure 5.113: Choose an extruder for your model.

6 Multiple Extruders

6.1 Calibrate XY offset

Printer->Printer Settings-> Extruders Count

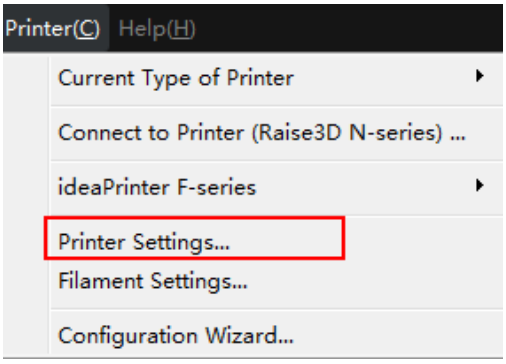


Figure 6.1: Select "Printer" ->"Printer Settings" from menu bar.

Select 2 in ***Extruder Count***, and assign the filament type to each nozzle.

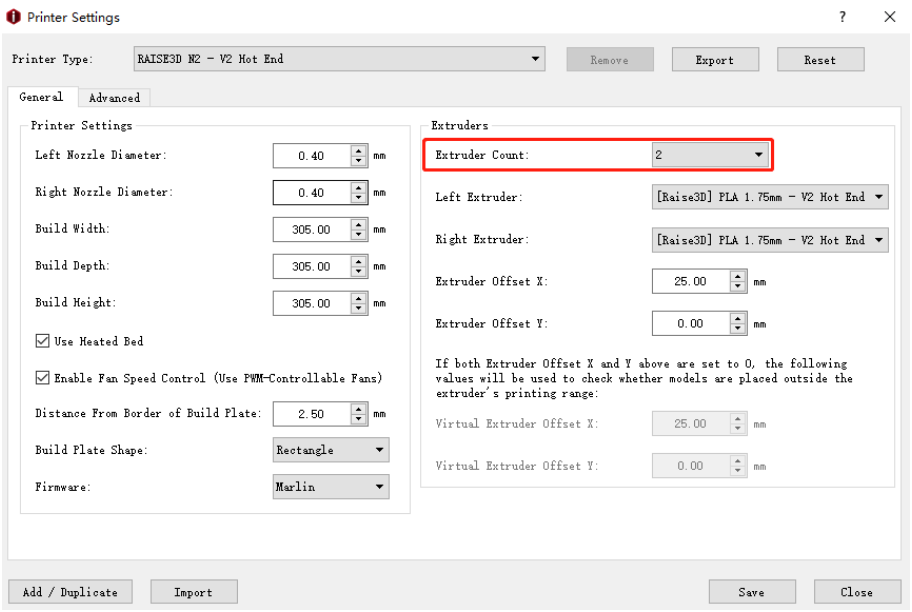


Figure 6.2: Set the "Extruder Count" to be 2. And set filament type for each extruder.

Extruder offset X refers to the offset between two nozzles in the direction of X. And we set left extruder as default nozzle. So when it comes to calculate the right extruder's position is to minus 25mm offset (which you set for X offset) in X direction.
Extruder offset Y refers to the offset between two nozzles in the direction of Y. And we set left extruder as default nozzle. So when it comes to calculate the right extruder's position is to minus 0mm offset (which you set for Y offset) in Y direction.

We may meet four cases:

First, there is a gap between two calibration boxes as ***Figure 6.3***:

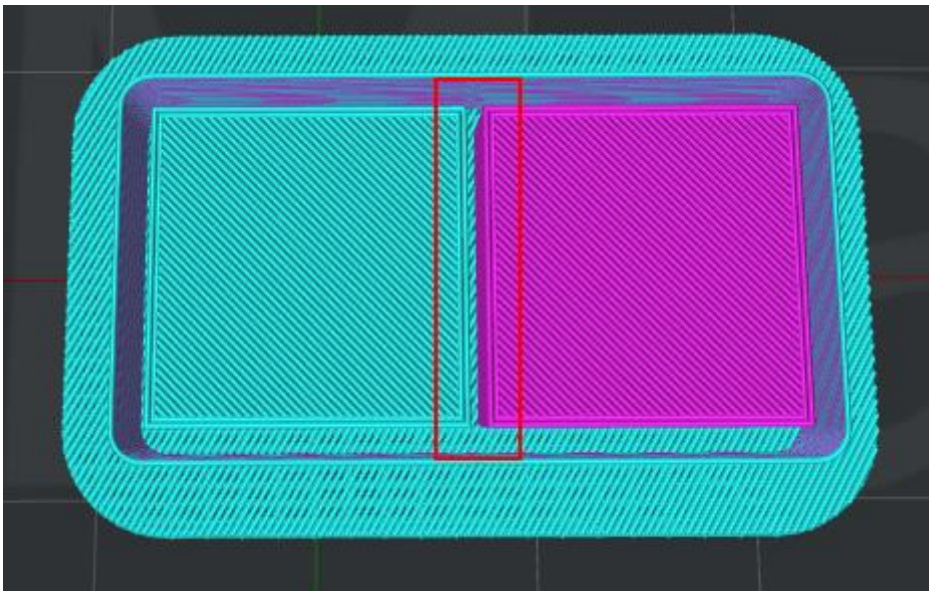


Figure 6.3: There is a gap between two calibration boxes.

That means the right nozzle needs to be closer to the left nozzle, and the default distance between right and left nozzle is 25mm.
So we need to measure how big the gap is and figure out the exact value which should be more than 25mm then refill it into the page below.

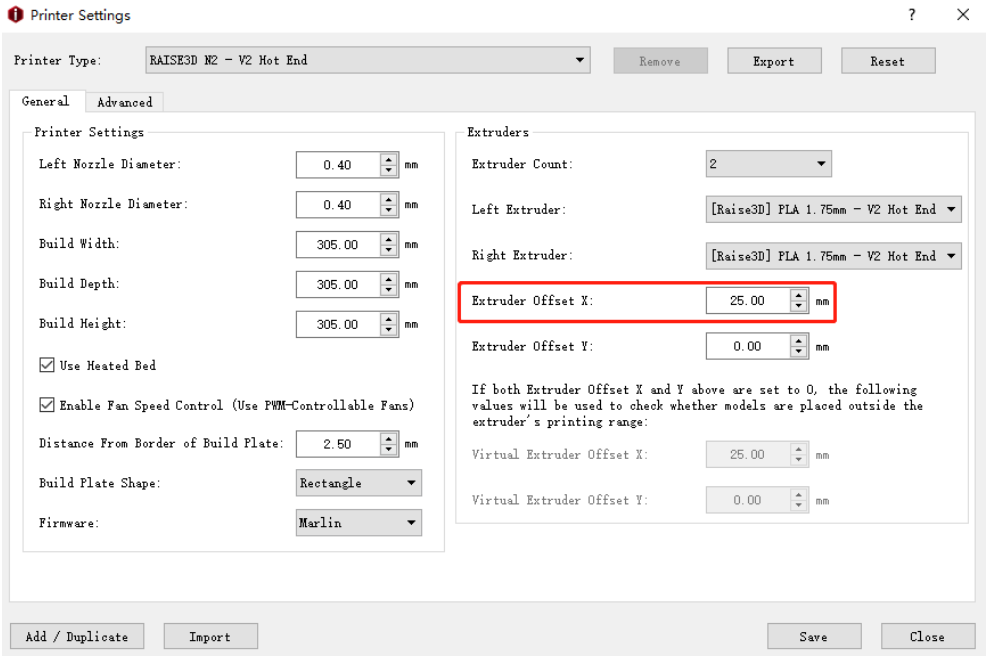


Figure 6.4: Fill the value of "Extruder Offset X".

Second, there is a superposition between two calibration boxes as ***Figure 6.5***:

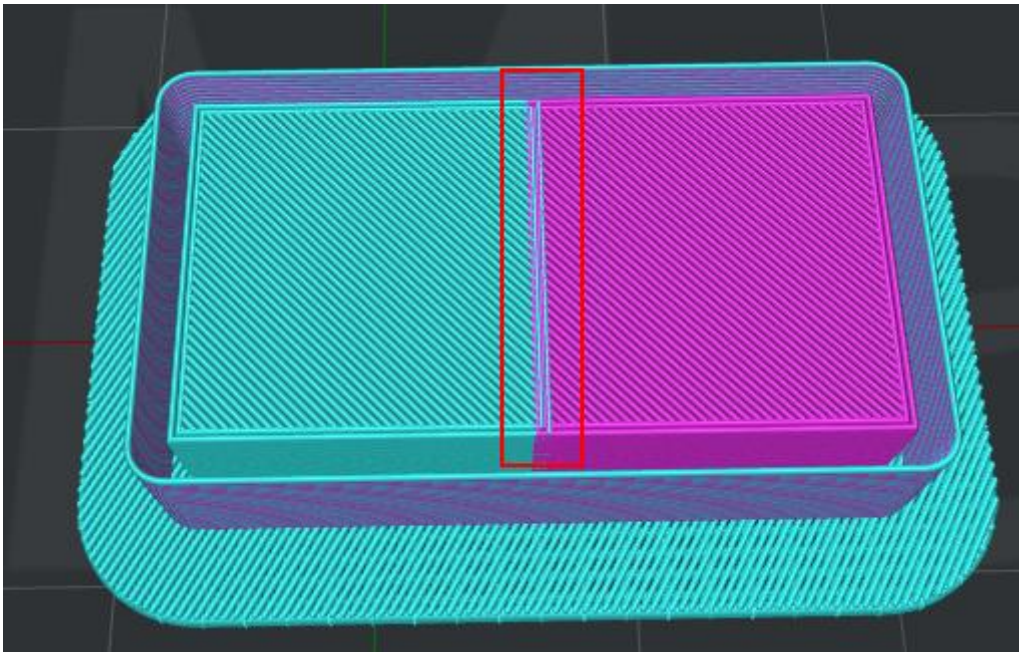


Figure 6.5: There is a superposition between two calibration boxes.

That means the right nozzle needs to be further to the left nozzle, and the default distance between right and left nozzle is 25mm.

So we need to measure the superposition's value and figure out the exact value which should be less than 25mm then refill it into the page below.

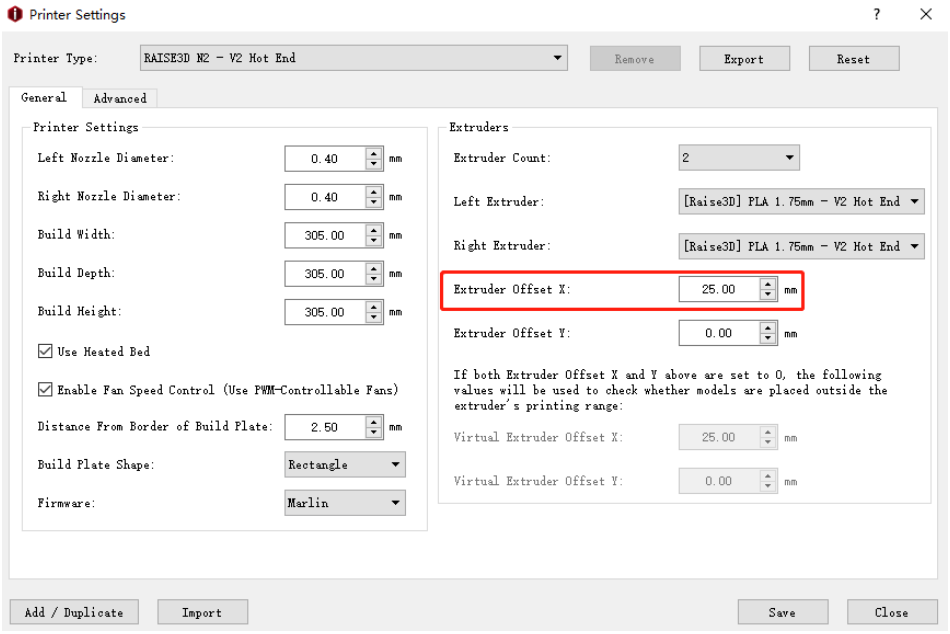


Figure 6.6: Fill the value of "Extruder Offset X".

Third, the right calibration box is backward more than the left calibration box as ***Figure 6.7***:

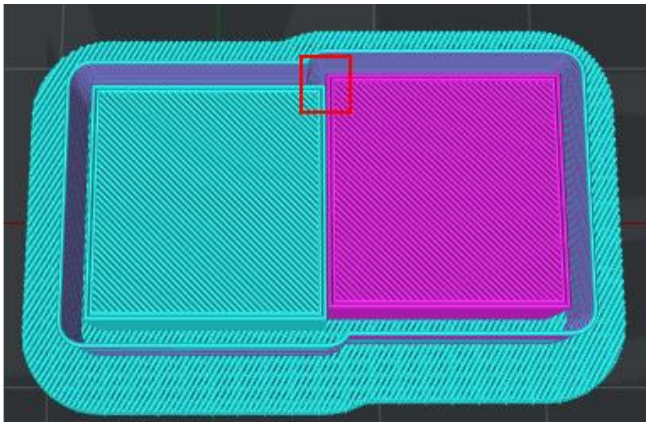


Figure 6.7: The right calibration box is backward than the left calibration box.

