ideaMaker Manual

RAISE3D

www.raise3d.com

Using ideaMaker

Basic information

What is ideaMaker?

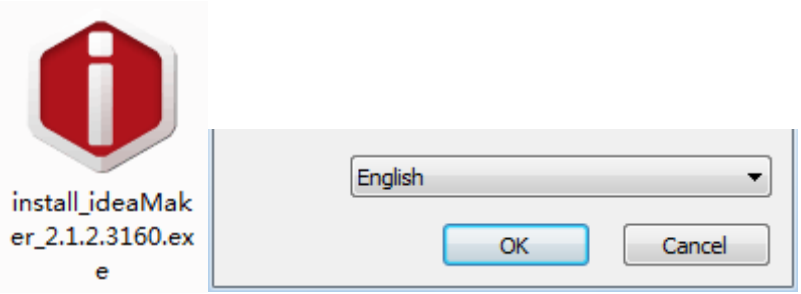
ideaMaker is slicing software that prepares 3D models for printing and turns them into .gcode file for your Raise3D printer.

Where to download ideaMaker?

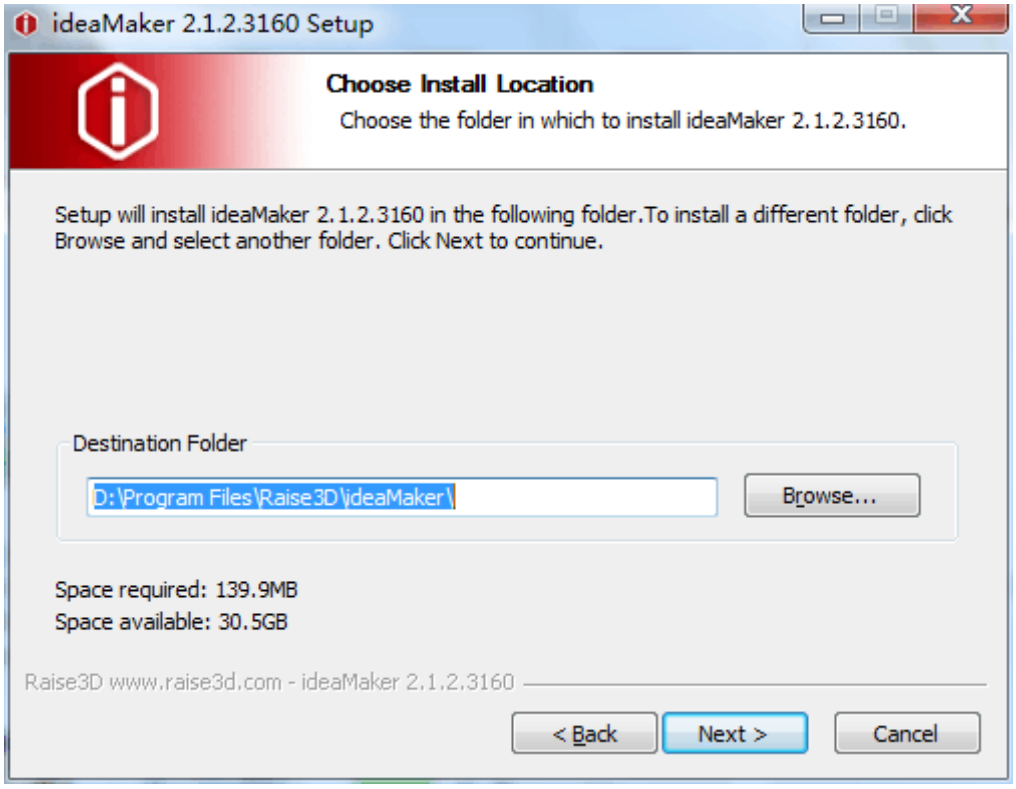
ideaMaker is available in the USB storage included in the accessory box with your printer. Or you can get the latest version on our website at: http://www.raise3d.com/pages/download

Install ideaMaker

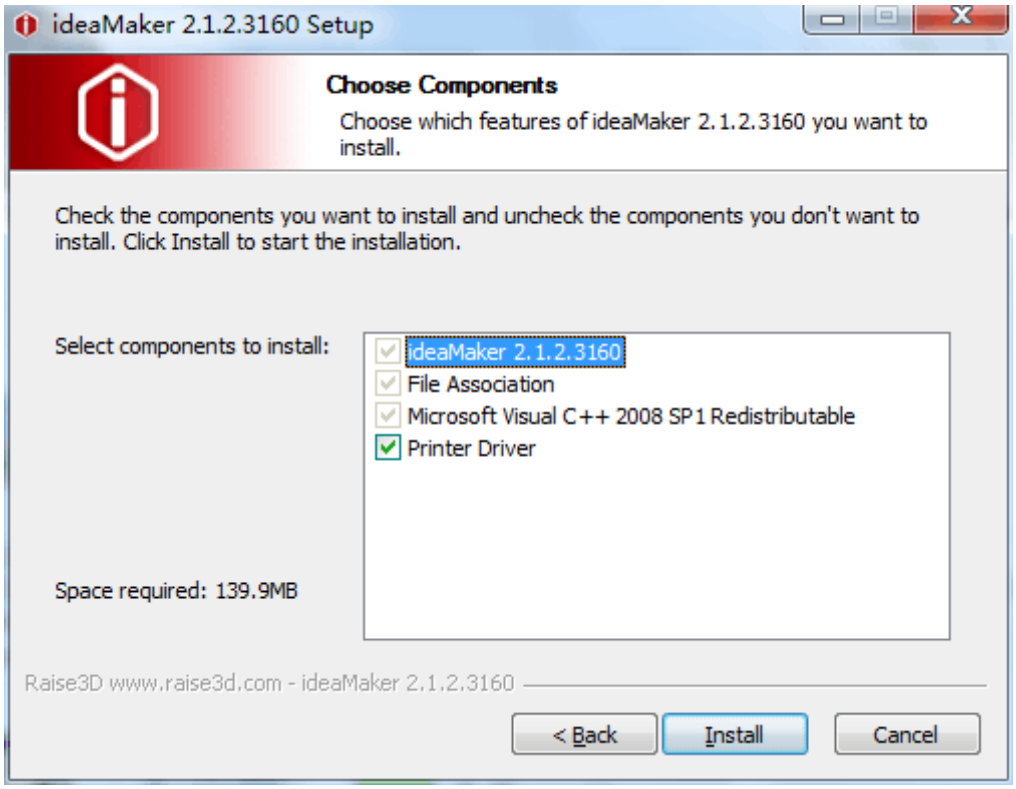
1. Open the installer and choose your language preference. Then click ***Next*** to move on to the next menu. If you are using ideaMaker before your installation, you need to close ideaMaker. So that the new version can be installed without any error.



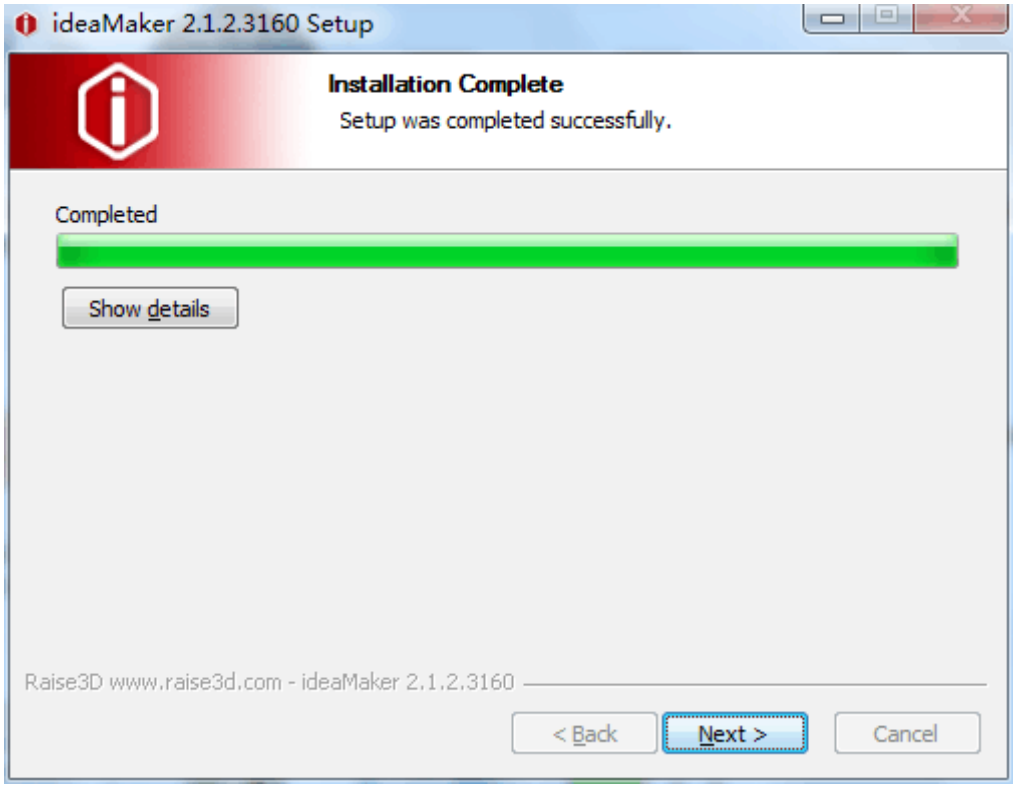
2. Select a path to install ideaMaker and then click ***Next*** to move on to the next menu.



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.



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

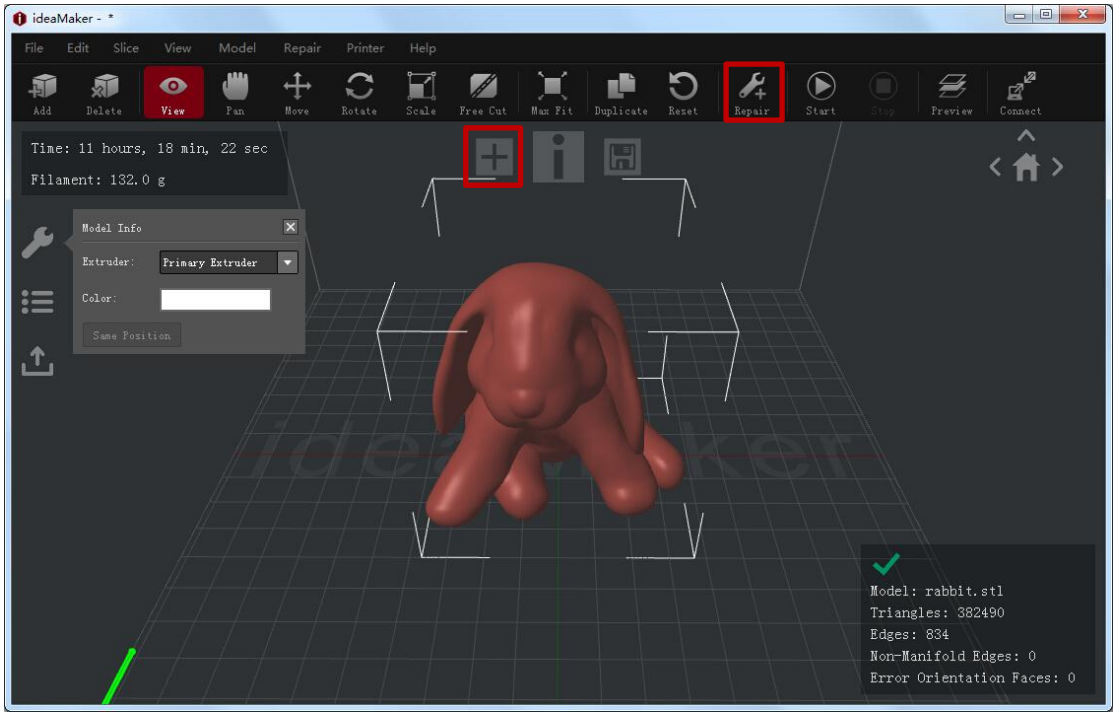


5. Click ***Finish*** and start your first print.

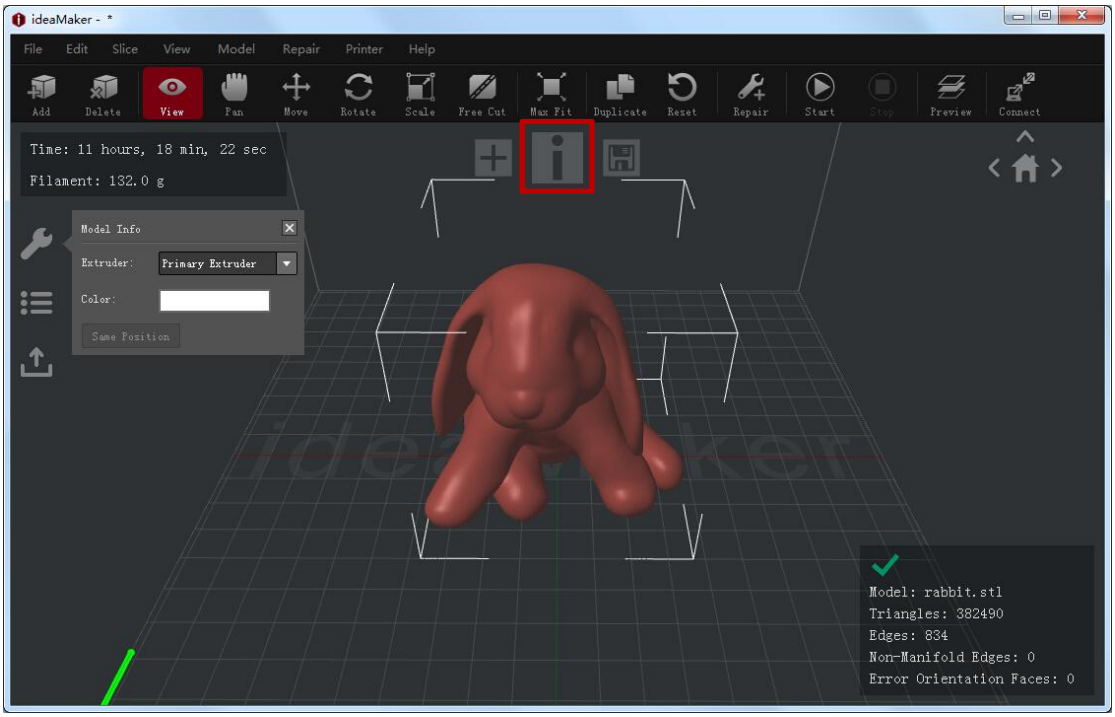
Let’s Print!

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

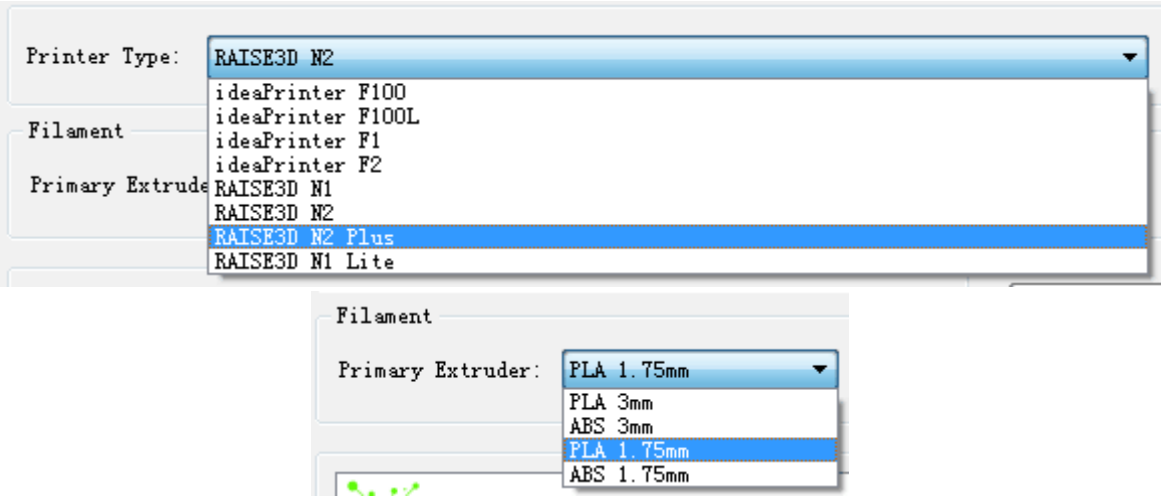
1. Click the button “ **+** ” to import a .stl model. Click the “***Repair***” button to execute an auto-repair of your model if the box at bottom-right corner shows error warnings.