That means the right nozzle's position needs to be forward more to match the left nozzle. So we need to measure the value of offset and figure out the exact value which should be less than 0 then refill it into below interface.

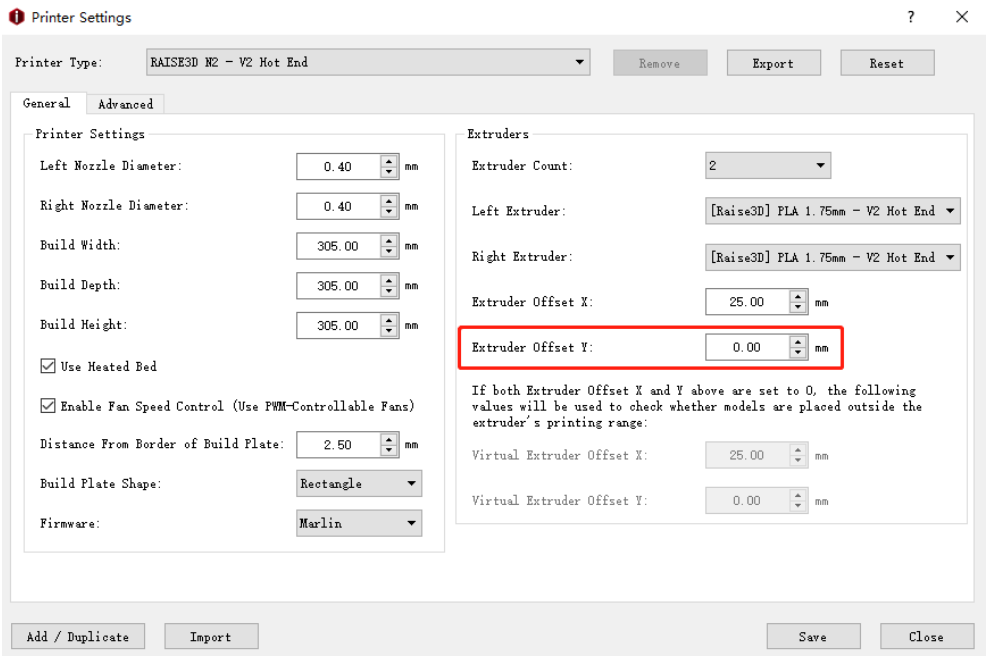


Figure 6.8: Fill the value of "Extruder Offset Y".

Fourth, the right calibration box is more forward than the left calibration box as ***Figure 6.9***:

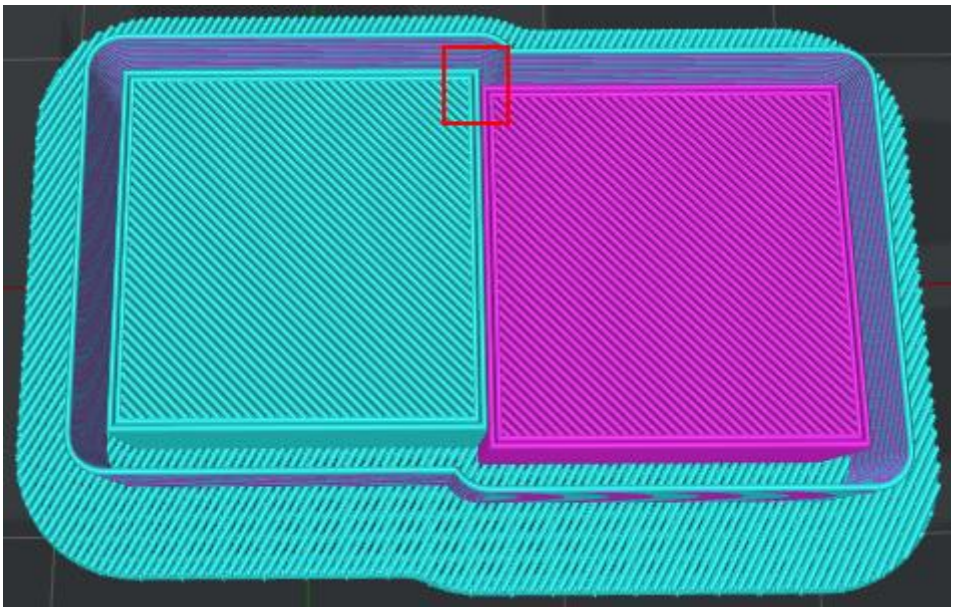


Figure 6.9: The right calibration box is more forward than the left calibration box.

That means the right nozzle's position needs to be backward more to match the left nozzle. So we need to measure the value of offset and figure out the exact value which should be more than 0 then refill it into below interface.

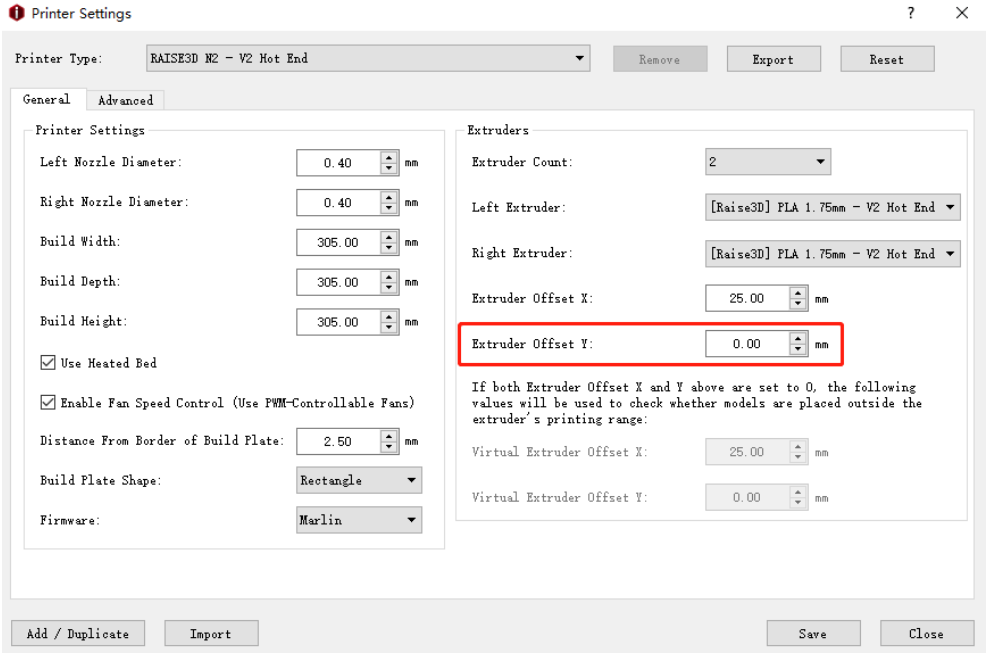


Figure 6.10: Fill the value of "Extruder Offset Y".

Please double check your model and calibrate it until the two calibration boxes reach the same X, Y position.

6.2 Advanced Settings

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 in ideaMaker before slicing.

Option 1: Enter ***Select Template*** tab and click ***Per-Model Extruder*** for your model.

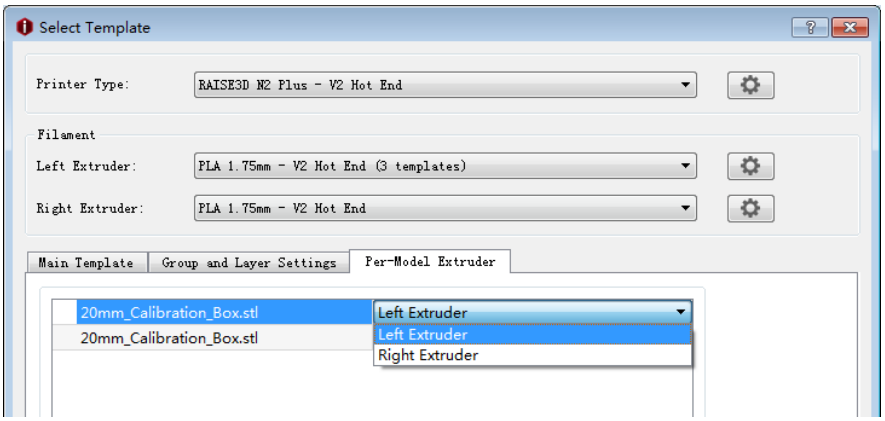


Figure 6.11: Select the different extruders to print different parts of the model.

Option 2: Click ***View*** in menu bar and click model then choose extruder for your model.

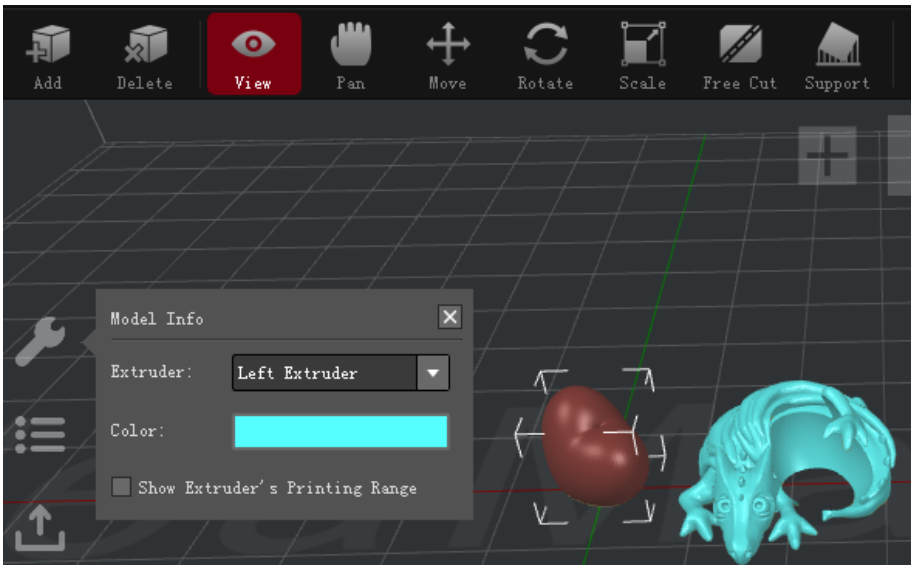


Figure 6.12: Select the different extruders to print different parts of the model.

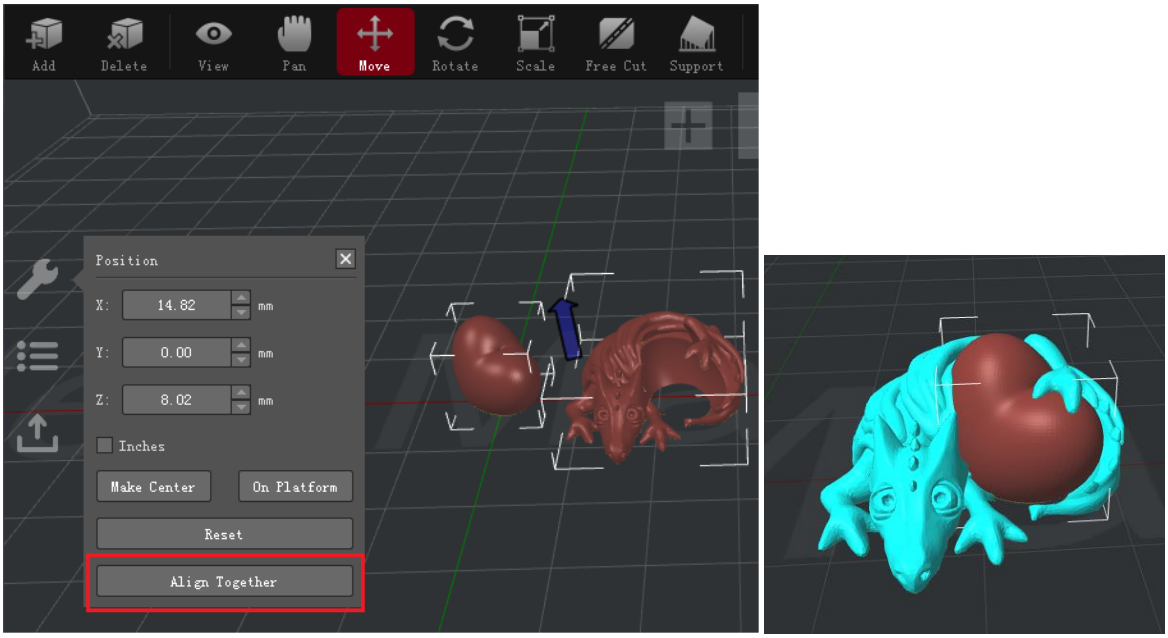


Figure 6.13: Try "Align Together" to align the models with "Move" enabled. If it doesn't work well, try to move the models manually.

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

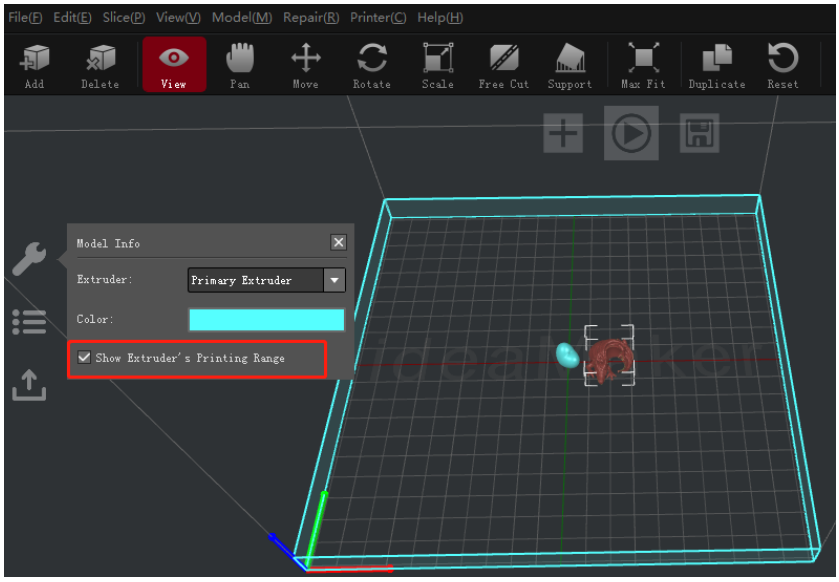


Figure 6.14: Check Right Extruder's printing range.

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

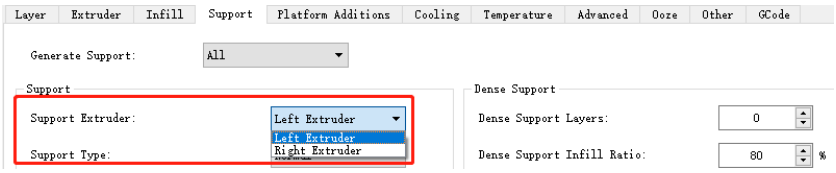


Figure 6.15: Select an extruder to print support.

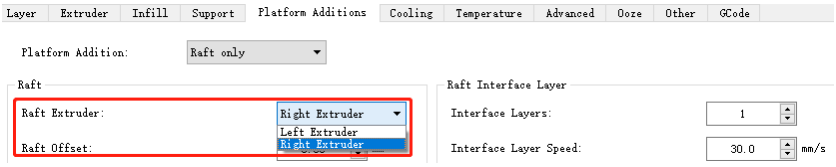


Figure 6.16: Select an extruder to print raft.

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).

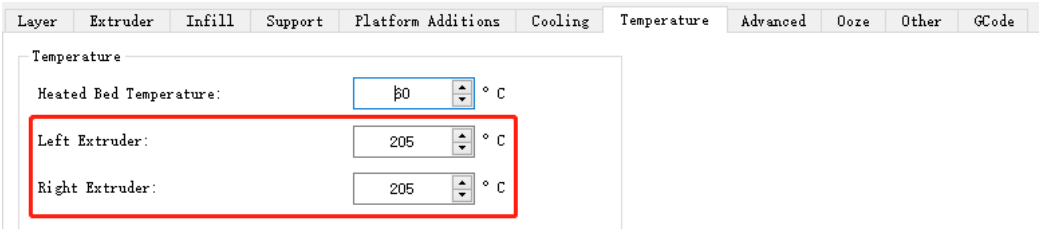


Figure 6.17: Check whether the temperature is suitable for your filament.

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

These settings as follows:

Force Retraction On Layer Change refers that the retraction will automatically be triggered the current layer print completes and the next layer starts.

Avoid Retraction Inside Model refers to disabling retraction when printing the inner structures of the model.

Extruder Switch Ooze Control refers to the retraction settings for extruder when one nozzle completes printing of one layer and the other nozzle starts printing.

Retraction Speed of Extruder-Switch refers to the speed of retraction when extruder switch.

Retraction Amount of Extruder-switch refers to the amount of filament retraction for an extruder change. 0 refers to no retraction at all.

Restart Speed of Extruder-switch refers to the extrusion speed after the retraction for an extruder change.

Extra Restart Amount of extruder-switch refers to the amount of additional filament compensation after the retraction for an extruder change.

Travel:

Avoid Traveling Through Holes refers to the printer will pass around the hole parts automatically to avoid leaving strings inside the holes.

Maximum Travel Path Length refers to that the extruder will move through the hole directly, if the travel path length of move around the hole is longer than the maximum value.

Enable Wipe Wall means when printing a model with dual-extrusion, a nozzle will print a number of extra shells around the model. This wipe wall will help to clean the nozzle while printing.

Wipe Wall Offset refers to the distance between the outline of the model and the wipe wall.

Wipe Wall Angle refers to the maximum allowed angle change which the wipe wall will generate (from 0 to 90).

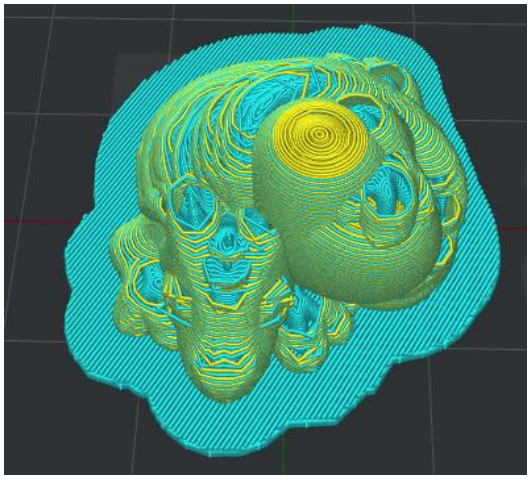


Figure 6.18: Set "Wipe Wall Angle" to be 90.

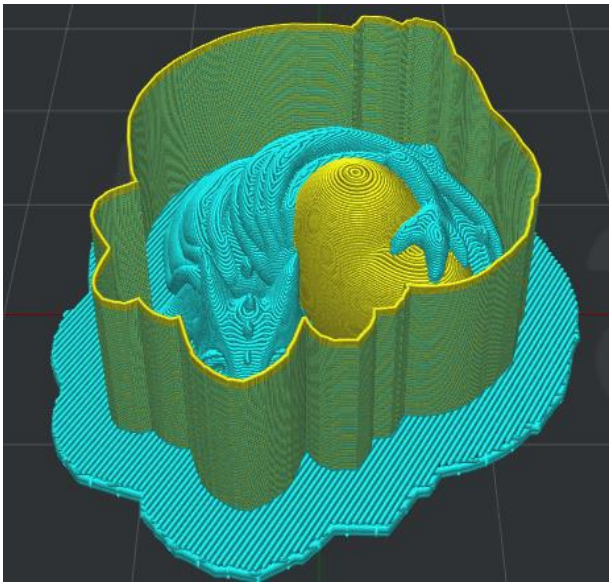


Figure 6.19: Set "Wipe Wall Angle" to be 0.

Wipe Wall Loop Lines refers to the quantity of the shells of wipe wall.

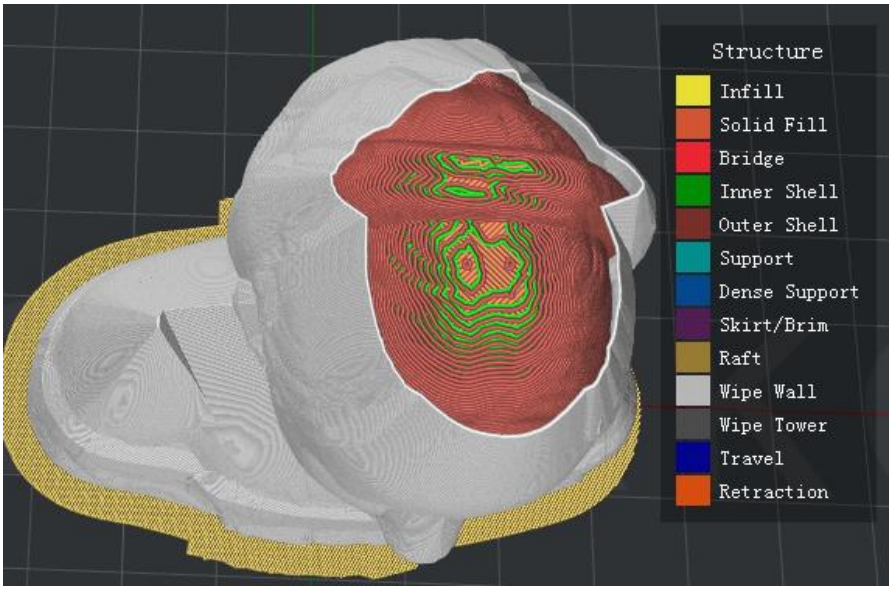


Figure 6.20: Set "Wipe Wall Loop Lines" to be one.

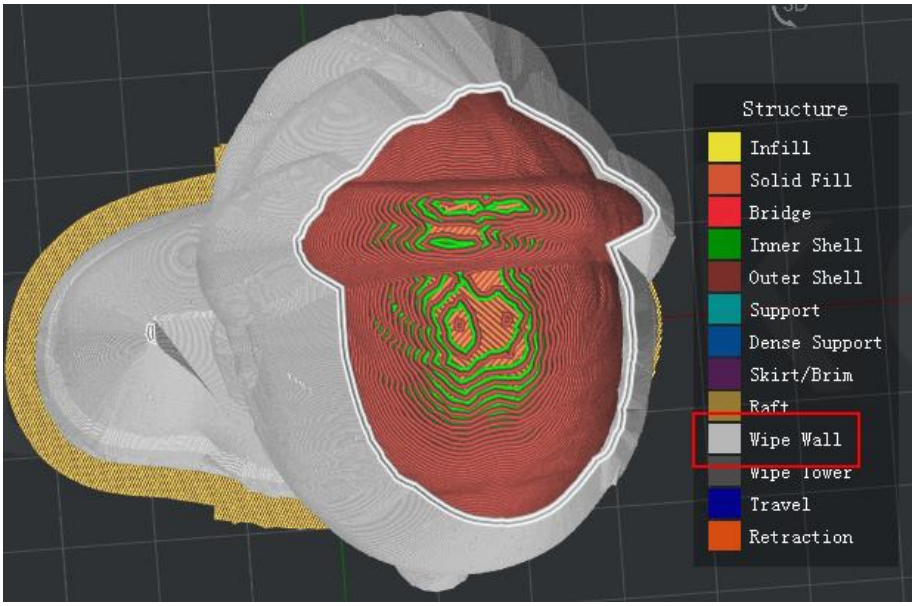


Figure 6.21: Set "Wipe Wall Loop Lines" to be two.

Wipe Wall Type refers to the shape of the wipe wall shells. The primary difference is the distance of model and wipe wall.

Contoured is the one which is closest to the origin model shape. ***Water fall*** is the type which will follow the origin shape, but a vertical drop if the lower structure is narrow that the above one. ***Vertical*** will find the widest part of the model and generate a vertical type wipe wall.

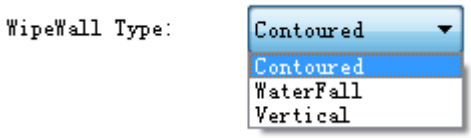


Figure 6.22: Three types of Wipe Wall.

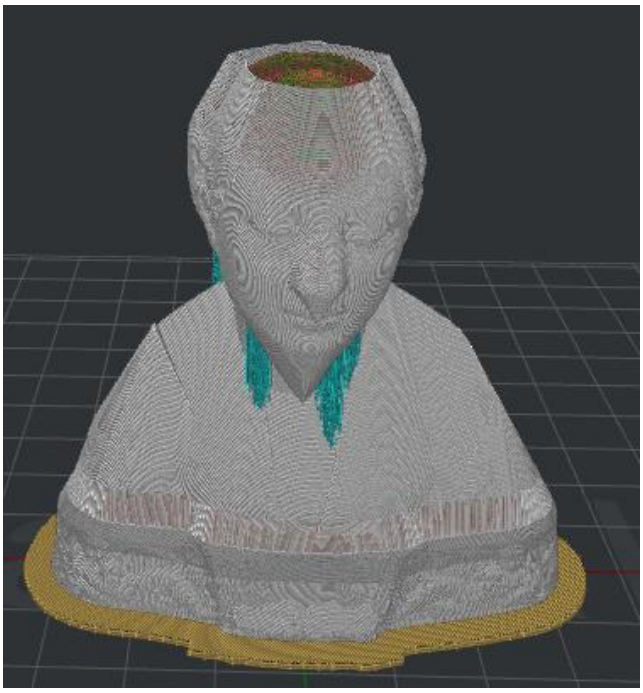


Figure 6.23: Set "Wipe Wall Type" to be "Contoured".