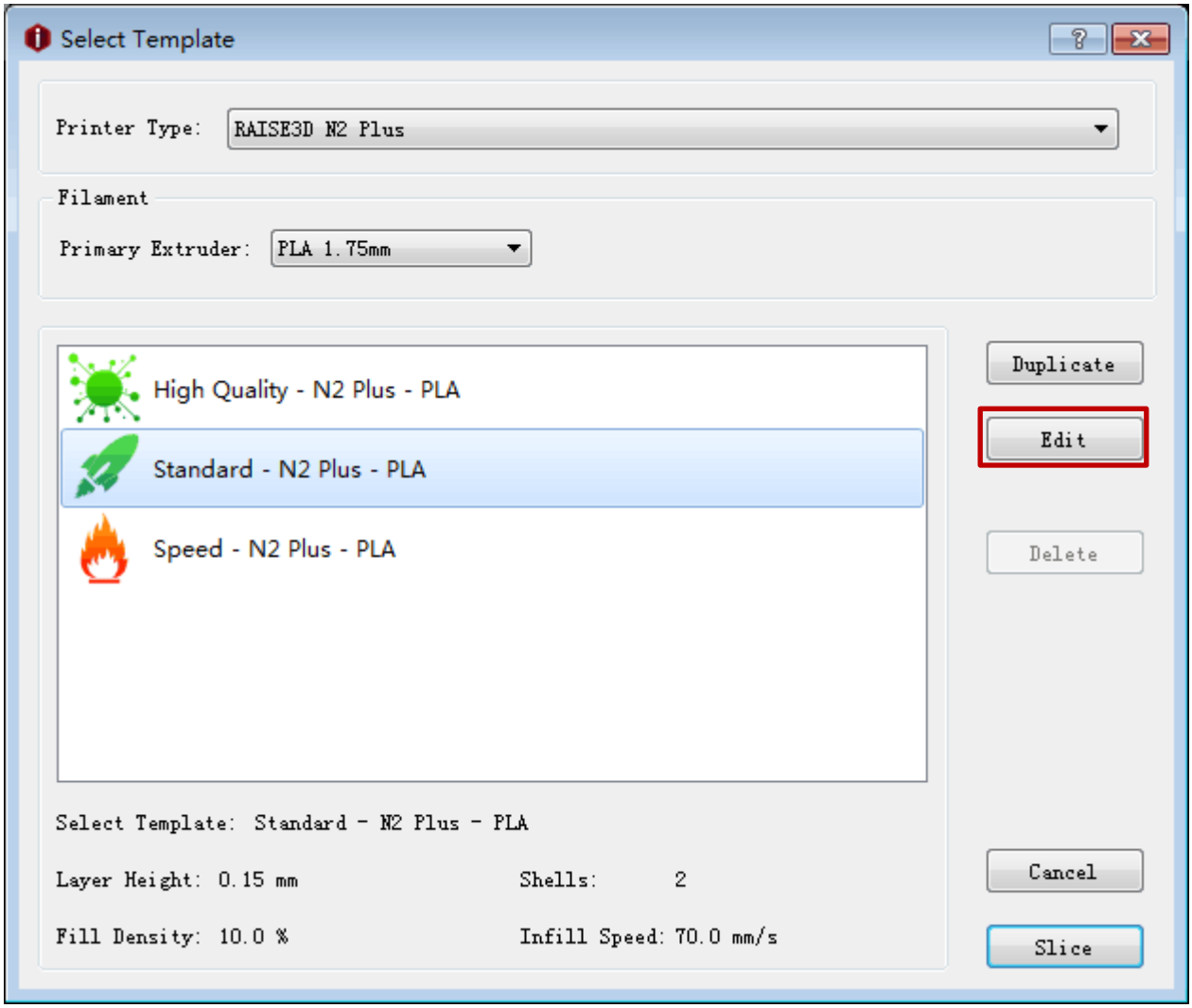
2. Click the button “ **i** ” to start slicing the model.



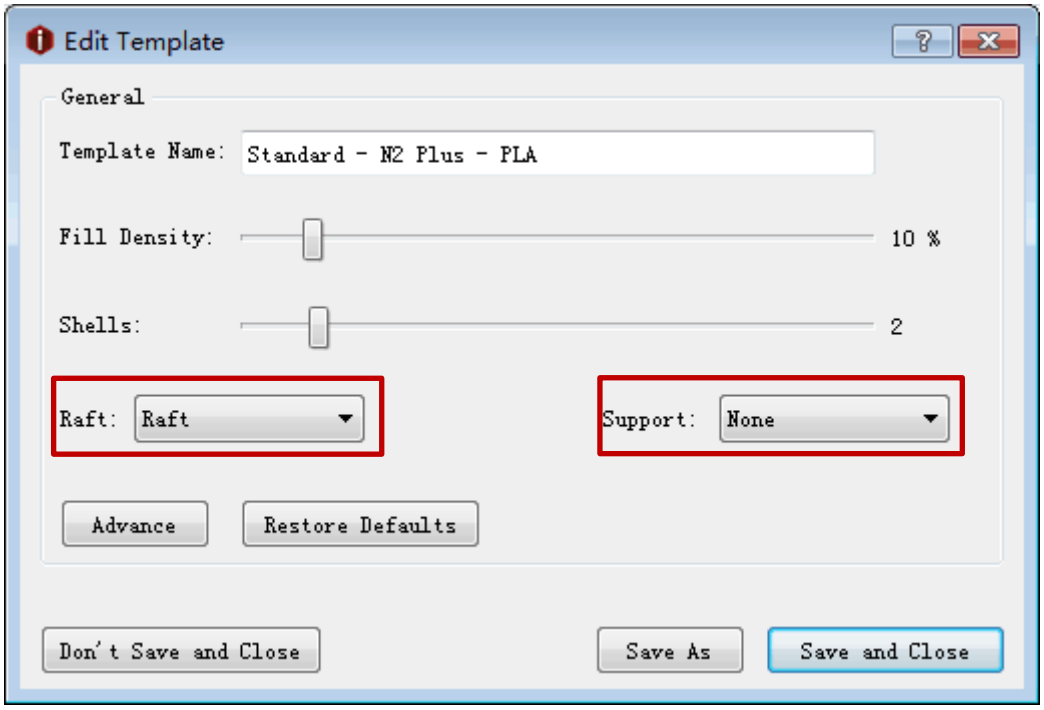
3. Select your printer type and filament type.



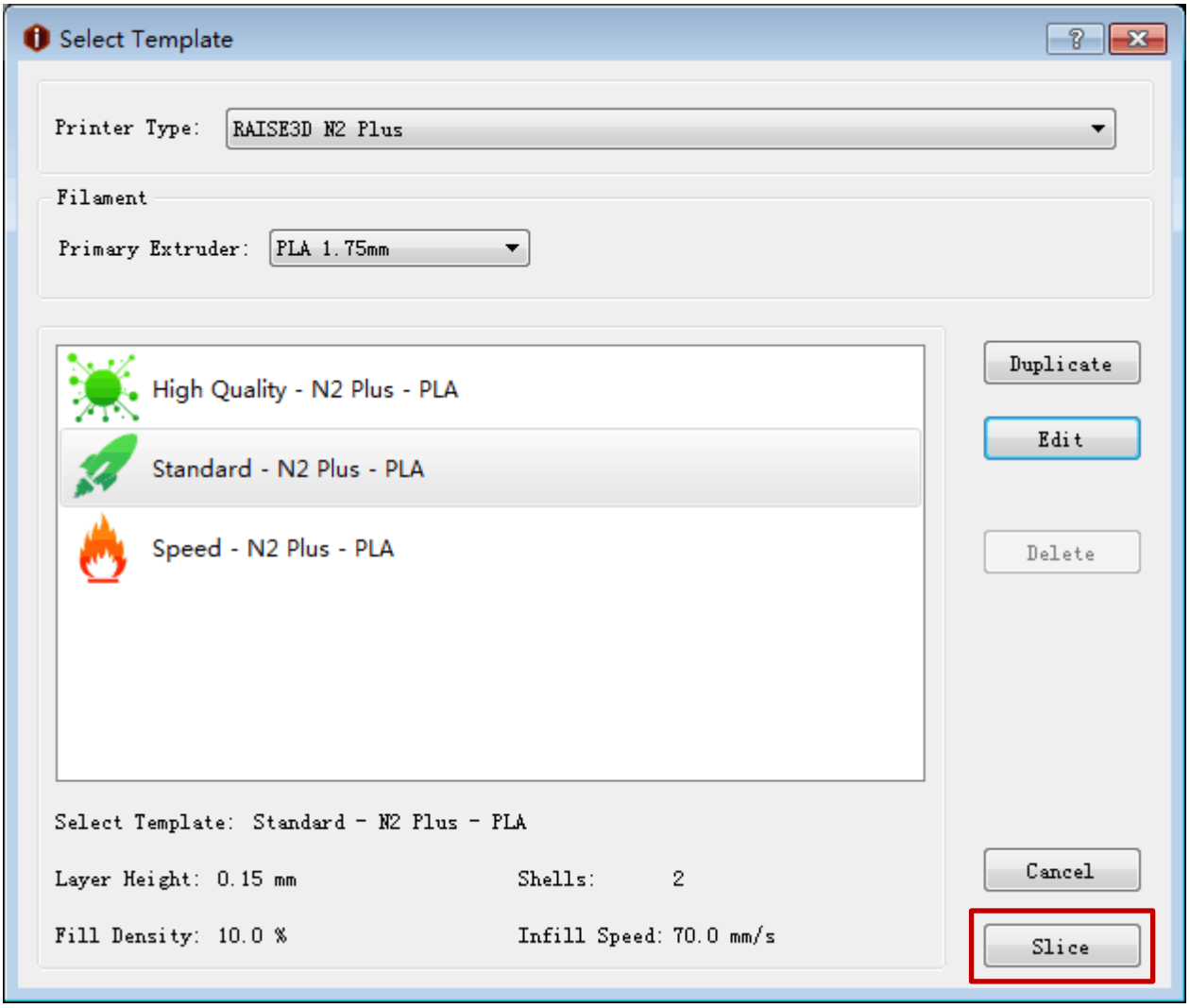
4. Select a slicing template or create a new template yourself by duplicating one of the three existing templates.



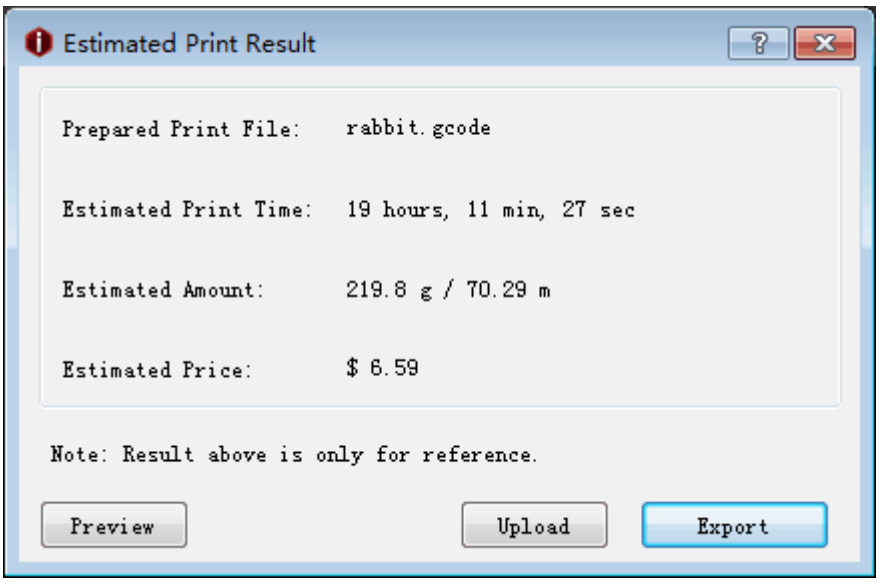
5. Click "***Edit***" button or double click the template (or not if you duplicate a new template). Select a '***Raft***' type and '***Support***' type, then click "***Save and Close***" button. (You can edit other parameters in '**Advance**' as well if you want. We will take about that in the later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



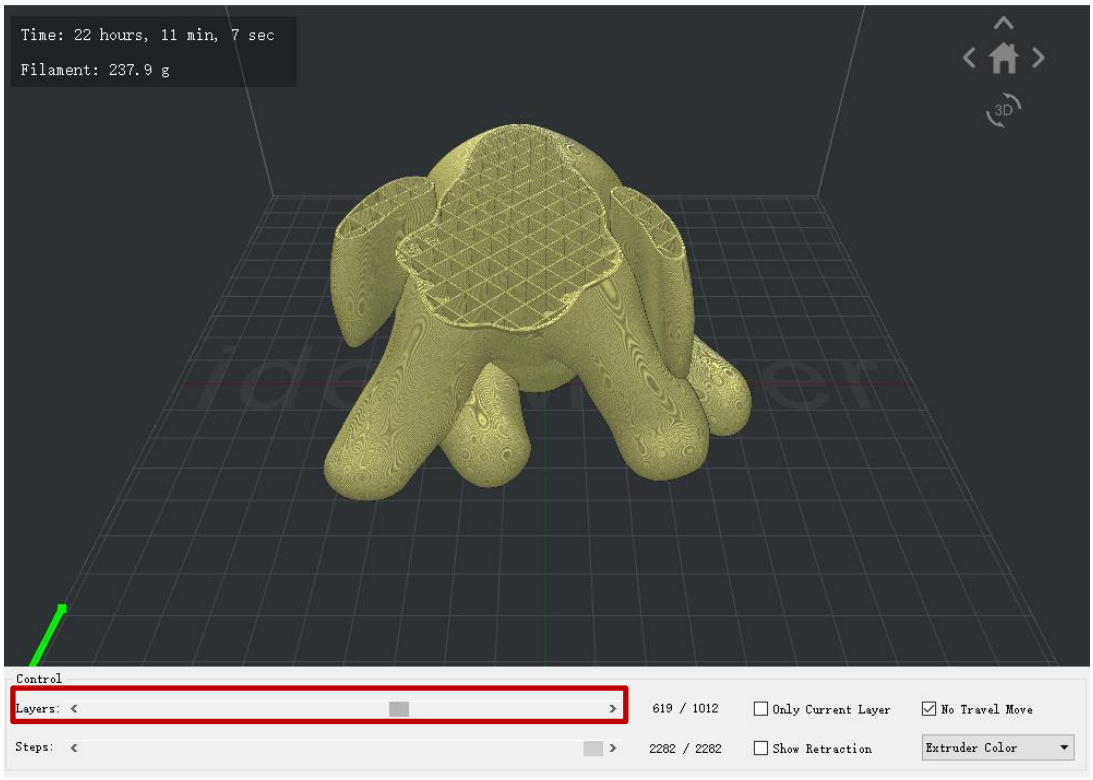
6. Click "***Slice***" button.



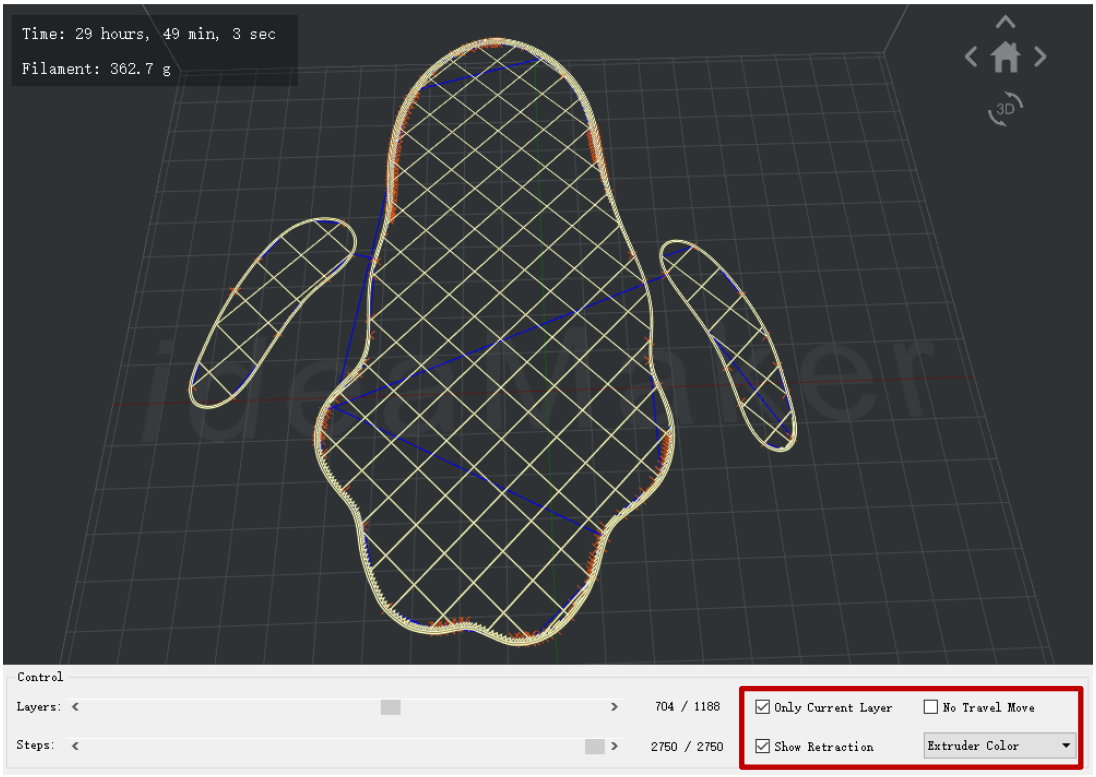
7. After slicing is done, ideaMaker will provide estimated data for your reference.