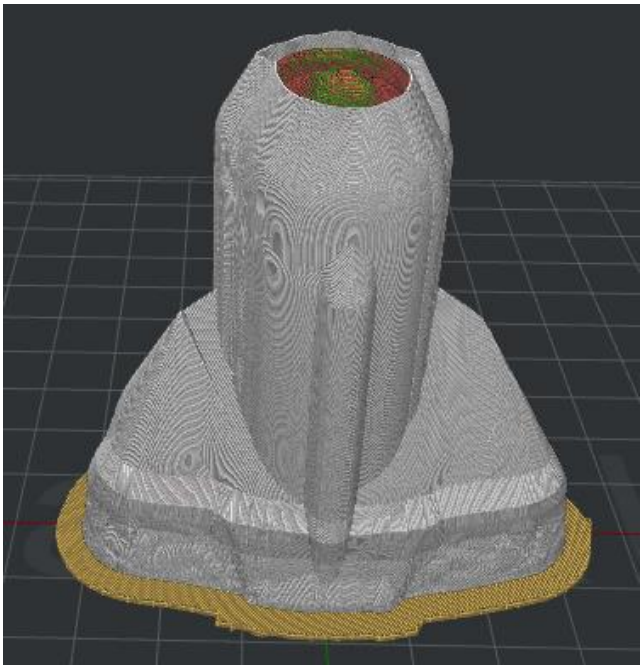


Figure 6.24: Set "Wipe Wall Type" to be "WaterFall".

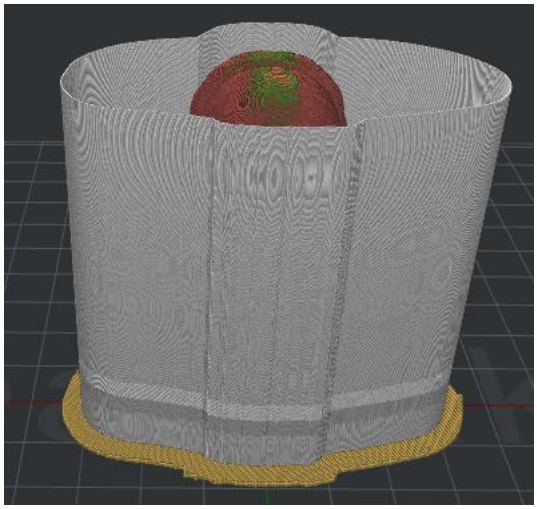


Figure 6.25: Set "Wipe Wall Type" to be "Vertical".

Wipe Wall Speed refers to the printing speed for wipe wall.

Wipe Tower refers to a cube created outside of the model which is used for wiping the nozzle which is going to start print.

Wipe Tower Width refers to width of the wipe tower.

Wipe Tower Infill Ratio refers to the density of wipe tower structure.

Wipe Tower Speed refers to the printing speed for wipe tower.

Placed at the Fixed Position of the plate refers to with this function enabled Wipe Tower's position would be fixed as ***Figure 6.26.***

Wipe Tower Fixed Position X refers to the position in X direction.

Wipe Tower Fixed Position Y refers to the position in X direction as ***Figure 6.27***.

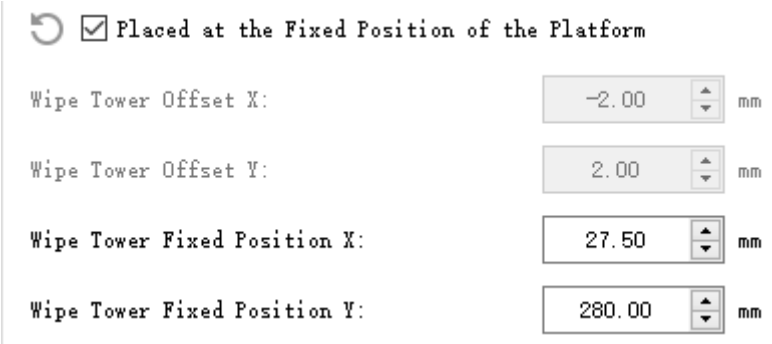


Figure 6.26: Wipe Tower Fixed Position.

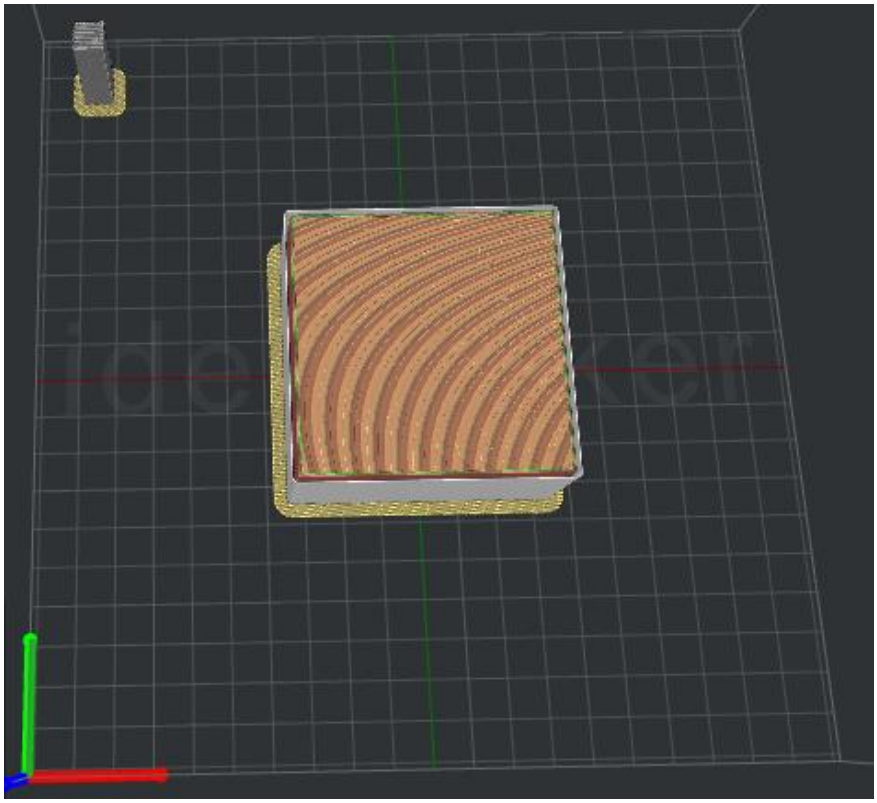


Figure 6.27: Enable the function of Placed at the Fixed of the Platform.

Wipe Tower Offset X refers to the distance between Wipe Tower and the model in the direction of X.

Wipe Tower Offset Y refers to the distance between Wipe Tower and the model in the direction of Y.

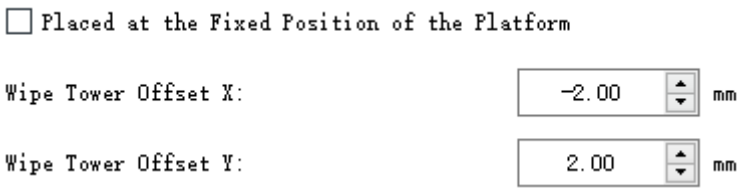


Figure 6.28: Wipe Tower X/Y offset.

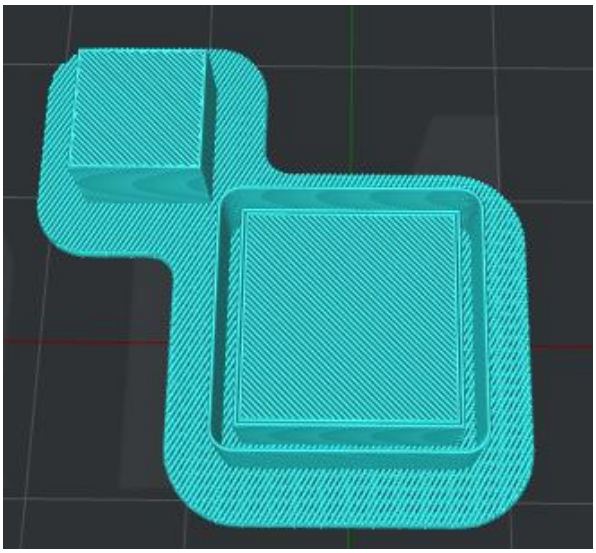


Figure 6.29: Wipe Tower.

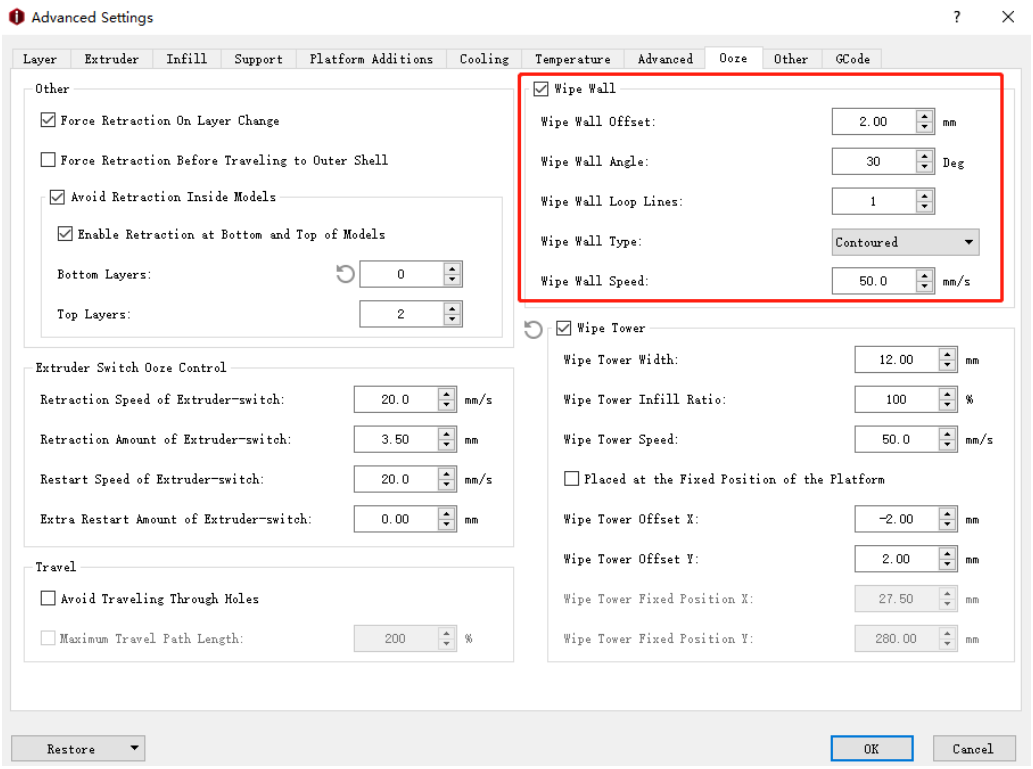


Figure 6.30: Wipe Wall will help decrease the effect of ooze from the other nozzle. But it is hard to totally avoid the ooze.

7 Filament Settings

7.1 Edit Filament Settings

When you want to edit the parameters of your filament, check the steps below:

Option 1: Edit the existing template.

Printer -> Filament Settings

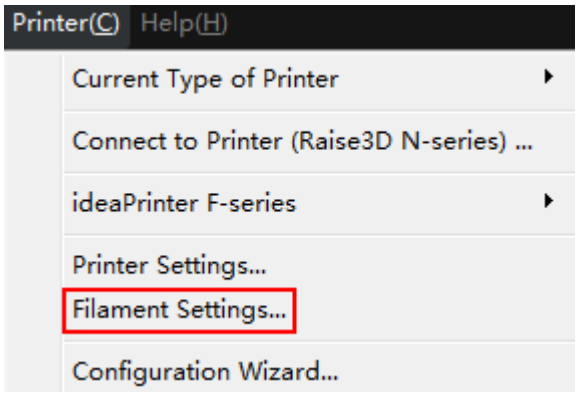


Figure 7.1: Select "Printer" ->"Filament Settings" from menu bar.

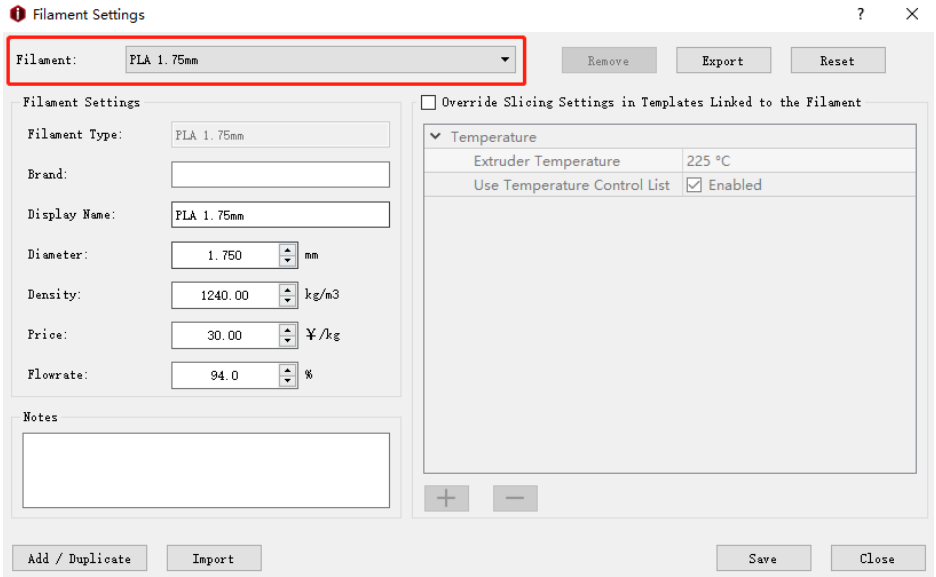


Figure 7.2: Select a filament from the drop list and edit its parameters.

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

Remove refers to deleting the current filament template.

Note: remove function only work for the template which was added manually, the default template cannot be removed.

Export refers to exporting filament template from ideaMaker to your computer.

Reset refers to resetting parameters to default settings.

Note: reset function only work for default template.

Import refers to importing a filament template from your computer.

Save refers to saving the current filament template.

Close refers to closing the current filament template.

Override Slicing Settings in Template Linked to the Filament refers that with this function enabled, below setting will be used to override setting in slicing template which is linked to current filament.

Press ***"+"*** to add more settings as ***Figure 7.3***.

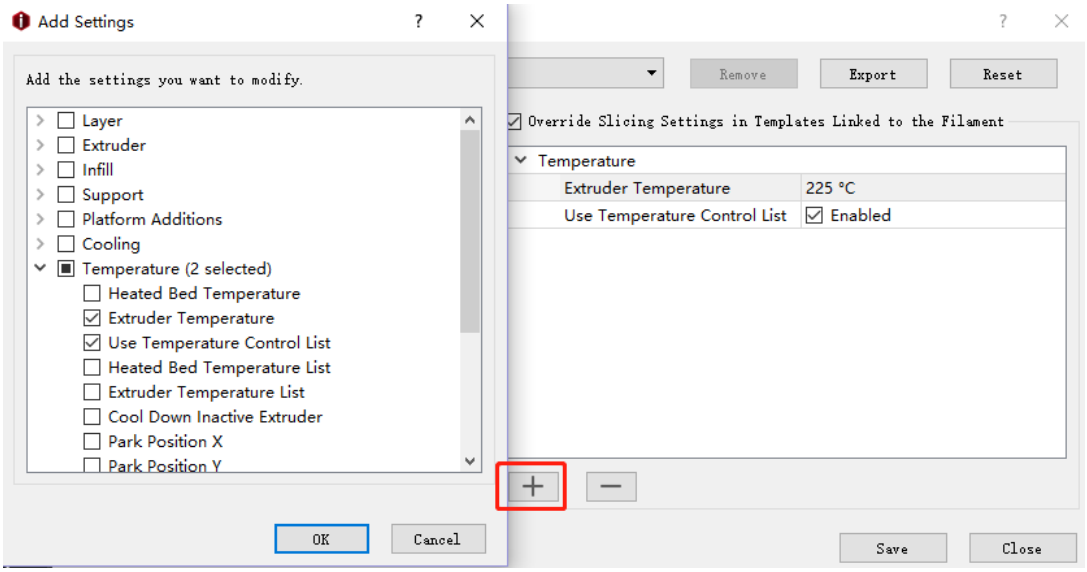


Figure 7.3: Override Slicing Settings.

Note: The settings which are override in filament can be edited by press **"**Edit**"** button as ***Figure 7.4***.

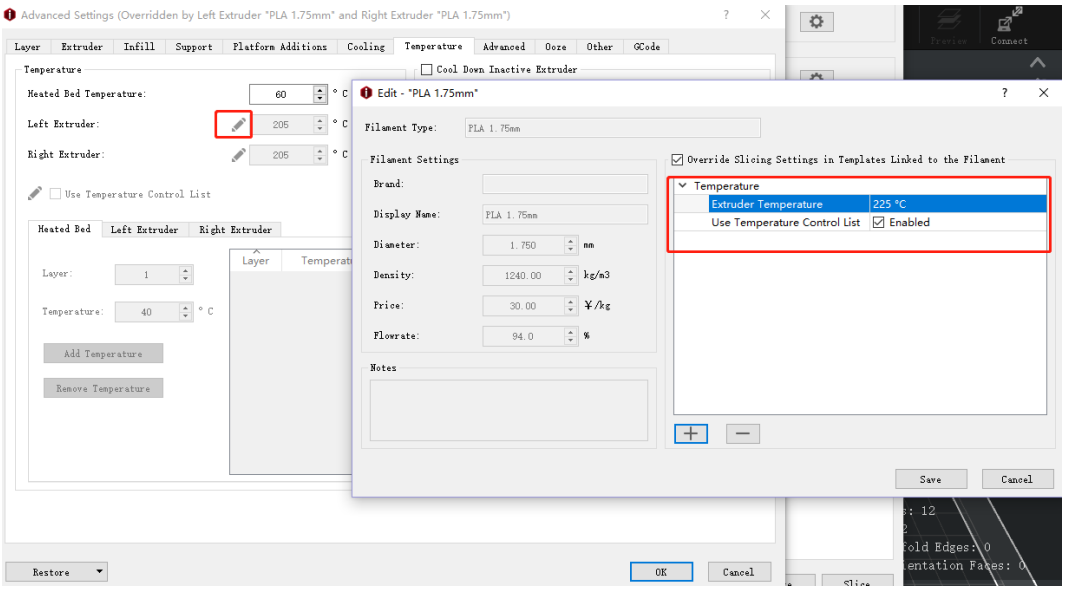


Figure 7.4: Edit the override settings in Filament Settings.

Option 2: Enter ***Select template*** tab and click setting button.

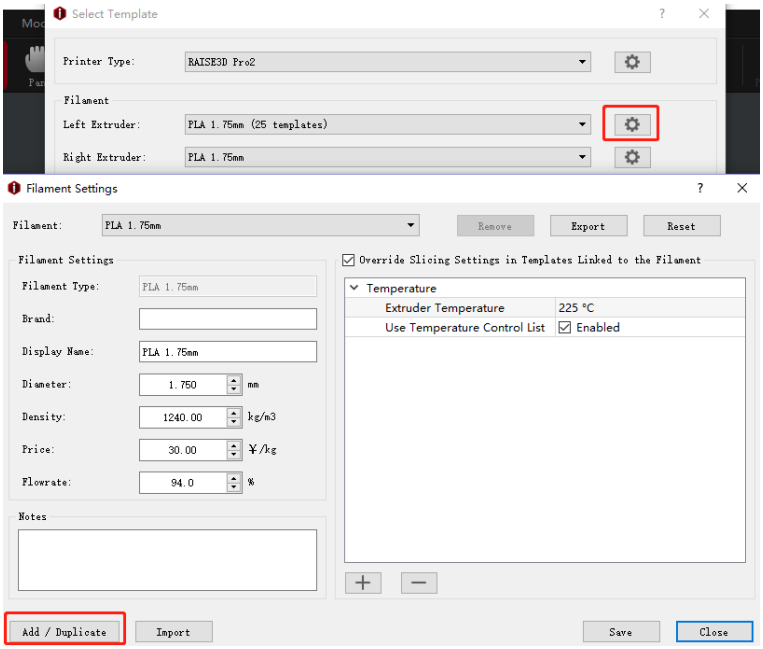


Figure 7.5: Clicking "setting button" and click "Add" button.

7.2 Add/Duplicate Filament

Press ***Add/Duplicate*** button to add a new Filament Settings

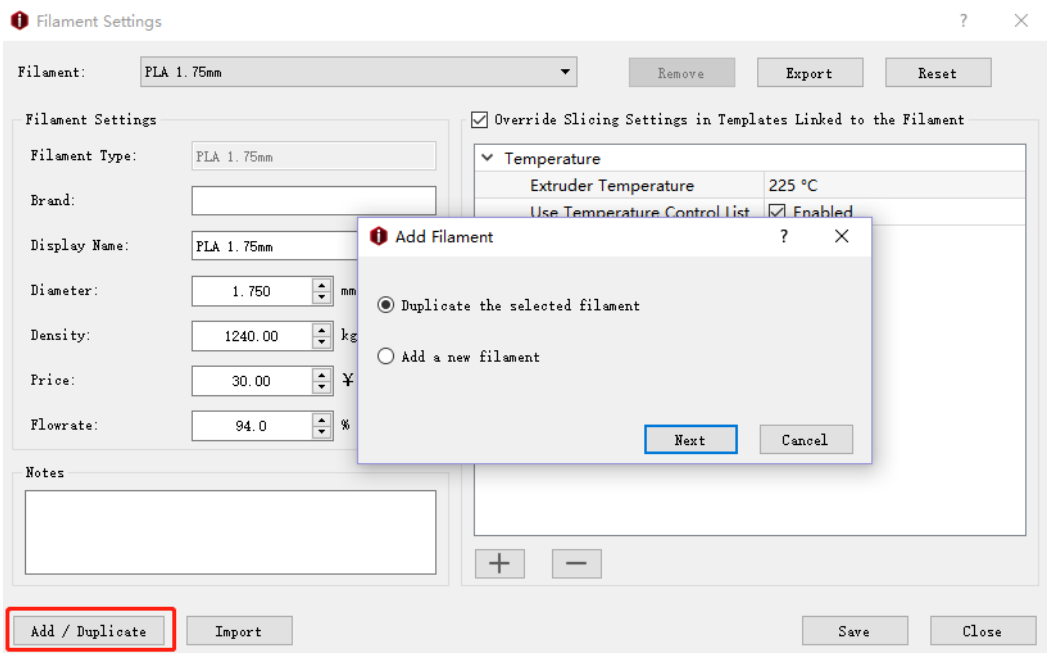


Figure 7.6: Add/Duplicate Filament template.

If you want to build a new filament, select ***Add/Duplicate Filament*** in last page. ***Duplicate the selected filament*** refers to duplicating the current filament template as ***Figure 7.7***.

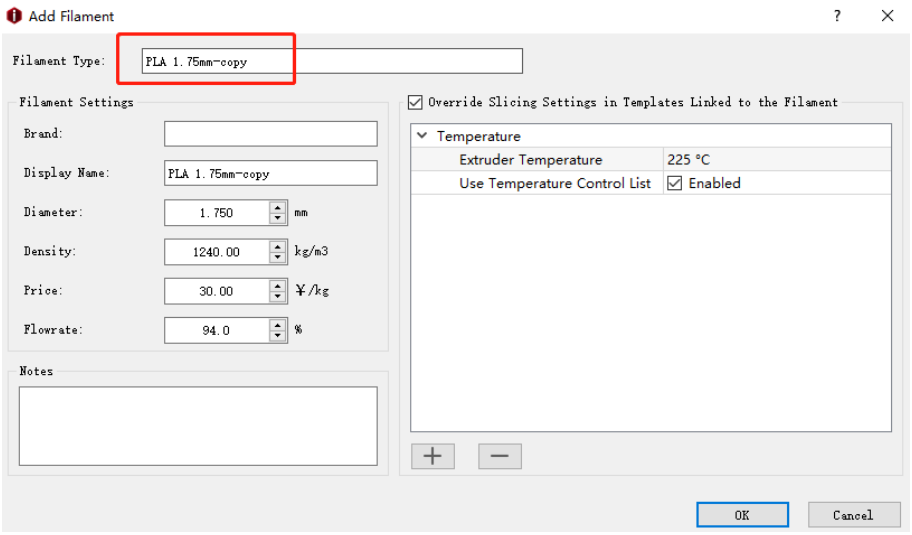


Figure 7.7: Duplicate the selected filament.

Add a new filament refers to creating a new filament template as ***Figure 7.8***.

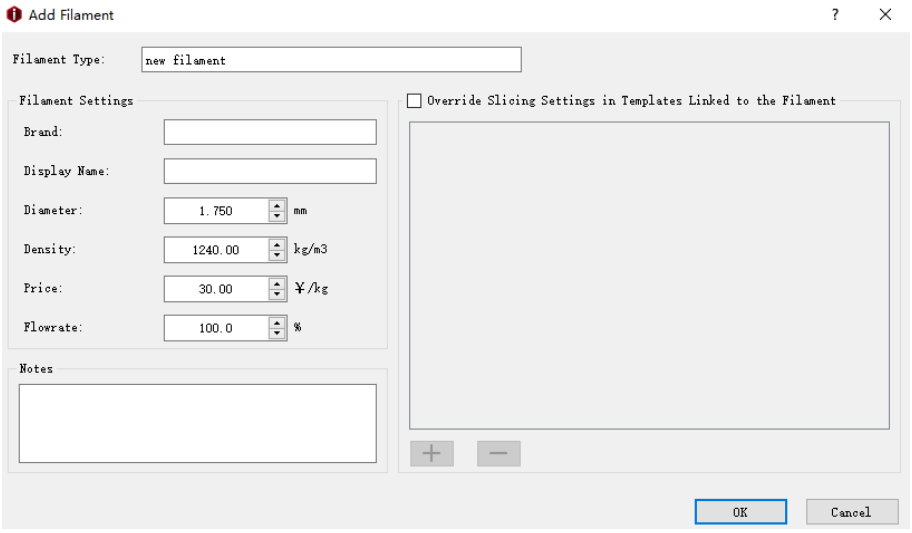


Figure 7.8: Add a new filament.