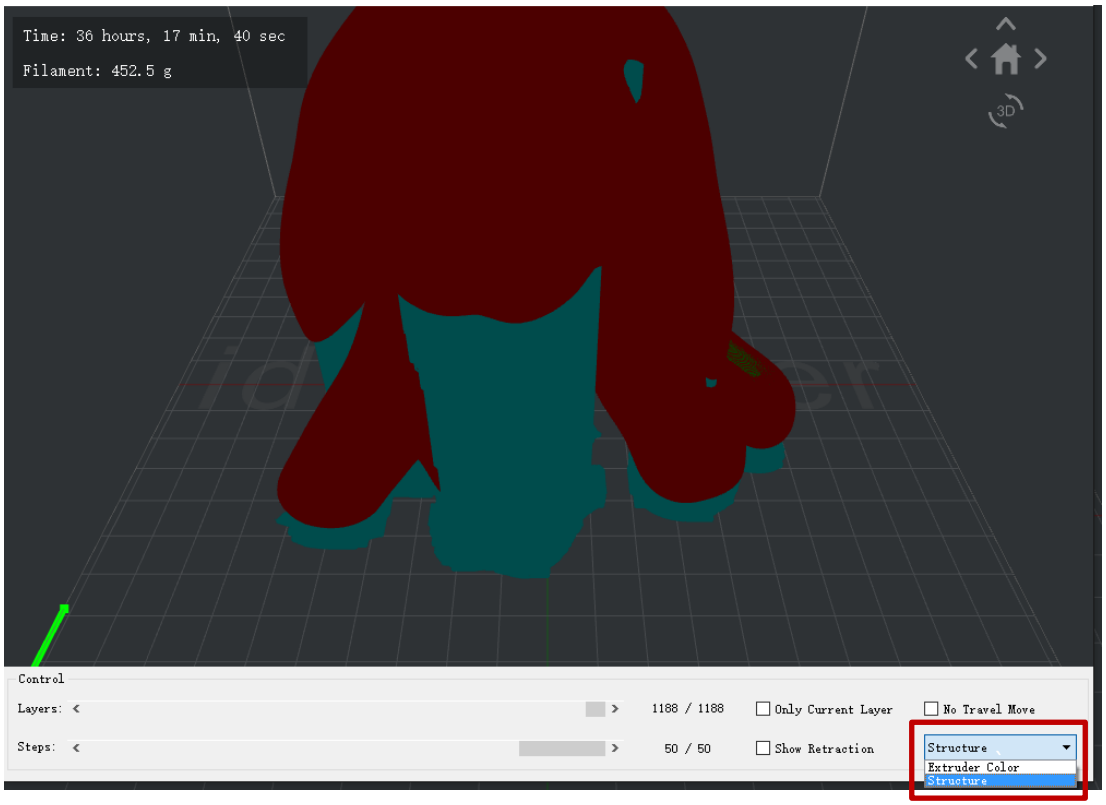
8. Click "***Preview***" button so that you can check the sliced model layer by layer.



And in this page you can check the retraction and travel move of the nozzle by selecting the corresponding options.
The yellow lines refer to the solid print part. The blue lines refer to the moving path of nozzle. The red marks refer to the retraction points.



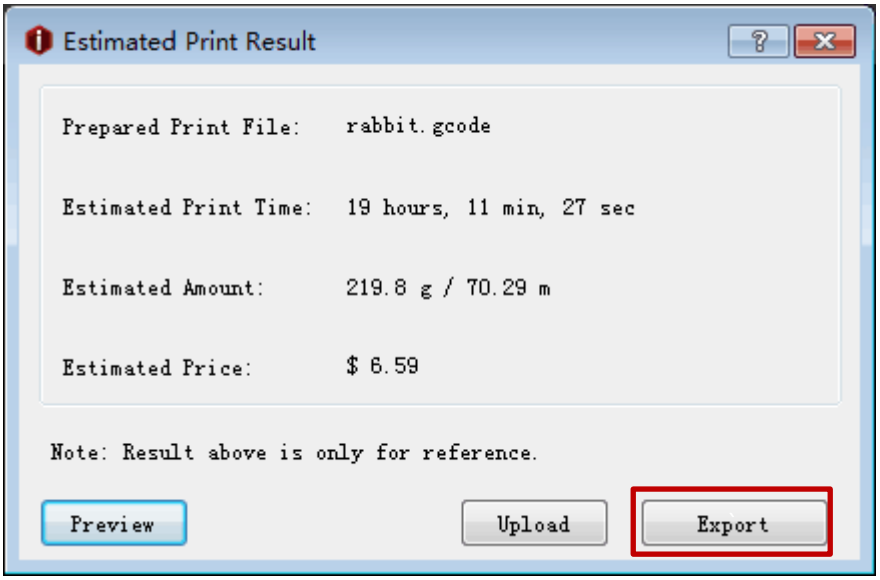
It can also show different structure in different color by selecting ***Structure*** in option box.



9. Close the preview dialogue box after confirm.

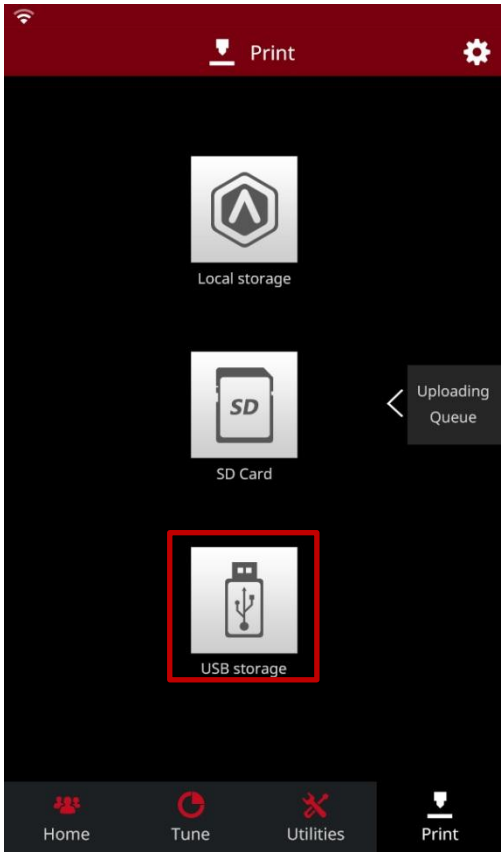
Now you will have 2 options to load the files to the printer.

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



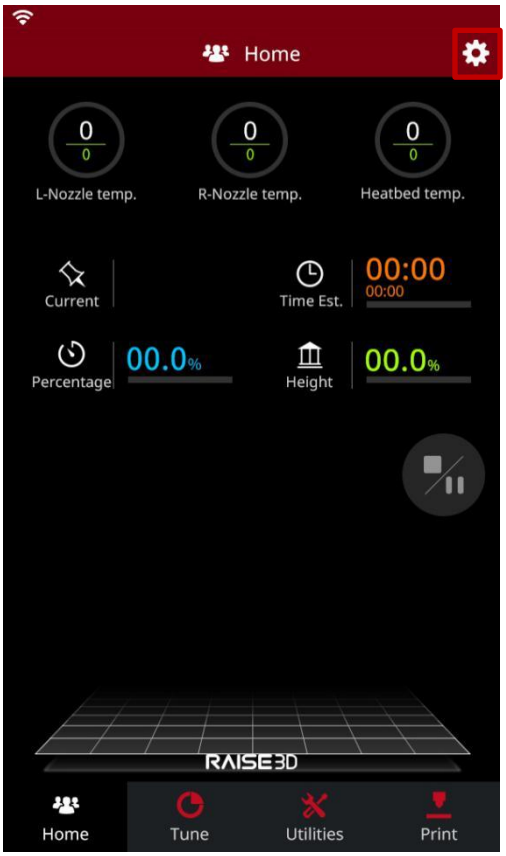
1. If you want to save the sliced files, you can export the files to folders in your computer as well. Then copy the sliced files (.gcode file and .data file) to USB storage. It is important to copy both files.

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

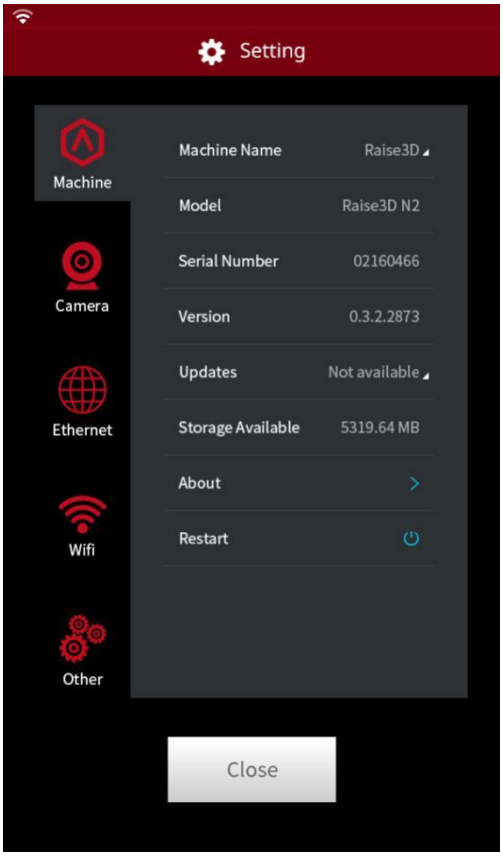


Option2: Upload the sliced file by WIFI

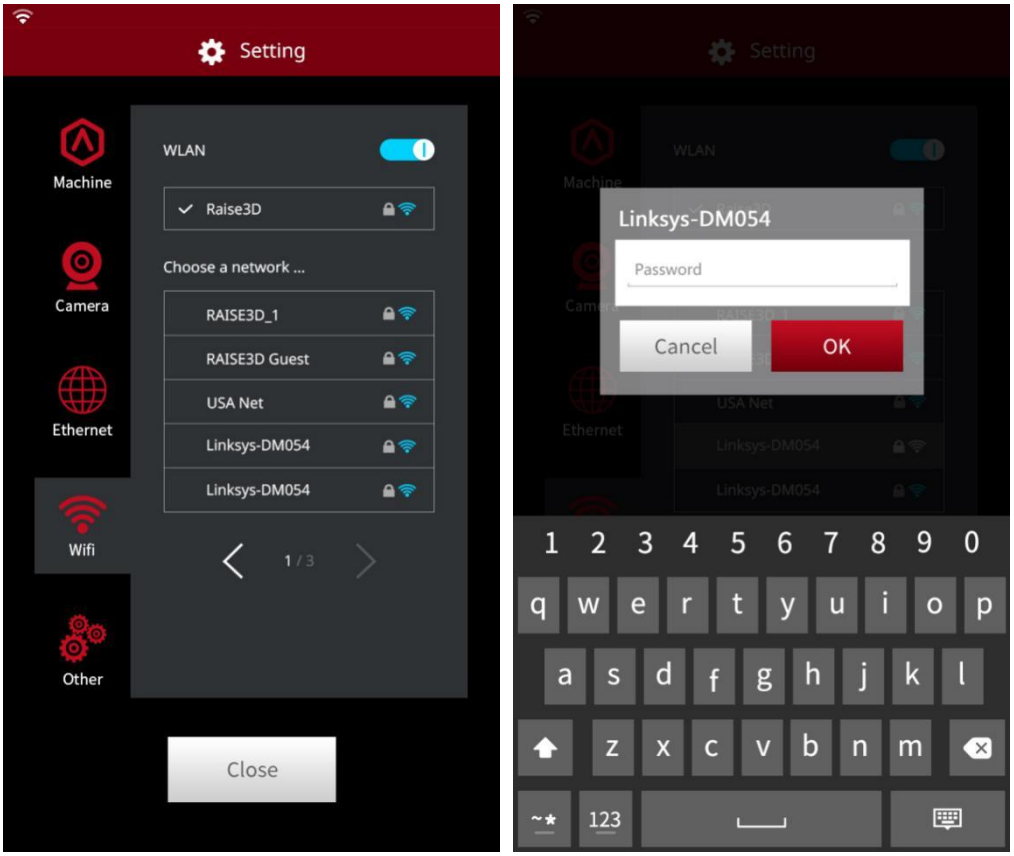
1. First you need to make sure the printer and your computer are connected to the same LAN. For WIFI connection, there is a little gear button at the top-right corner of the screen.



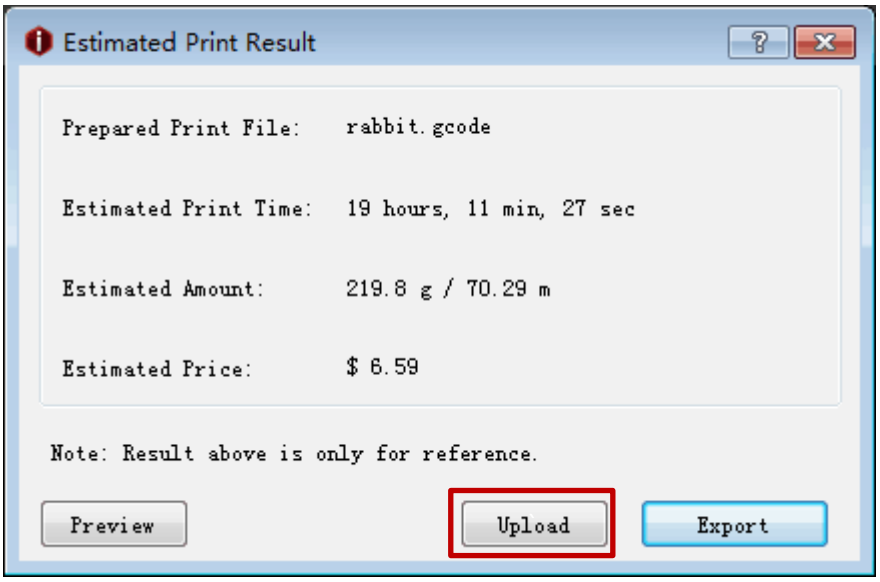
2. Press the little gear button to go to the setting page.

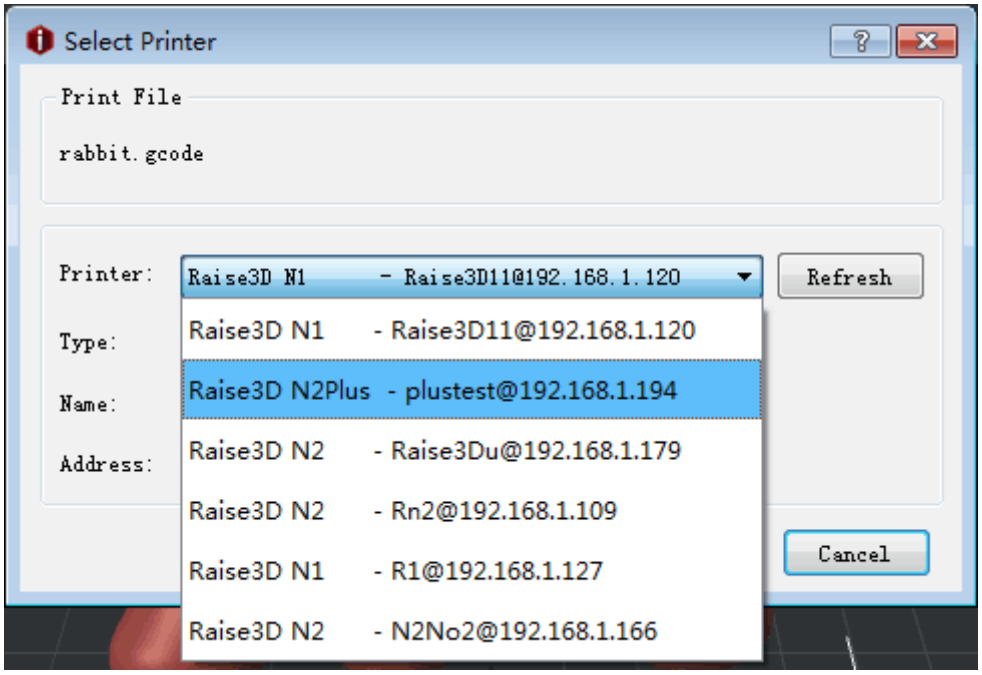


3. Choose ***Wifi*** tab. Enable ***WLAN*** and choose your network from the list then input the password.

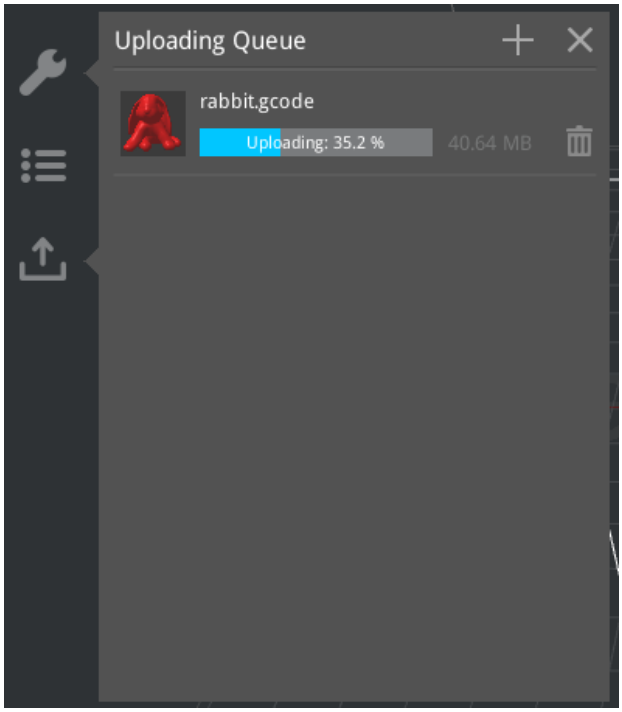


4. After connecting WIFI for your printer, select ***Upload*** and then you will see the ***Select Printer*** page. You can choose the printer you want to print with here.



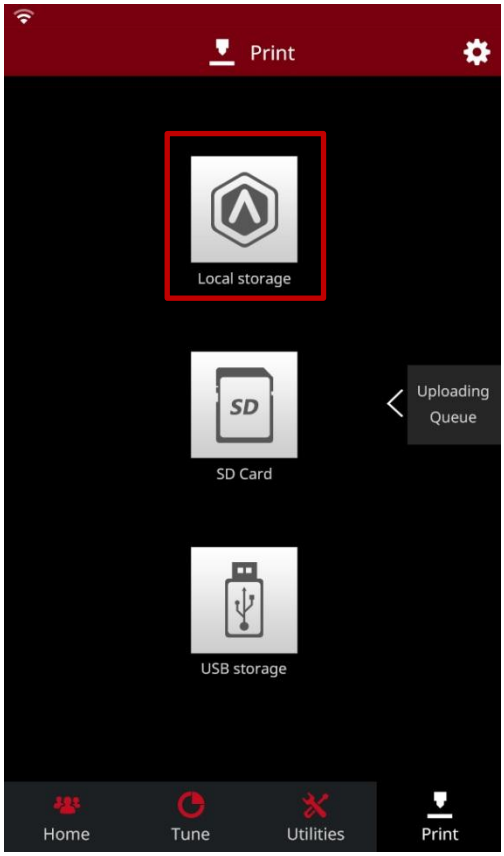


5. After you clicking ***Upload***, the ***Uploading Queue*** will appear on the left side of you screen. You can check the loading progress here.

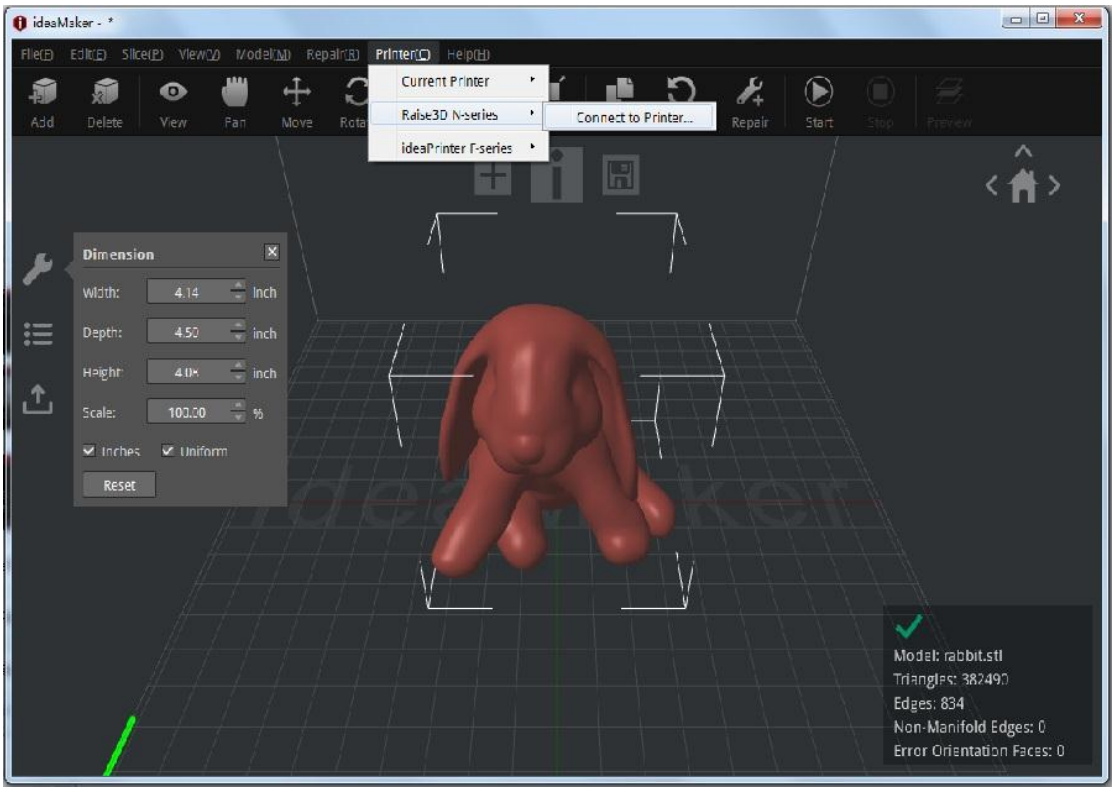


6. When finish uploading, you can start a print.

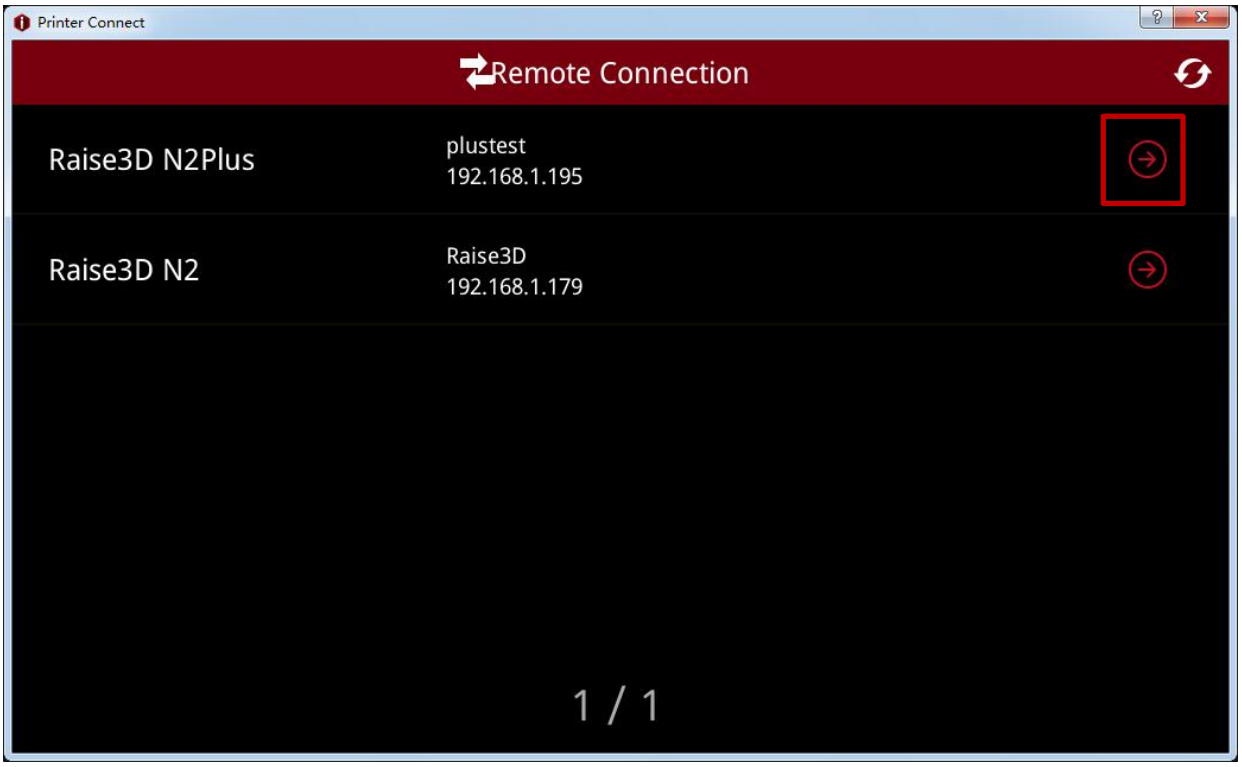
7. You can choose the print from the touchscreen on the printer. The uploaded files are in ***Local storage***.



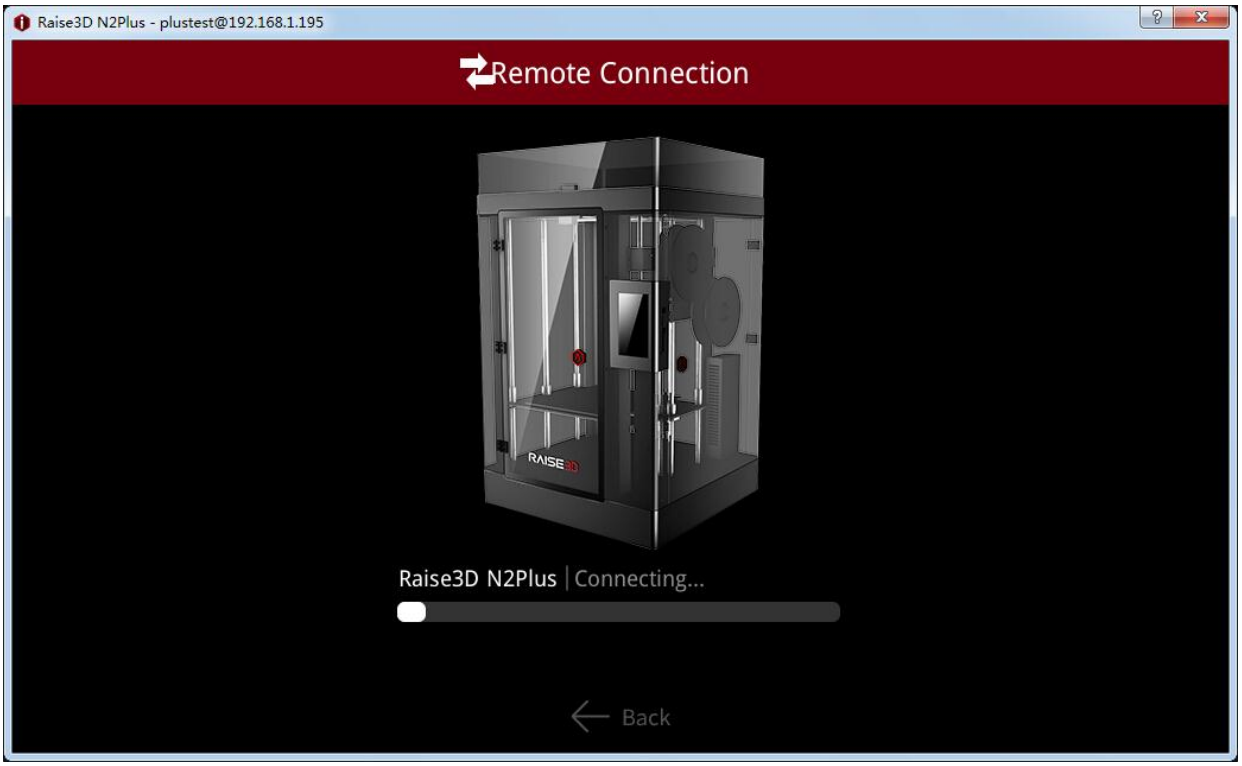
Or through a remote connection to your printer through ideaMaker. Choose ***Printer*** -> ***Raise3D N-series*** -> ***Connect to Printer***.



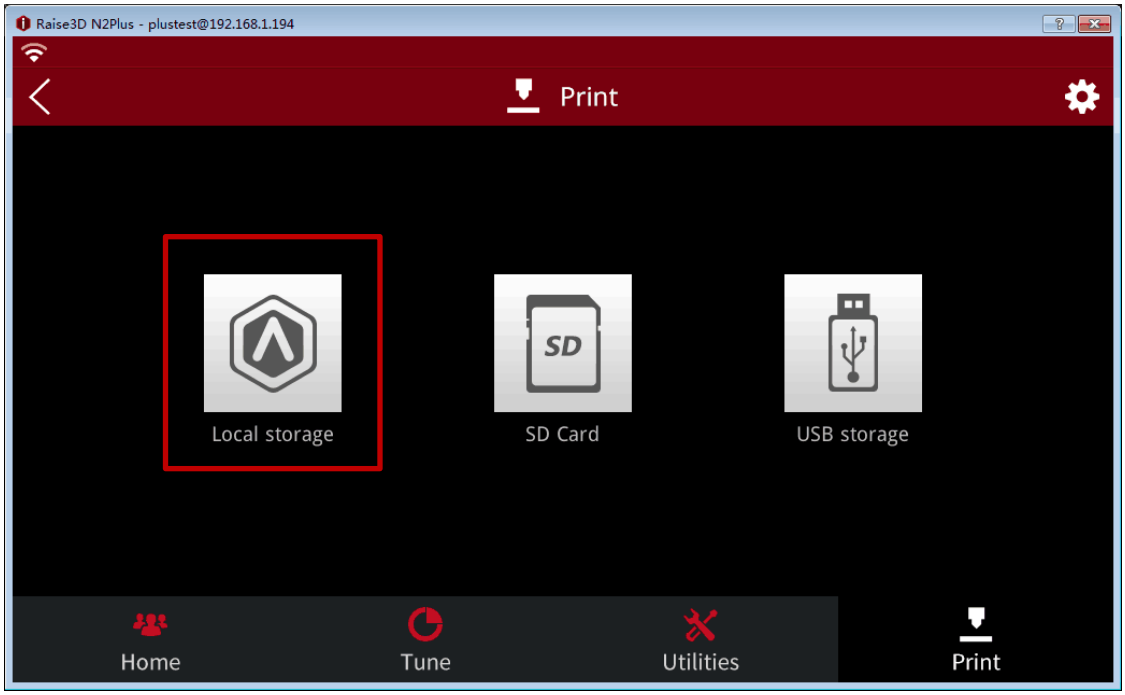
8. Choose the printer you want to connect with in this page.



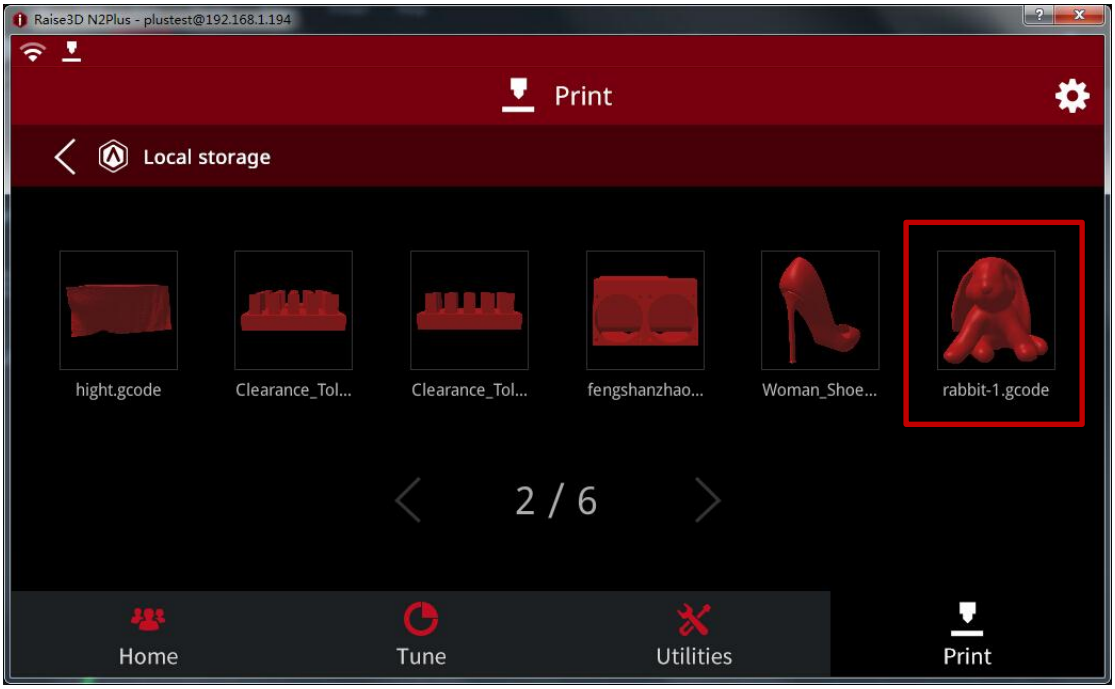
9. The connecting page.