Here we have a little tip, in most cases filament doesn't need flowrate compensation. But PLA and flexible materials need to have flowrate compensation. PLA material has a default setting of flowrate at 94%. Print other materials with a PLA setting G-Code file may cause wrong extrusion.

7.3 Remove Filament

In this version, the customer's filament template can also be removed. Choose the filament template you want to remove from the Filament Type drop list and select Remove button to remove it.

Note: This function only works for the template which was added by duplicate.

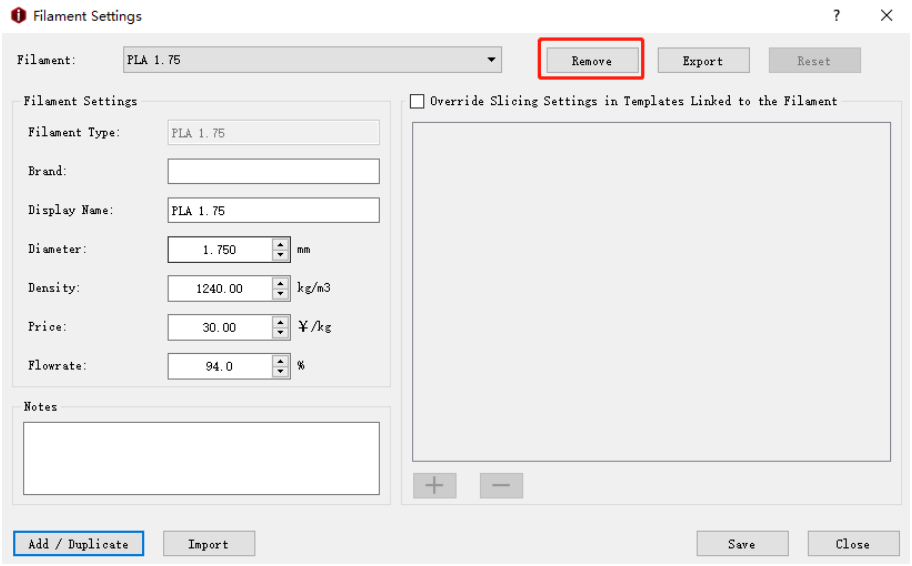


Figure 7.7: Remove the selected filament settings.

8 Printer Settings

Distance From Border of Build Plate refers to keeping a safe distance from the border of the build plate as ***Figure 8.1***.

Note: This function is to check if models are placed outside of the extruder's printing range.

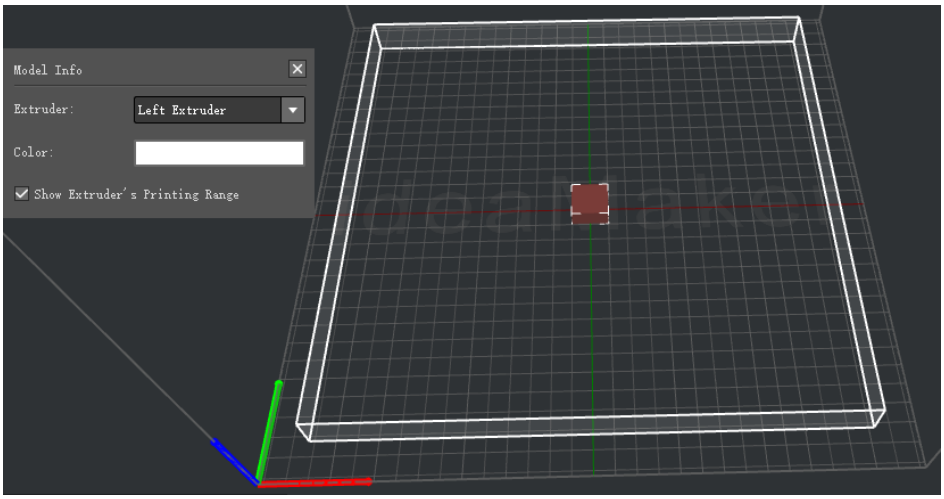


Figure 8.1: Set Distance From Border of Build Plate 20 mm.

Build Plate Shape refers to the shape of build plate.

Firmware refers to Raise3d speedy.

Virtual Extruder Offset X refers to the right extruder's offset in X axis.

Virtual Extruder Offset Y refers to the right extruder's offset in Y axis.

Note: This function is for Pro 2 series printer to check if models are placed outside of the right extruder's printing range.

For example, set Virtual Extruder Offset X to be 0 mm, then choose printer extruder to be right extruder, you will see the printer range is same as left extruder as ***Figure 8.2***.

Set Virtual Extruder Offset X to be 25 mm, then choose printer extruder to be right extruder, you will see the printer range as ***Figure 8.3***.

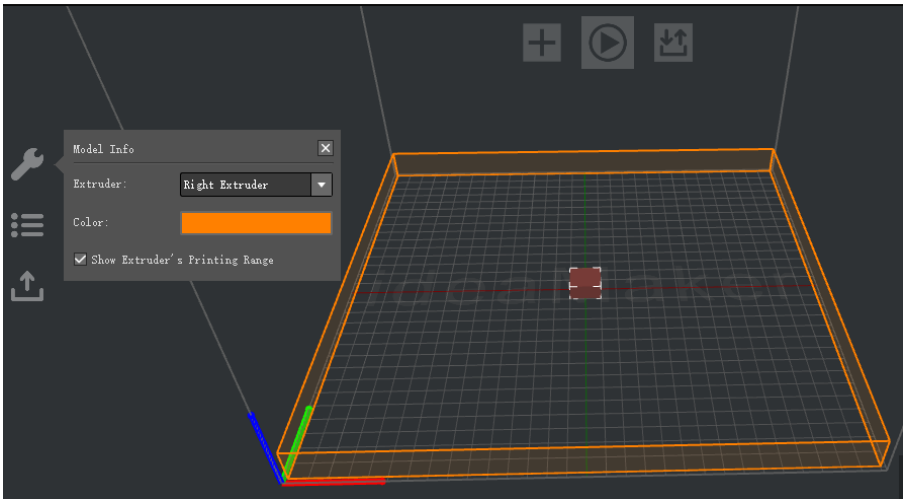


Figure 8.2: Set Virtual Extruder Offset X to be 0 mm.

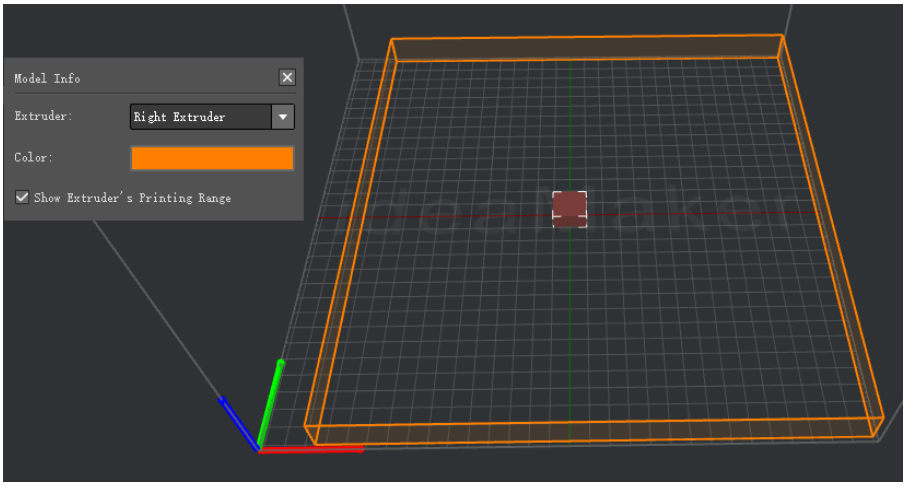


Figure 8.3: Set Virtual Extruder Offset Y to be 25 mm.

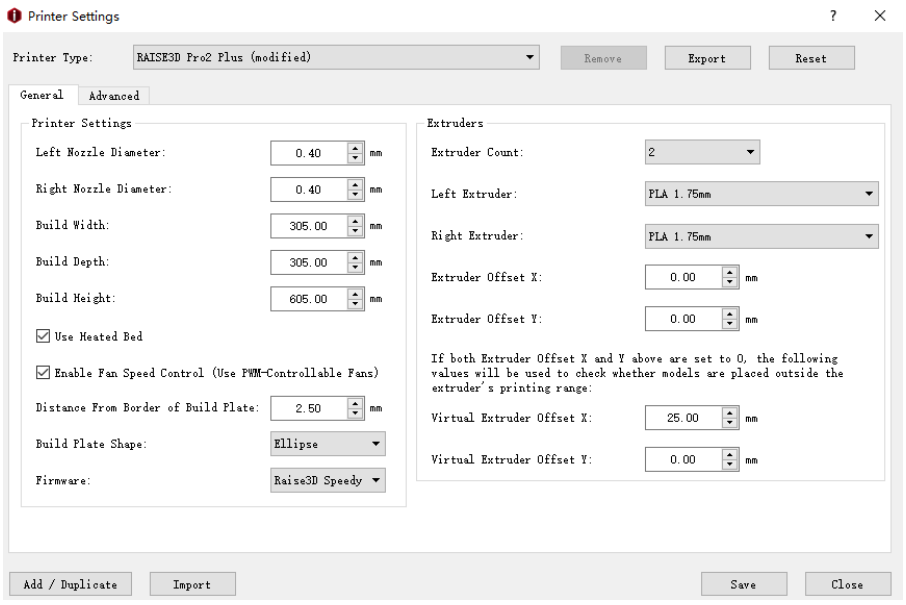


Figure 8.3: Printing Settings.

Custom Command

Omit RaiseTouch Specified Gcode refers that with this function enabled, ideaMaker will omit outputting M1001, M1002 and M2000 for 3rd party printers.

M1001 refers to the End of Start Gcode.

M1002 refers to the Beginning of End Gcode.

M2000 refers to the Pause at Height Gcode.

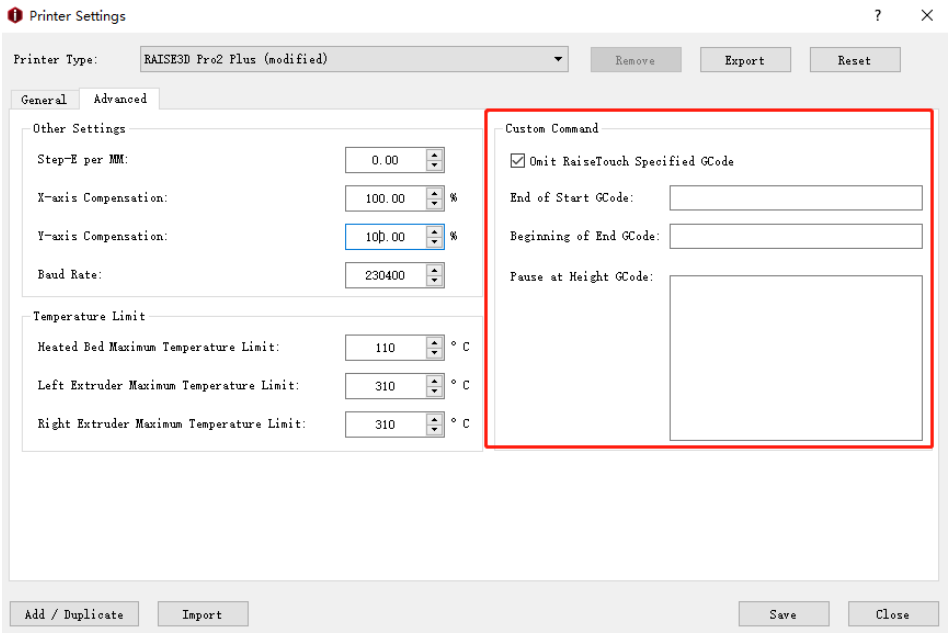


Figure 8.4: Custom Command.

8.1 Edit Printer Settings

Option1: Edit the existing template in ***Printer->Printer Settings***

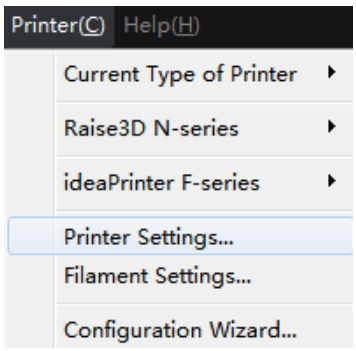


Figure 8.5: Select "Printer" ->"Printer Settings" from menu bar.