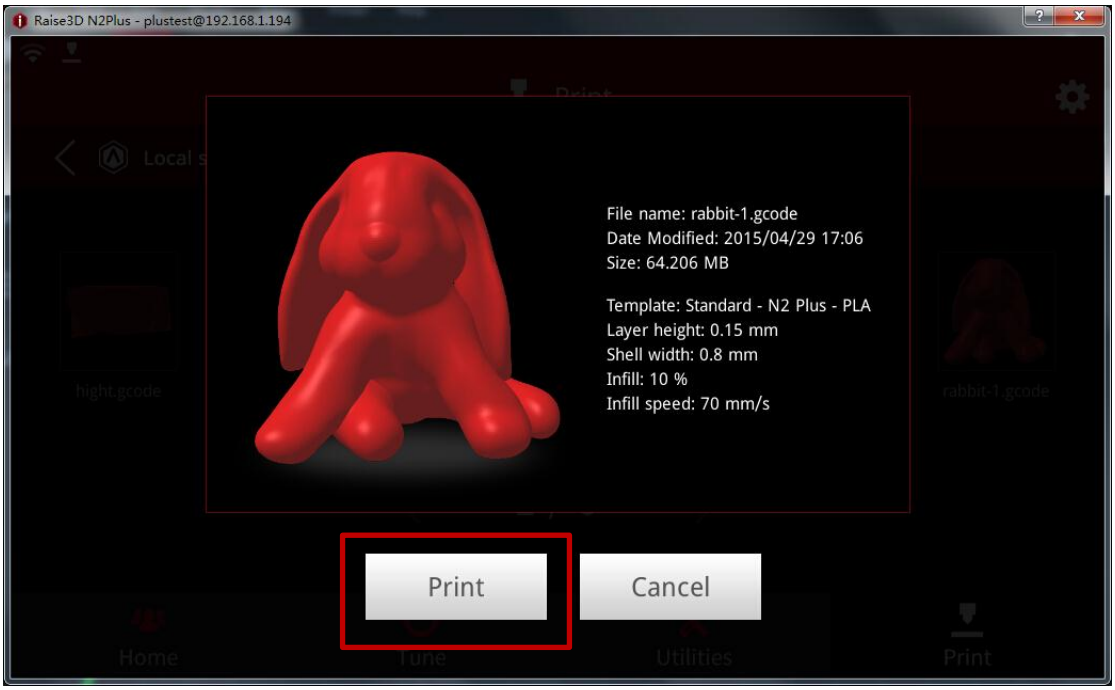


10. Now you can control your printer remotely. ideaMaker has the same operation interface as the touch screen on your printer. You can control the printer directly from here as well. The uploaded files are in local storage.



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

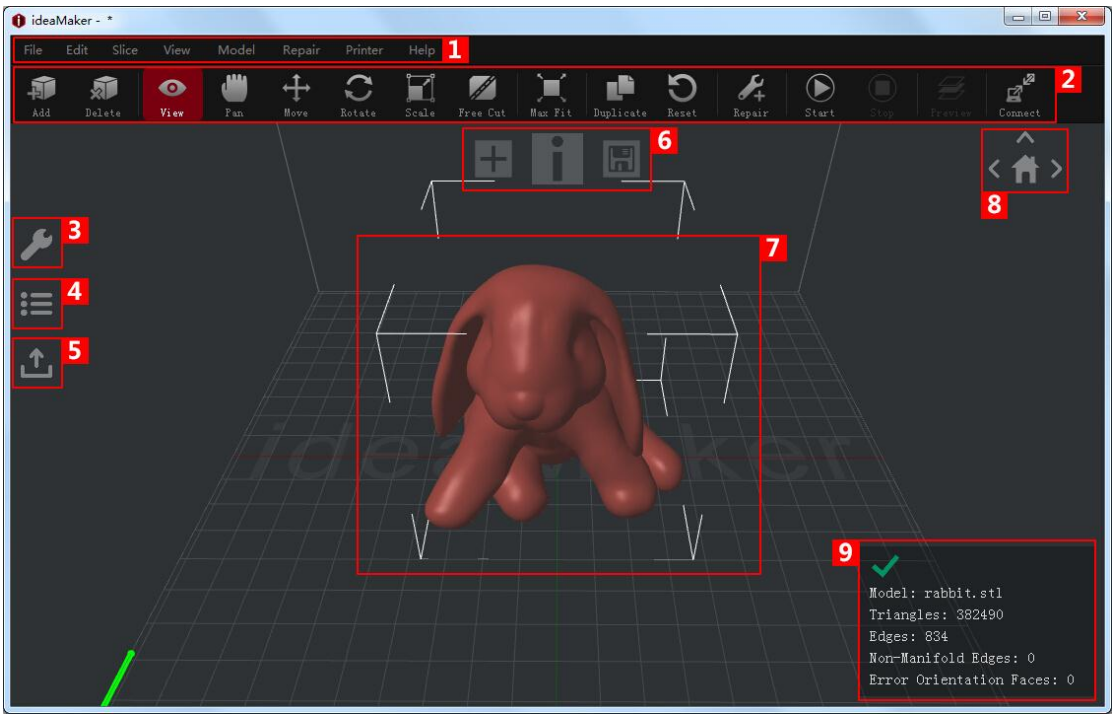



Raise3D N2Plus - plustest@192.168.1.194
File name: rabbit-1.gcode
Date Modified: 2015/04/29 17:06
Size: 64.206 MB
Template: Standard - N2 Plus - PLA
Layer height: 0.15 mm
Shell width: 0.8 mm
Infill: 10 %
Infill speed: 70 mm/s
Print
Cancel
Home
Tune
Utilities
Print

How to use ideaMaker?

Interface

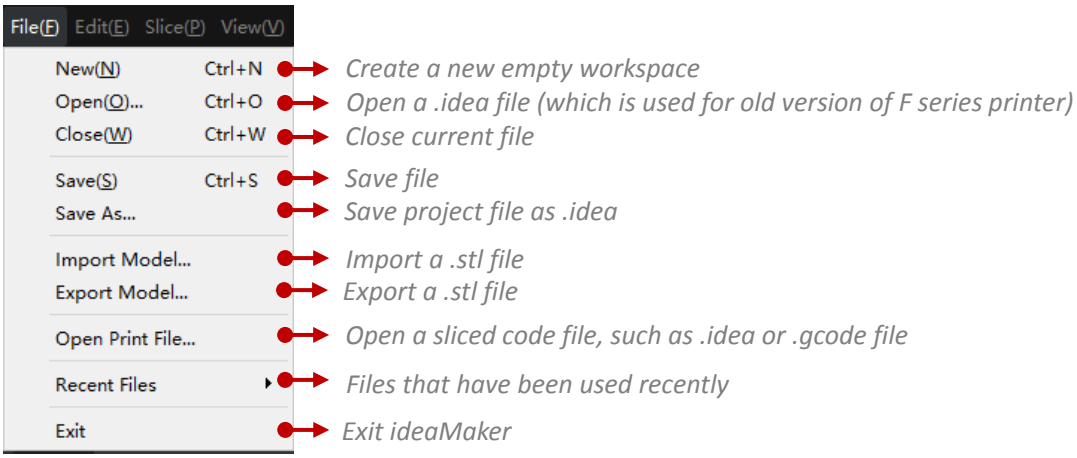
When you open ideaMaker, the main screen is as shown below. We divide it into 9 sections and we will go through them one by one.



1. Menu Bar

Menu bar includes all the operation commands and advanced setting.

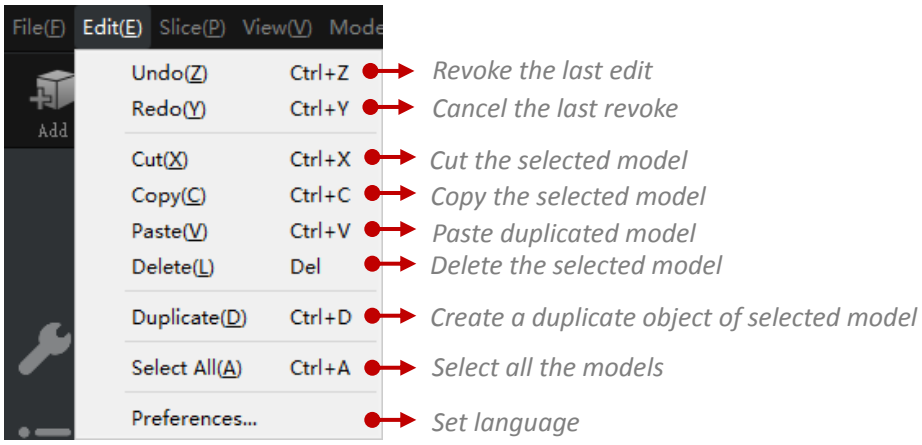
1> File



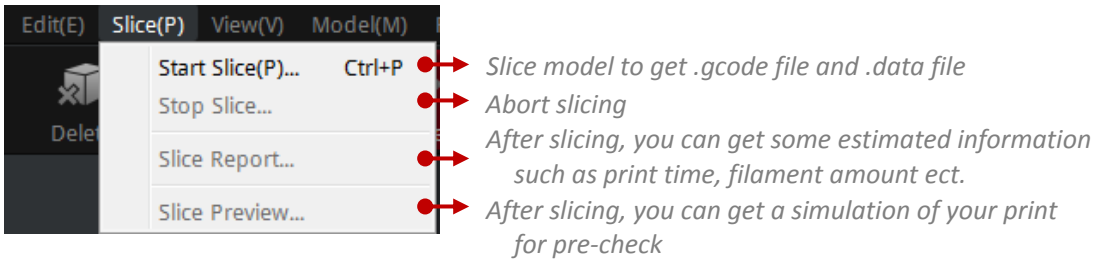
Note:

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

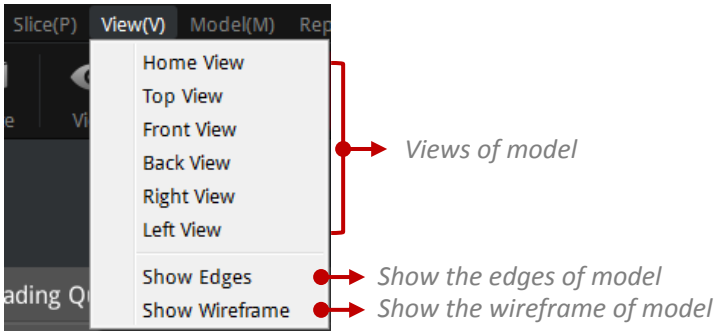
2> Edit



3> Slice



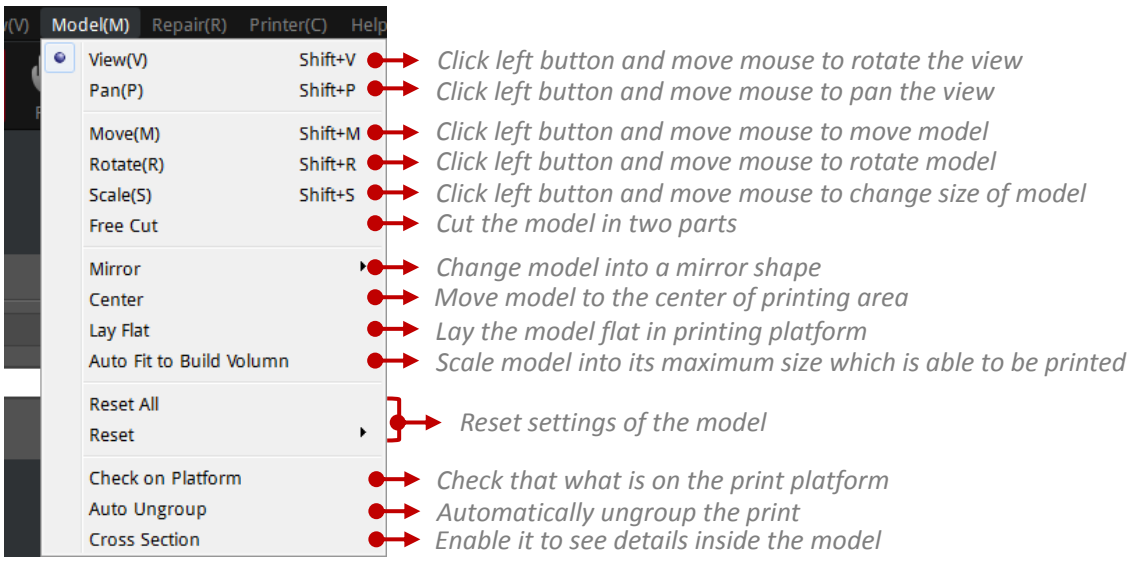
4> view



Note:

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

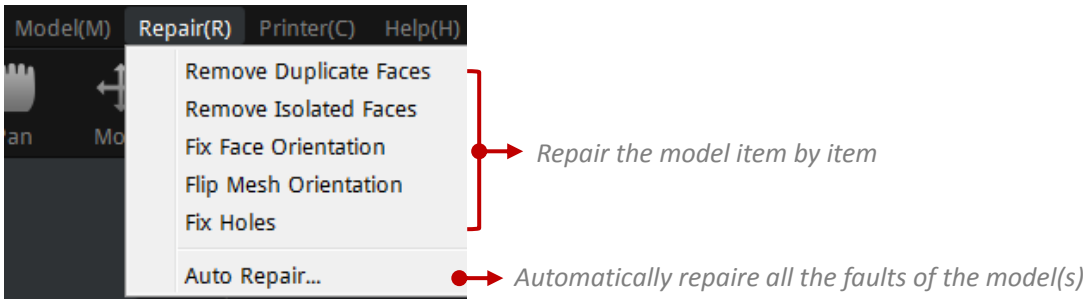
5> Model



Note:

Many tools have been settled in tool bar, thus you can use them conveniently.

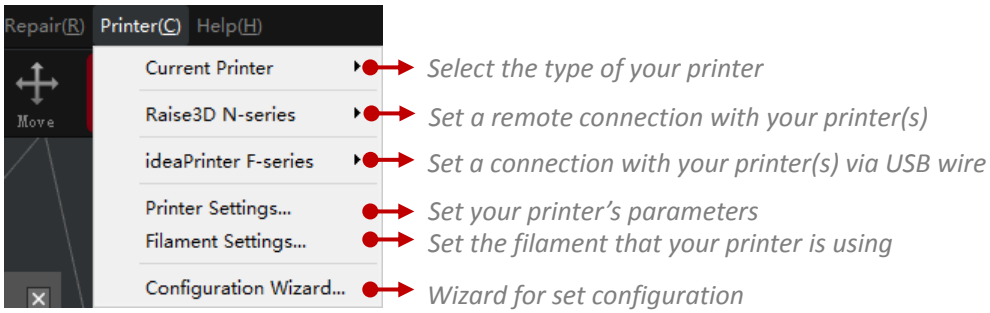
6> Repair



Note:

You can also find auto repair icon in tool bar.

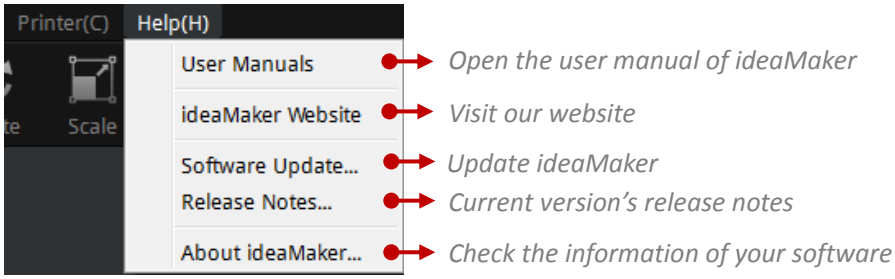
7> Printer



Note:

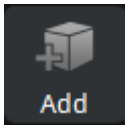
Filament Settings: You can set type, diameter, density, price and compensation of the filament. Also you can build a new filament template for you own filament here.

8> Help

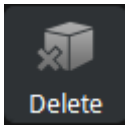


2. Tool Bar

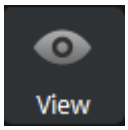
Tool buttons for slicing models, these buttons are shortcut of menu bar.