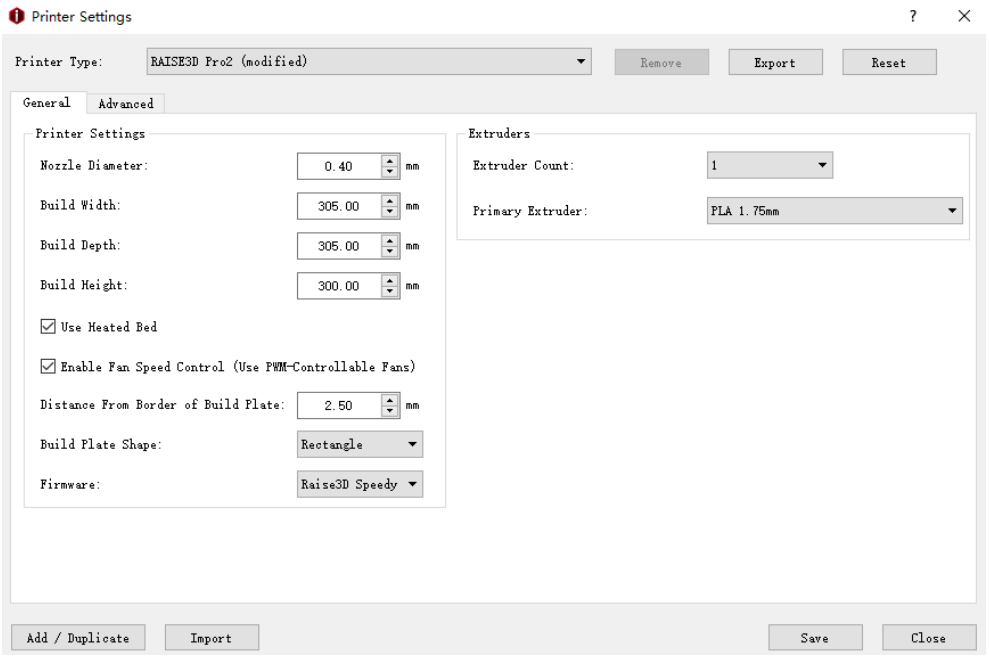


Figure 8.6: Select a printer from the drop list and edit its parameters.

Option2: Enter Select template tab and click setting button.

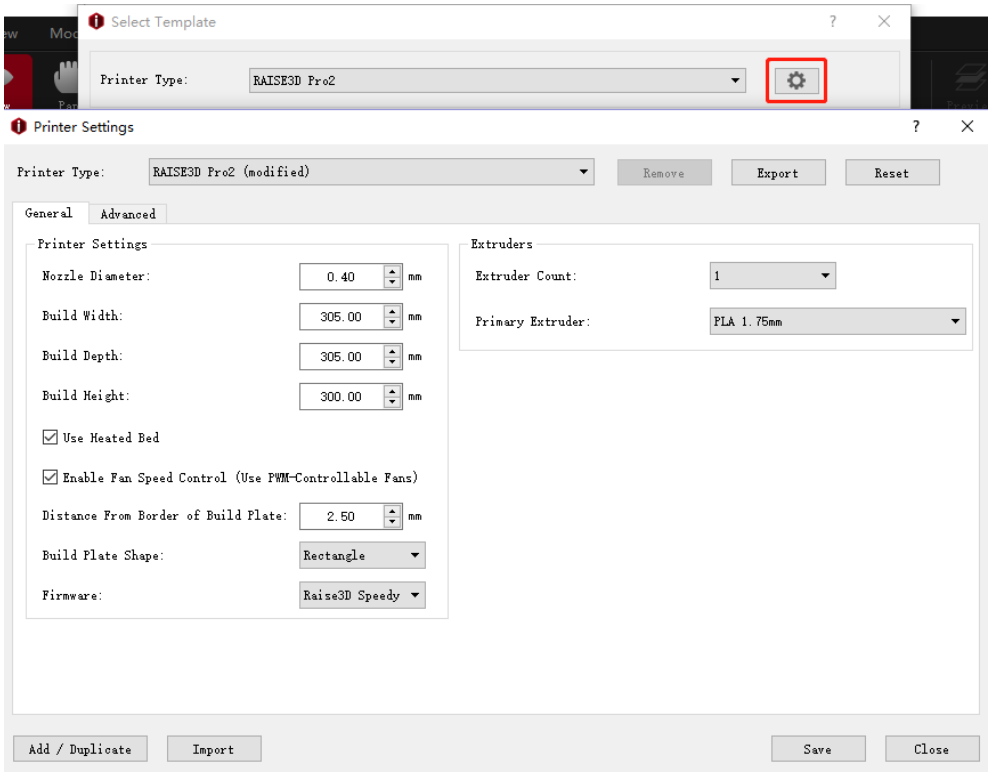


Figure 8.7: Clicking "Setting" button and click "Add" button.

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

8.2 Add/Duplicate Printer

Option 1: Add printer in ***Printer->Printer Settings***

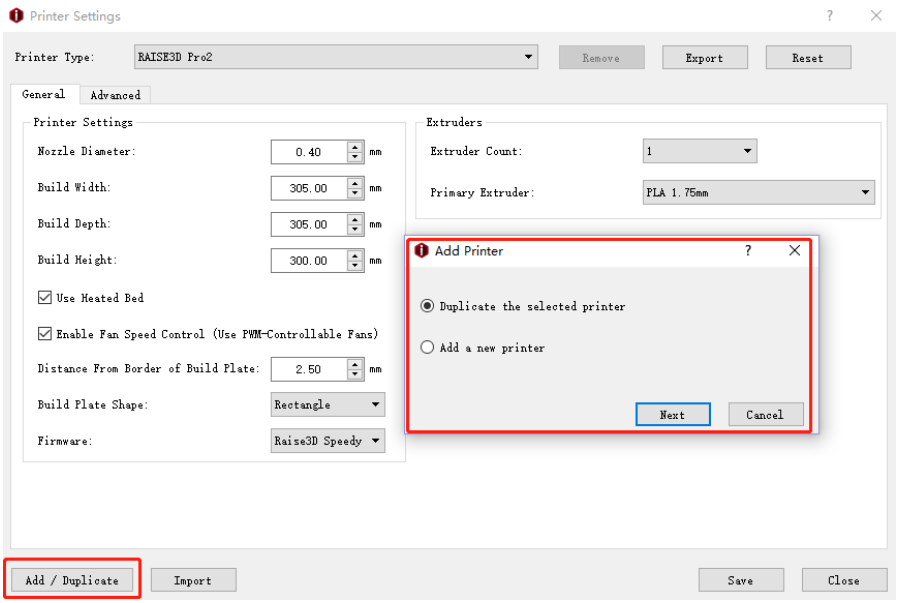


Figure 8.8: Add/Duplicate Printer.

Duplicate the selected printer refers to creating a duplicate object of selected Printer Settings as ***Figure 8.5***.

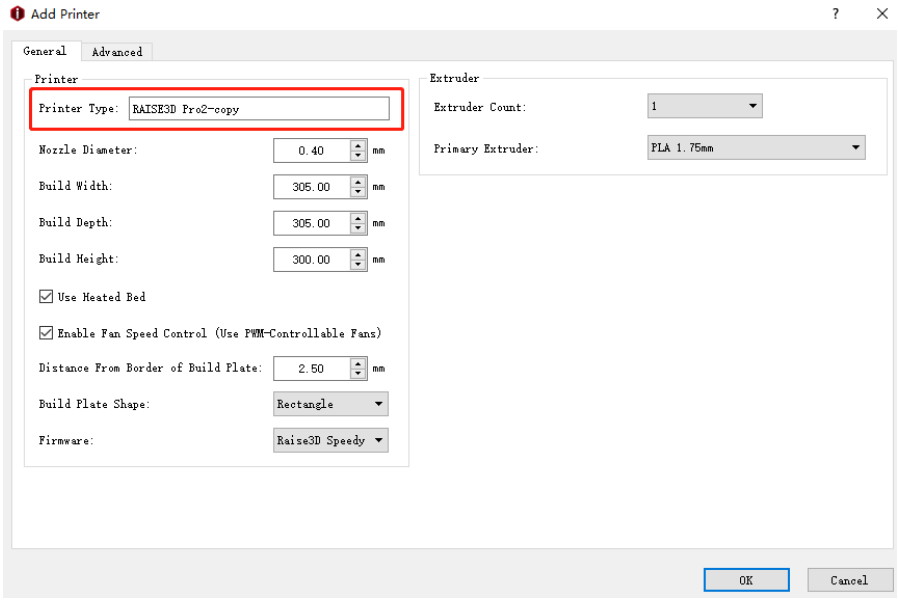


Figure 8.9: Duplicate the selected printer.

Add a new printer refers to creating a new printer template as ***Figure 8.10***.

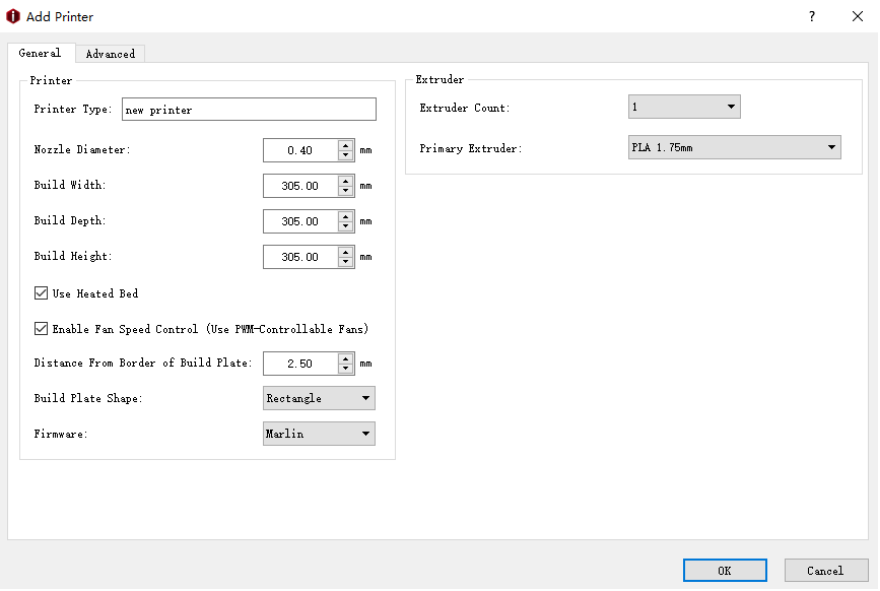


Figure 8.10: Add a new printer.

Option2: Enter into Select template tab and click setting button.

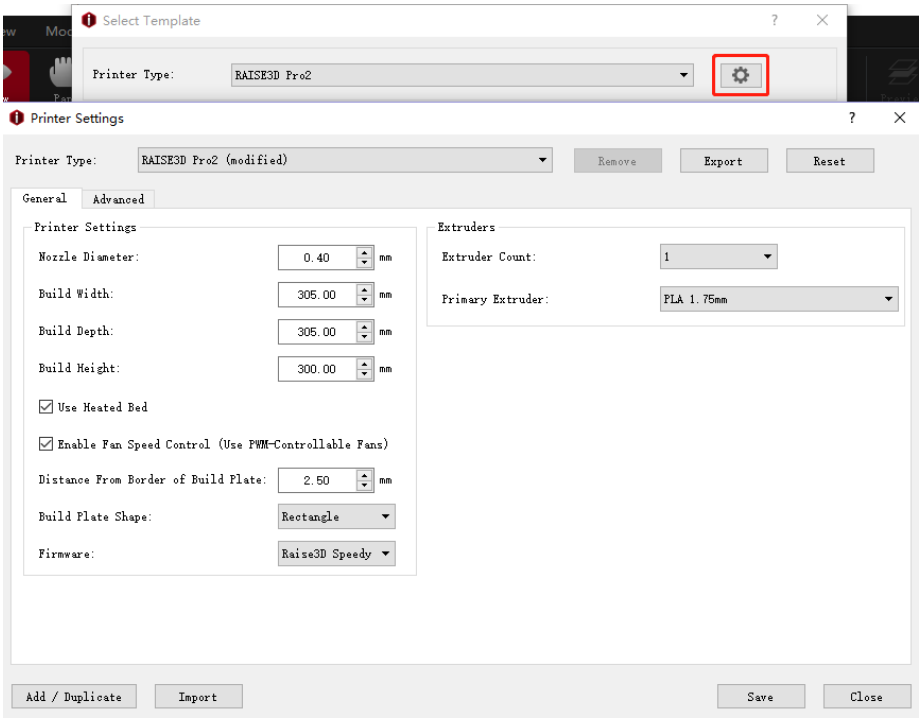


Figure 8.11: Add new printer.

8.3 Import Printer

Export refers to exporting printer template from ideaMaker to your computer.

Reset refers to printer parameters to default settings.

Note: reset function only work for default template.

Import refers to importing a printer template from your computer.

Save refers to saving the current printer template.

Close refers to closing the current printer template.

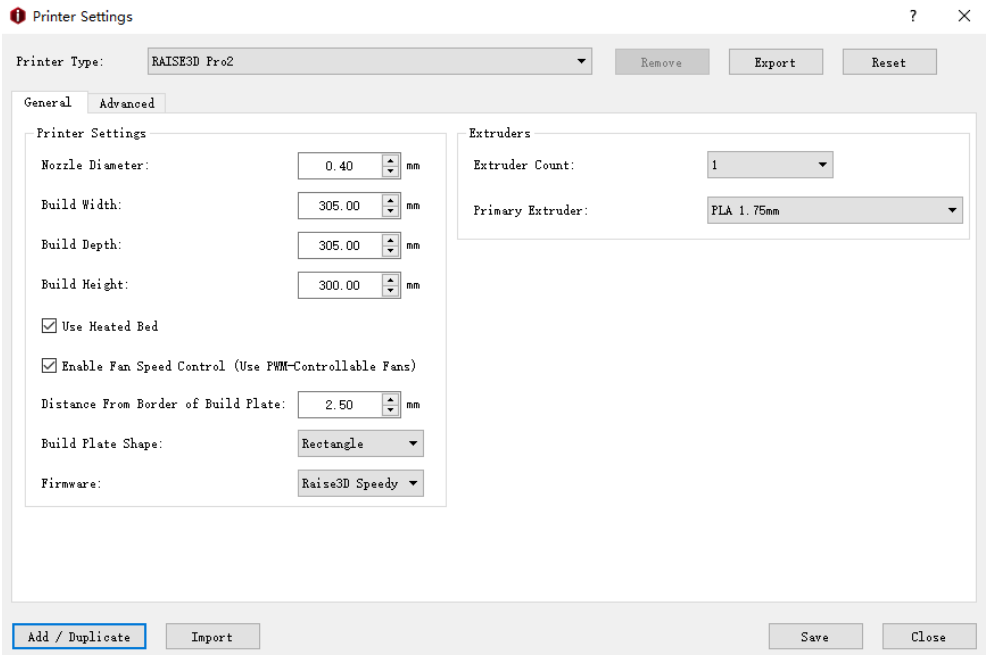


Figure 8.12: Printer Settings.

8.4 Remove Printer

Remove refers to deleting selected ***Printer Settings***. Choose the printer template you want to remove from the Printer Type drop list and click ***Remove*** button.

Note: This function only works for the template which was added by duplicate.

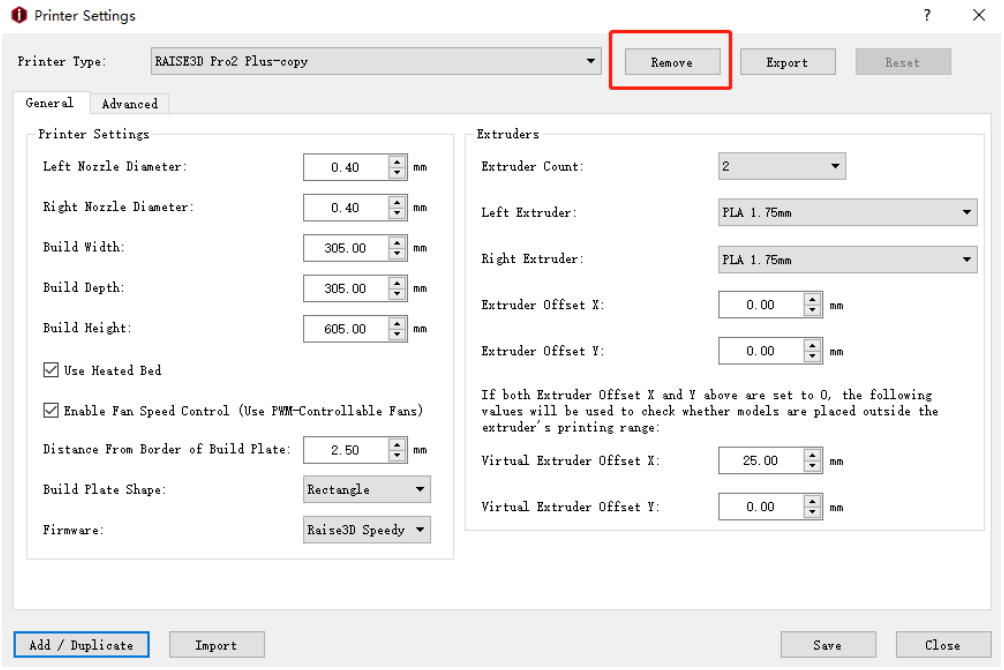


Figure 8.13: Remove the selected Printer Settings.

If you want to remove the slicing templates under this printer as a group, then check the ones you want to remove from the list on the left side. Then select ***"OK"***. The selected slicing templates and the printer templates will be removed all together.

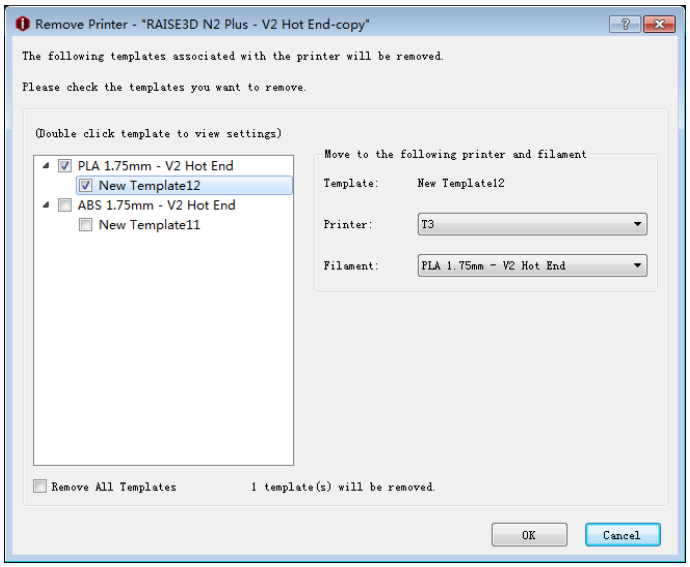


Figure 8.14: Remove the slicing templates.

If you do not want to keep any slicing template under this printer type, check ***Remove All templates*** below. It will check all the slicing templates above. Then click ***"OK"*** button. All the slicing templates under this printer type will be removed together with this printer template.

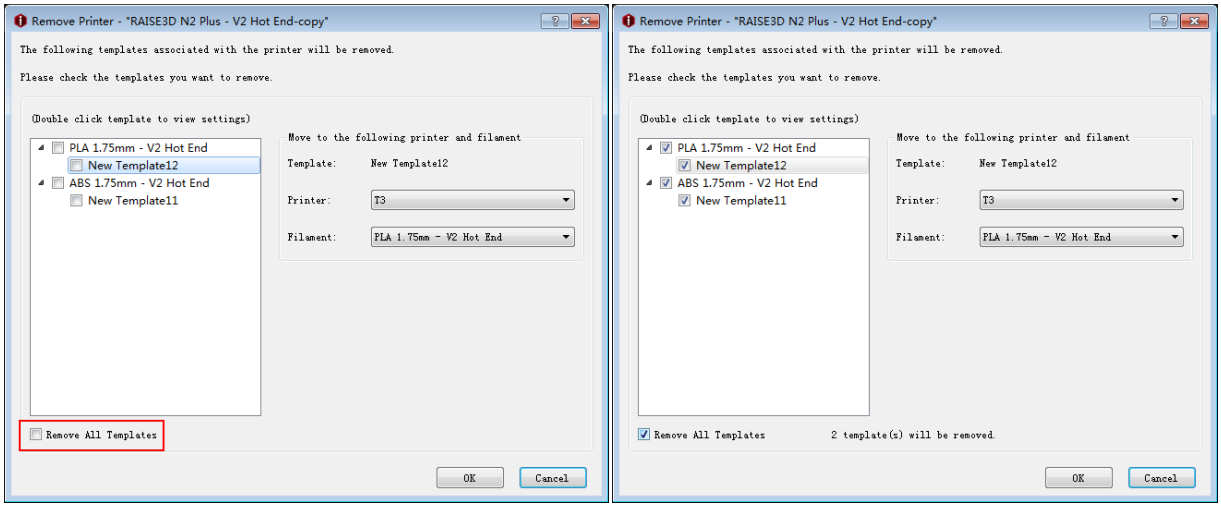


Figure 8.15: Remove All Templates.

If you want to keep one slicing template, please first make sure that it has been unchecked.

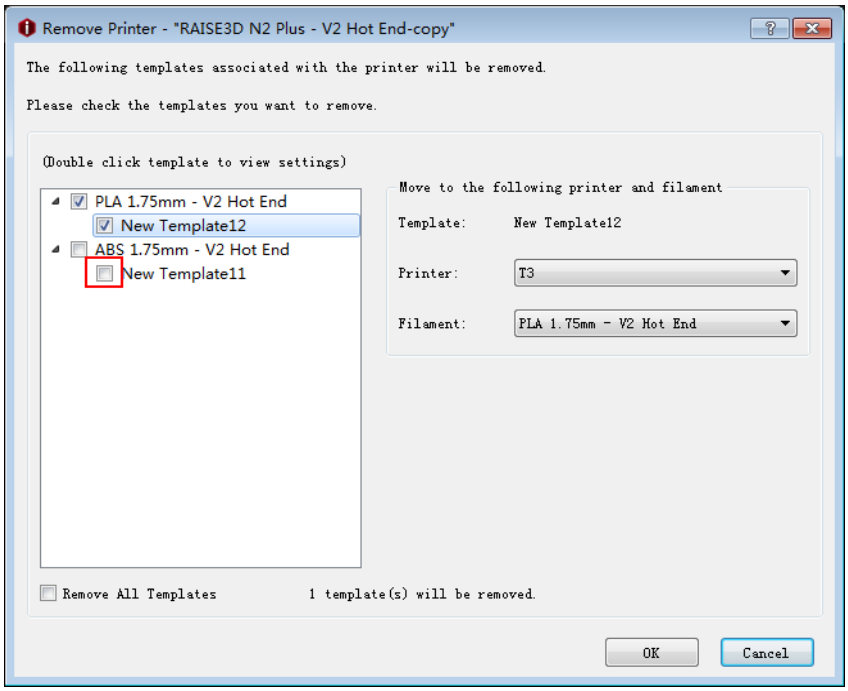


Figure 8.16: Remove the selected Templates.

Select it and choose the ***Printer*** and ***Filament*** at right side for it. Then press ***"OK"*** button below. The printer template will be removed, but this slicing template will be saved under ***"T3 – PLA 1.75mm-V2 Hot End"*** in this case.

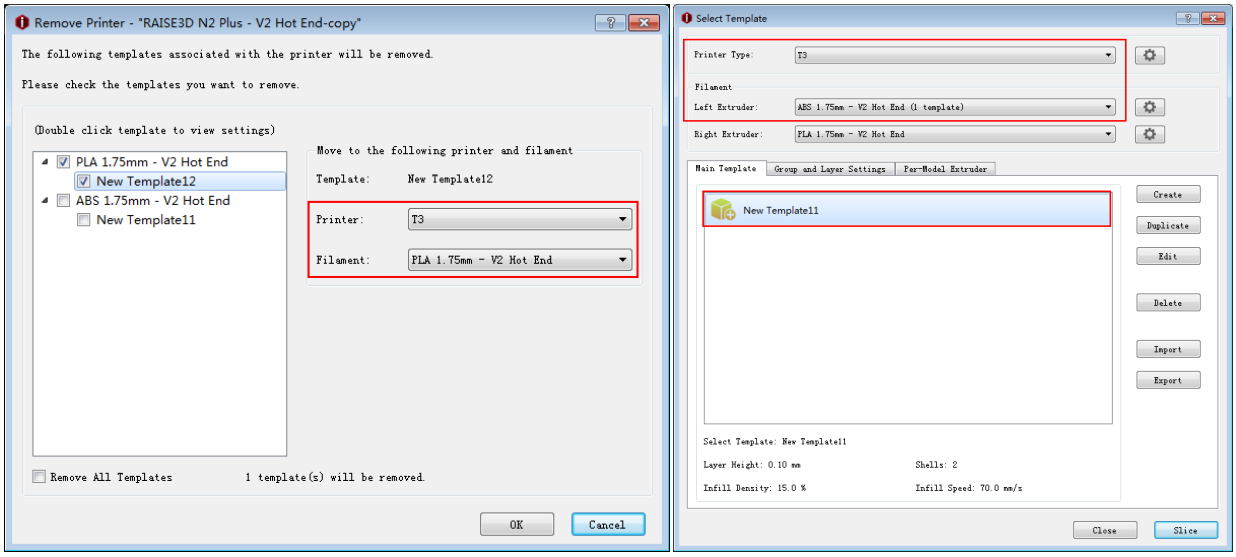


Figure 8.17: Remove the selected Templates.

RAISE3D

www.raise3d.com

Office: 43 Tesla, Irvine, California 92618
Tel: +1-888-963-9028
China Office: Floor 4 B5, 1688 North Guoquan Road, Yangpu District, Shanghai, 200000
Tel: +86 21 65337855