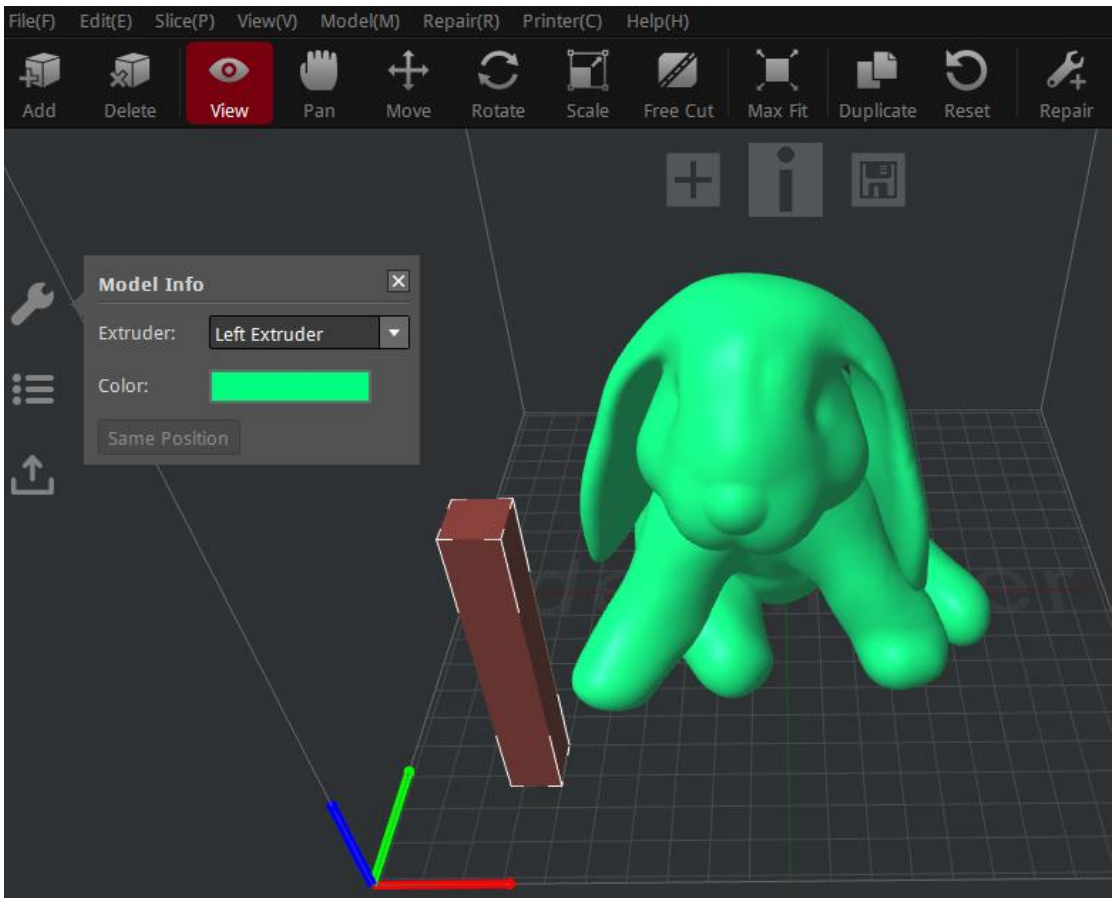
: Add new .stl or .obj file.



: Delete the selected model.



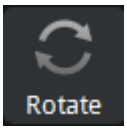
: When you enable this button, you can set model’s color and which nozzle to print it (if you have our dual-extruder printer).



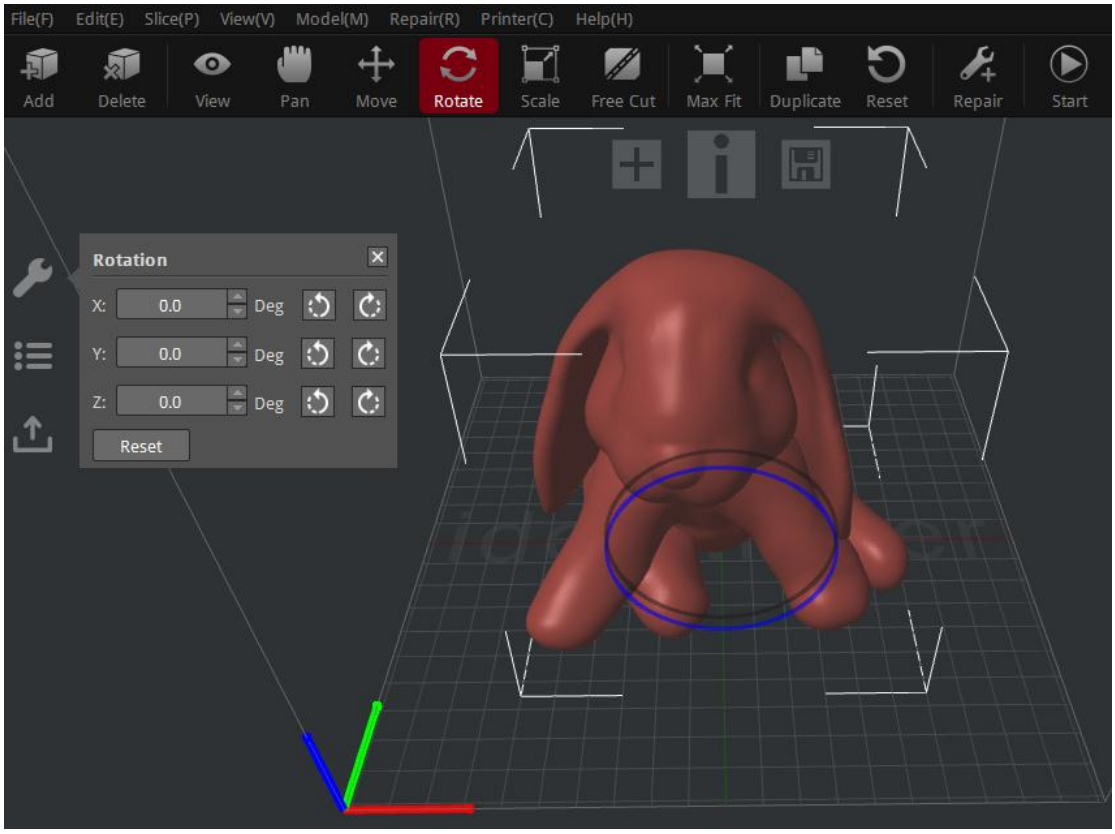
Enabling this button also allows us to watch model in different angles by left-clicking your mouse.

: When you enable this button, you can move ideaMaker’s model window by left-clicking your mouse.

: When you enable this button, you can move model to another place by left-clicking your mouse. You can also set the exact X/Y coordinate value to move the model in the operation property zone.



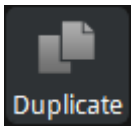
: When you enable this button, you can rotate model to another angle by left-clicking your mouse. You can also set the exact angle value to rotate model in the operation property zone.



: When you enable this button, you can amplify model's size by left-clicking your mouse. You can also set the exact amplify rate value to change model in the operation property zone.

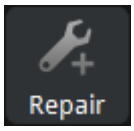
: When you enable this button, there will be a plane which you can move to cut the model apart.

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



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

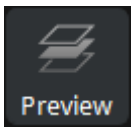
: Revoke all the settings of the selected model.



: Automatically repair model’s defects.

: Start to slice the model.

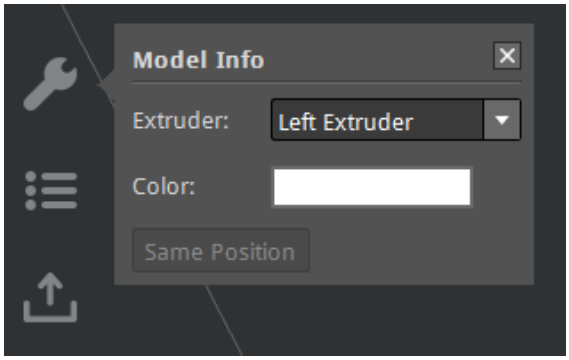
: Abort slicing.



: Show the sliced file in layers.

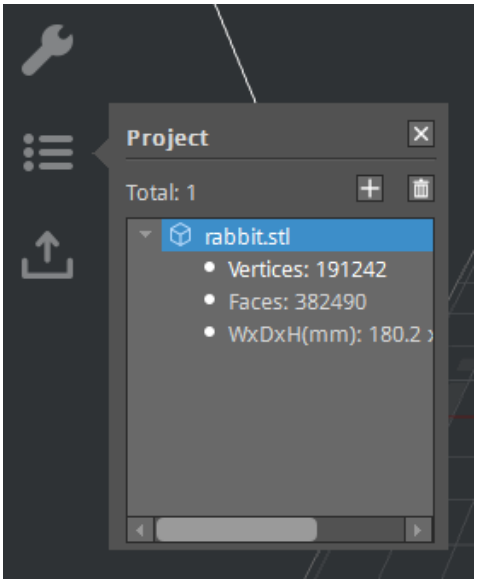
: Connect to the printer with your computer via WIFI.

3. Operation property



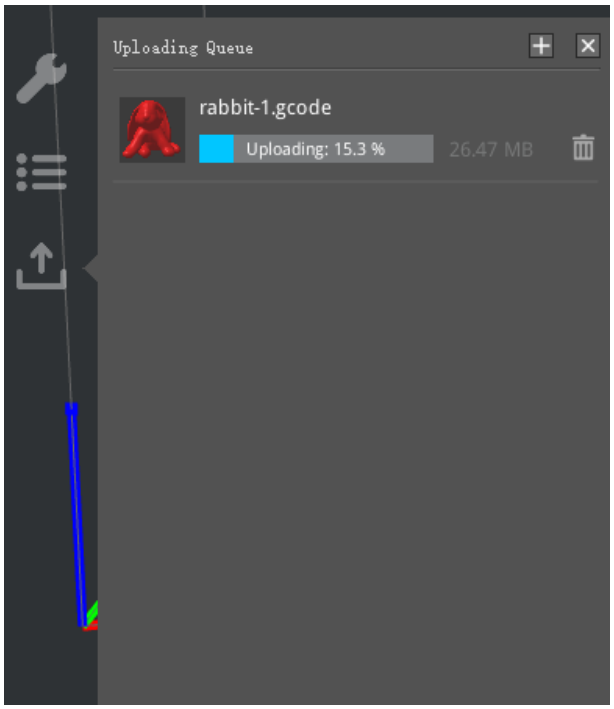
This section shows the information and settings of the selected model.

4. Model list



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

5. Uploading queue



This section shows the uploading list to your printer.

6. Instant operation bar

 : Add models

 : Start slicing directly

 : Save as another printing file

7. Model preview

The section is designed for previewing the current model.

8. Perspective transformation

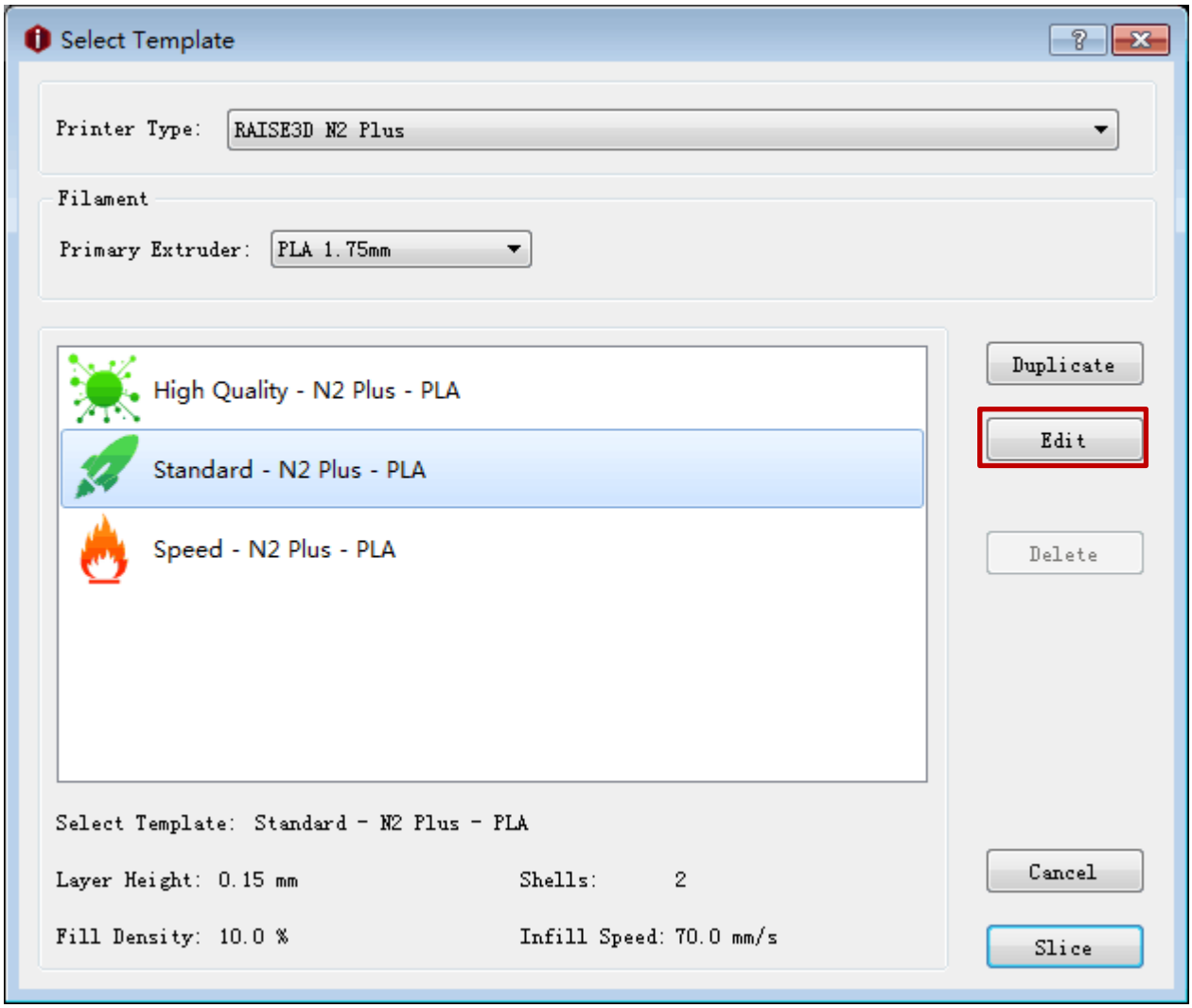
Set ideaMaker to the default view of the object.

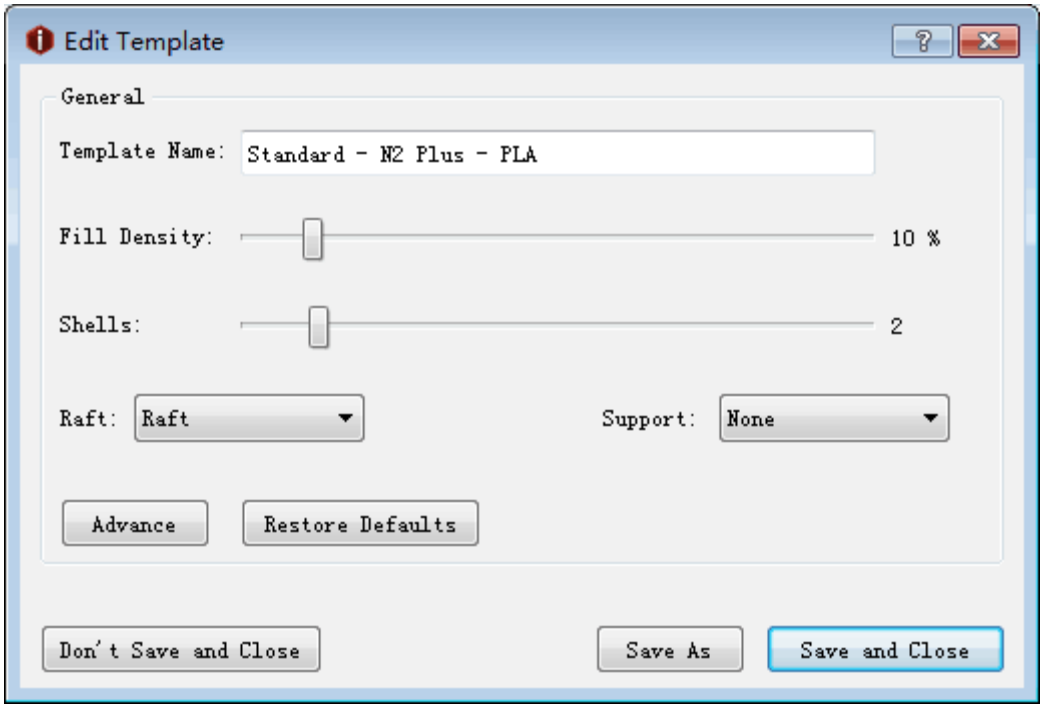
9. Detecting Information

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

Advanced slicing settings

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





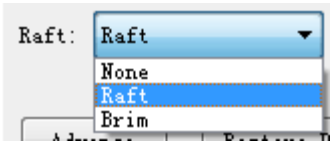
Fill Density refers to the density of infill inside the model, the more infill the model will be the more solid.

Shells refer to the thickness of model's wall.

Raft refers to the type of bottom layer.

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

The ***Brim*** layer will only print a single layer of shell.

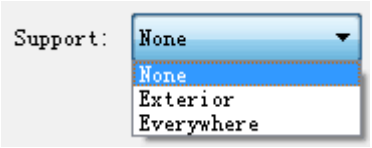


Support means that the printer will print support structure for model's overhang part.

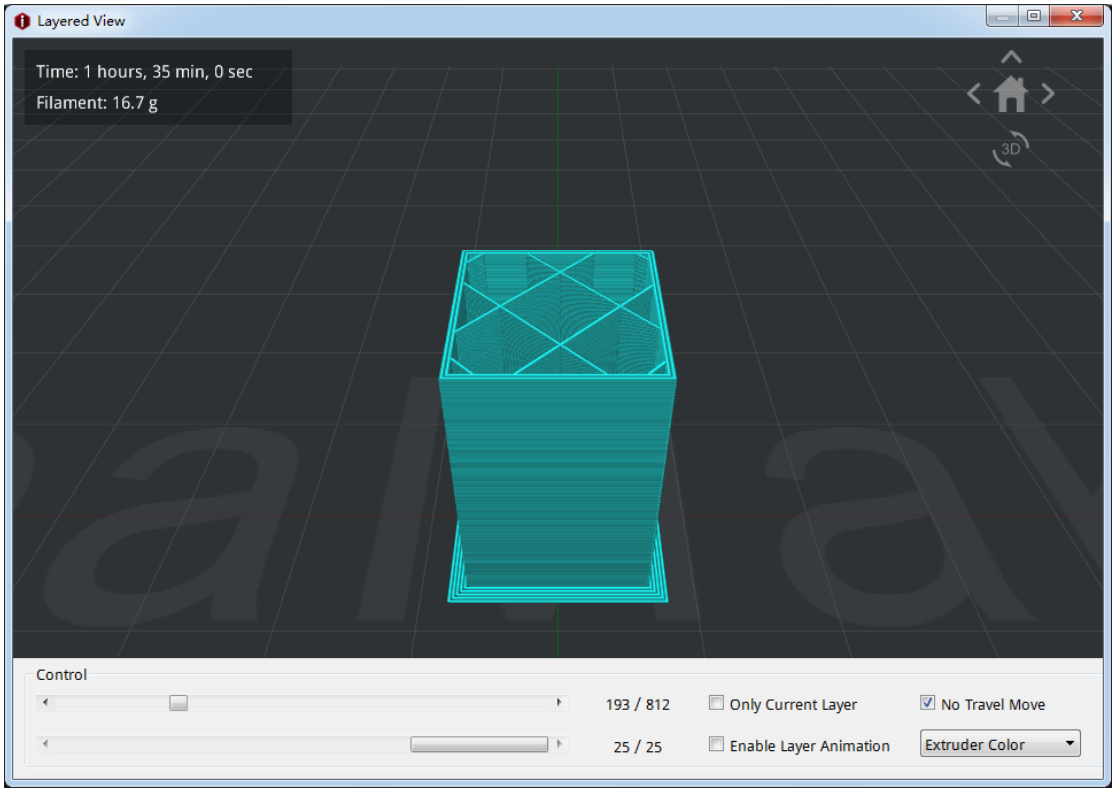
The ***None*** setting refers to no support structure for the model.

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

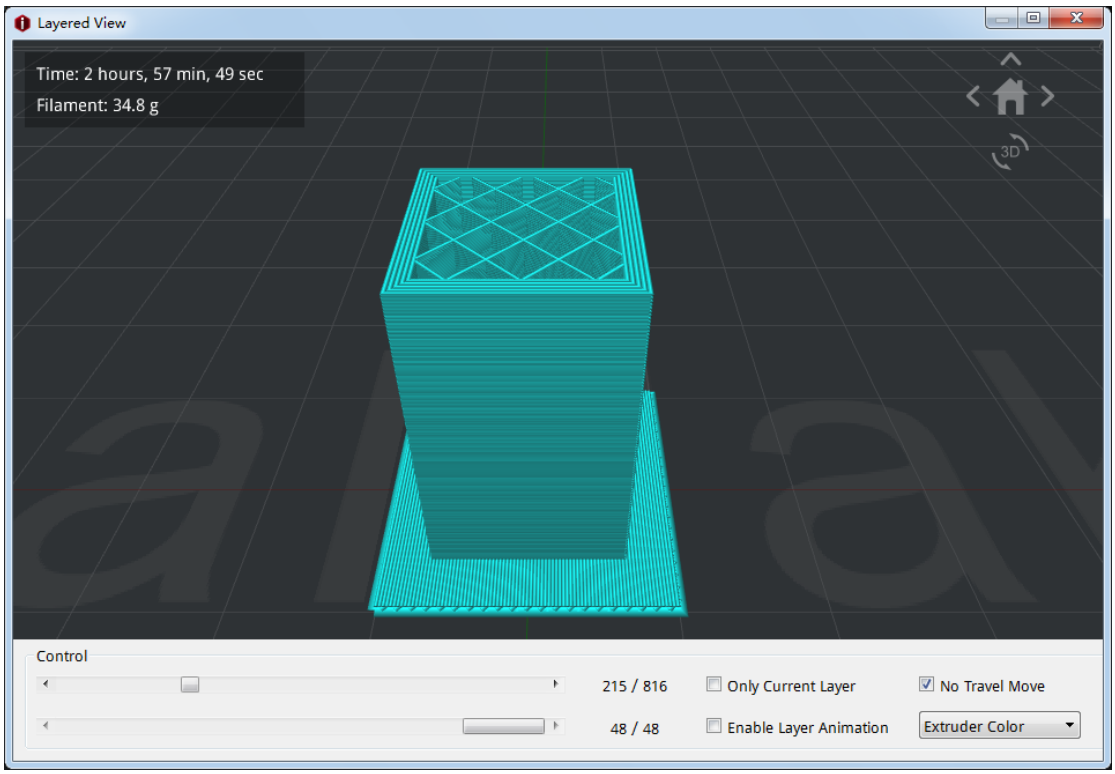
The ***Everywhere*** setting refers to adding support structure to all the overhang part of the model.



With 2 shells, 10% fill density and Brim

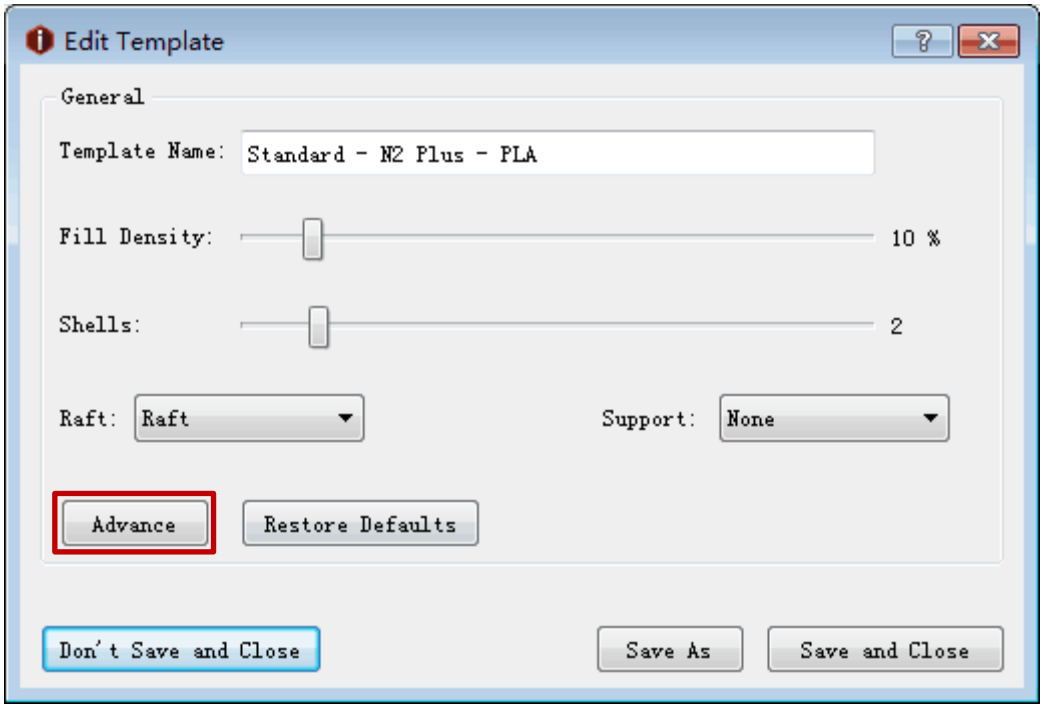


With 5 shells, 20% fill density and Raft



Advance settings

In ***Edit Template***, click ***Advance*** to go to ***Advance Settings*** interface



Layer

General:

Layer Height refers to the height of every single layer.

Speed:

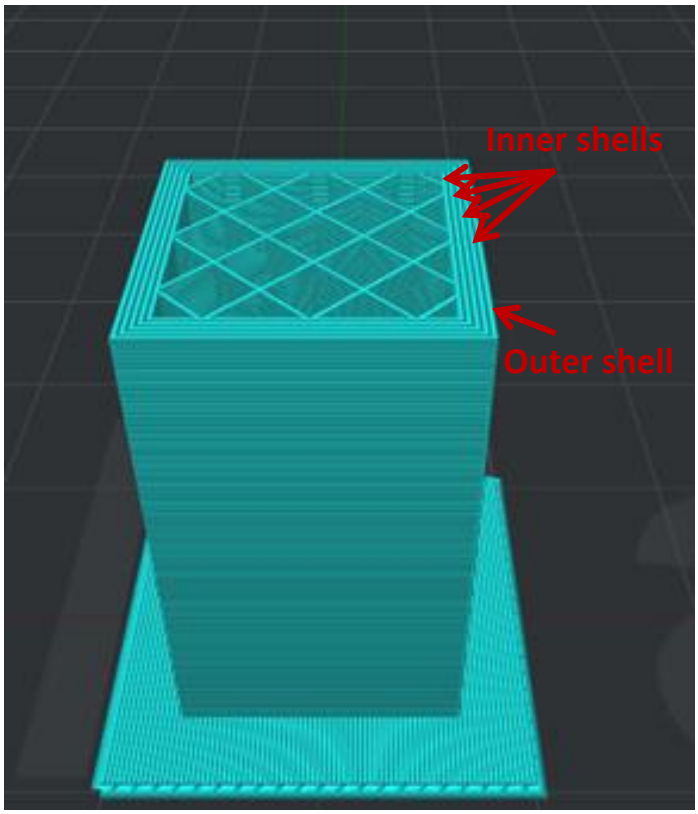
Default Printing Speed refers to the speed of printing non-specified area.

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

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

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

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



Skirt：

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

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

First Layer Settings:

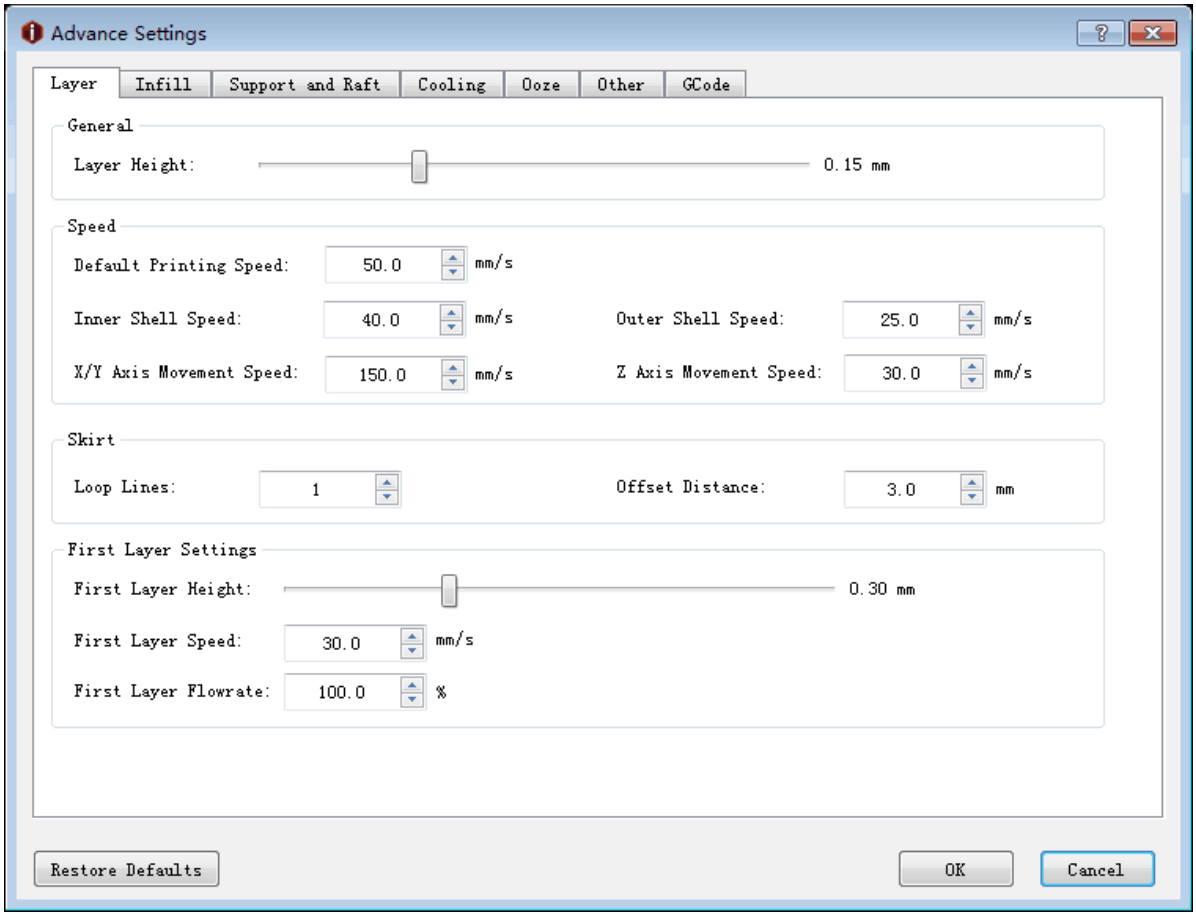
First Layer Height refers to the height of the model's first layer.

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

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

Reset:

Reset Defaults refers to going back to the factory defaults.



Infill

Infill:

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

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

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

Infill Pattern Type refers to infill structure. ***Grid*** refers to the crossing network structure.

Top and Down Solid Part:

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

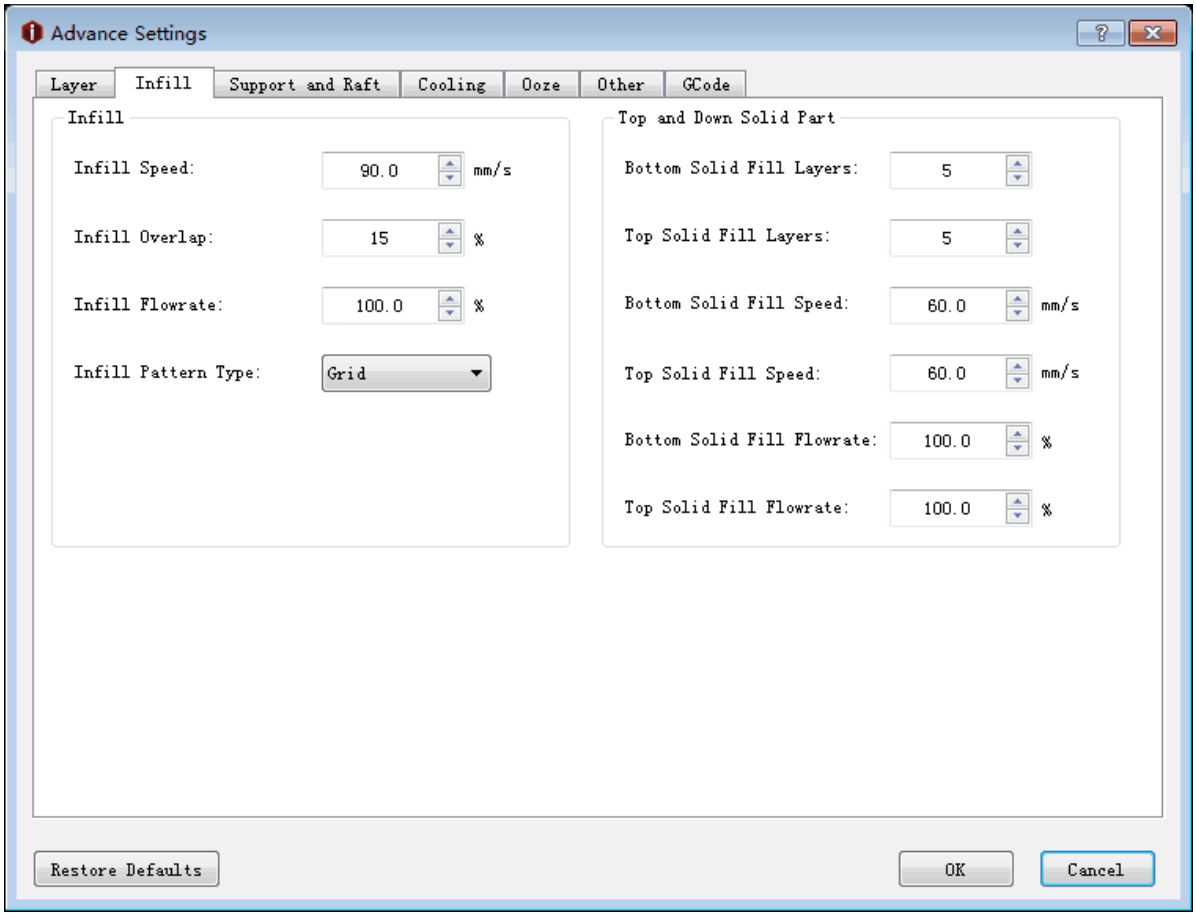
Top Solid Fill Layers refers to the amount of solid top layers.

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

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

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

Bottom Infill Flowrate refers to the infill filament flowrate of the bottom layer.



Support and Raft

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

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

Support：

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

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

Max Overhang Angle refers to the minimal overhanging angle which needs support to print.

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

Vertical offset refers to the distance of the support material from the top or bottom in the vertical directions.

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

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

Sparse Connection refers to enabling the loose connection with the print and the support.

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

Raft:

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

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

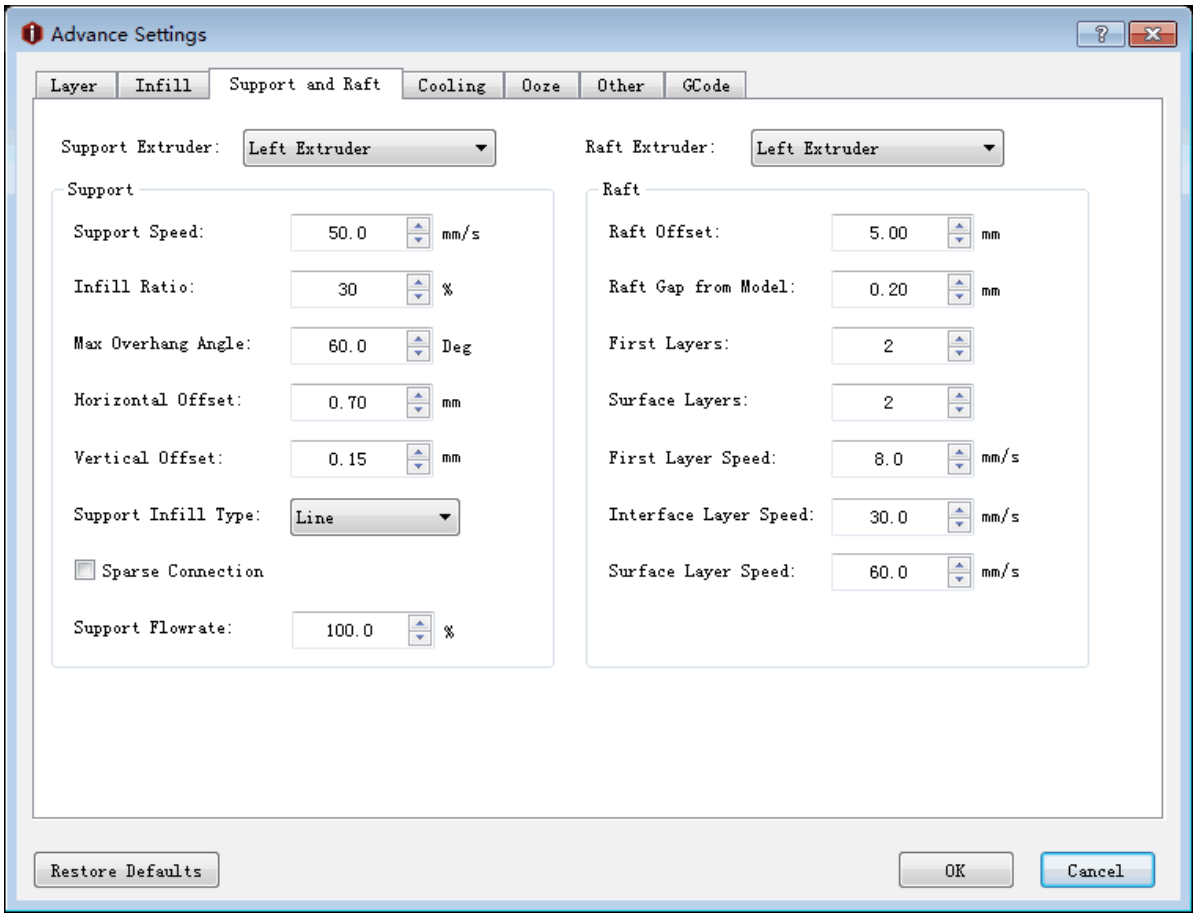
First Layers refers to the amount of layers of the first layer.

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

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

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

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



Cooling

Cooling:

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

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

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

Turn Fan On At layer refers to the layer at which the fan is turn on completely.

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

Temperature:

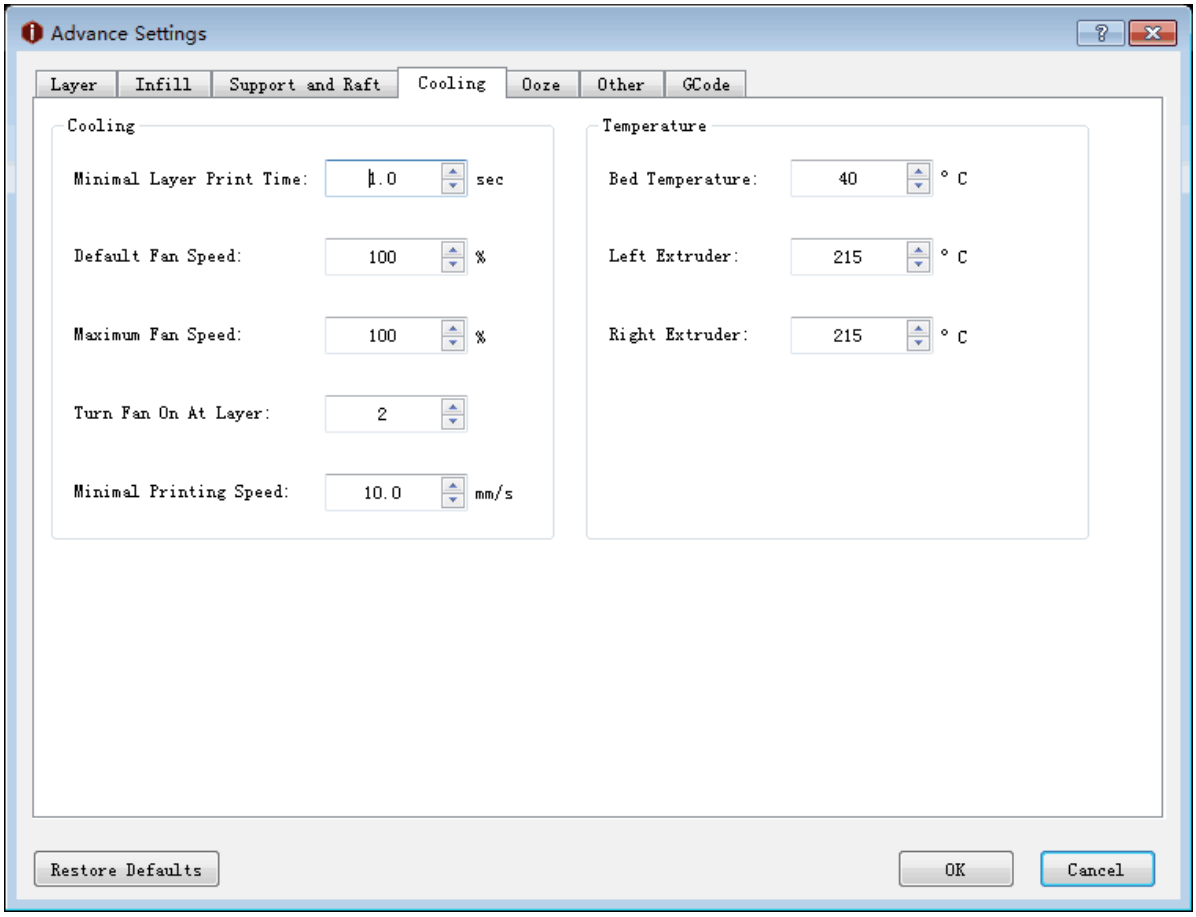
Bed Temperature refers to the temperature of the bed when printing.

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

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

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

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



Ooze

Retract:

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

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

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

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

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

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

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

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

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

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

Multiple Extruders Ooze Control:

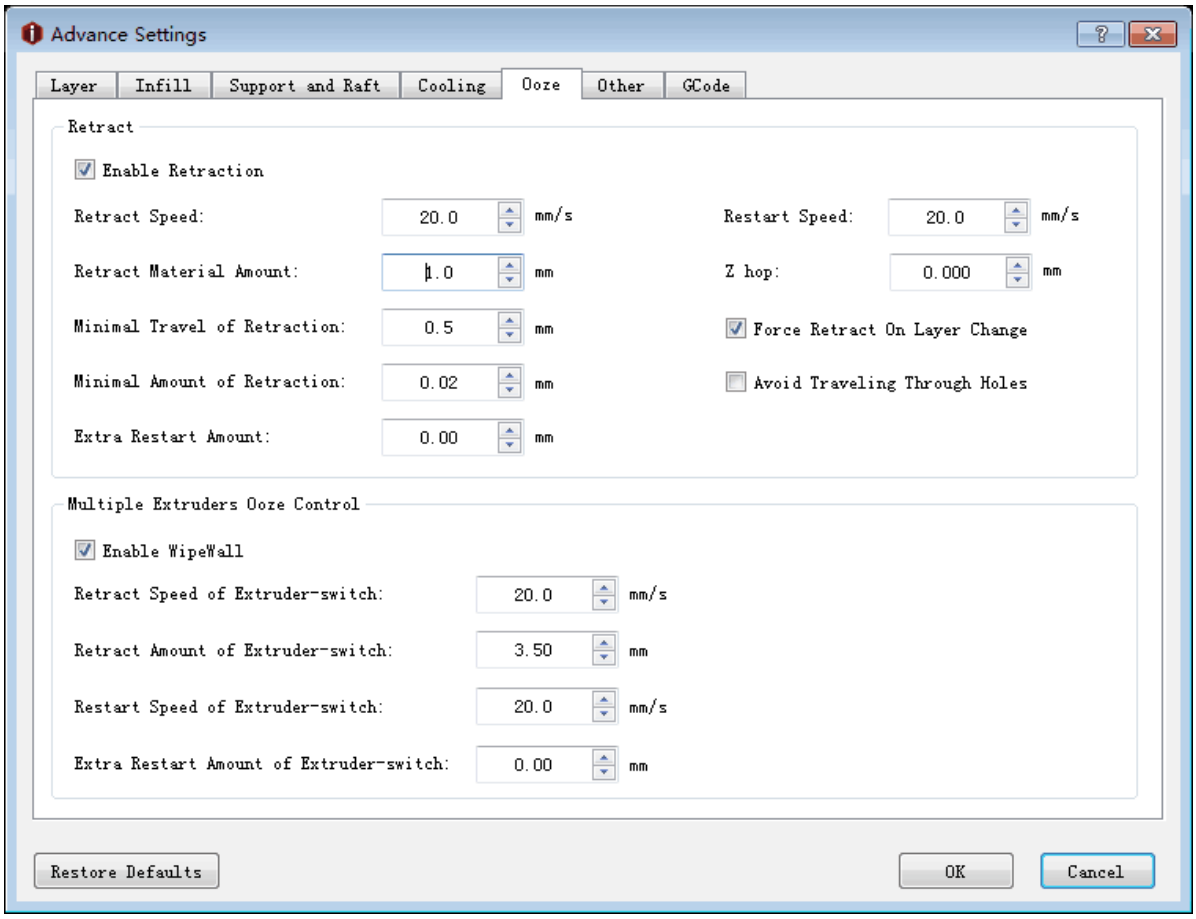
Enable Wipe Wall means when printing a model with dual-extrusion, a nozzle will print a thin wipe wall around the model. This wipe wall will help to clean the nozzle while printing.

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

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

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

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



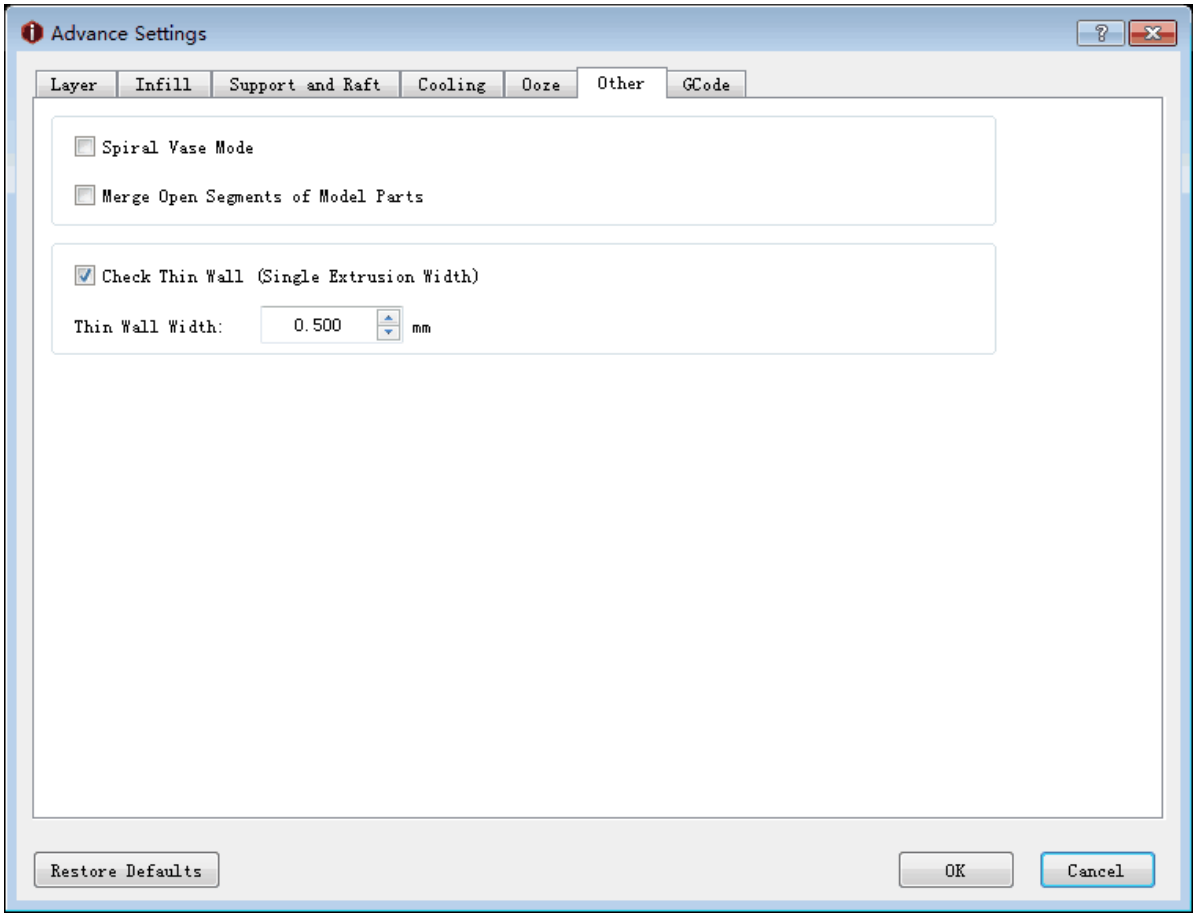
Other

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

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

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

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

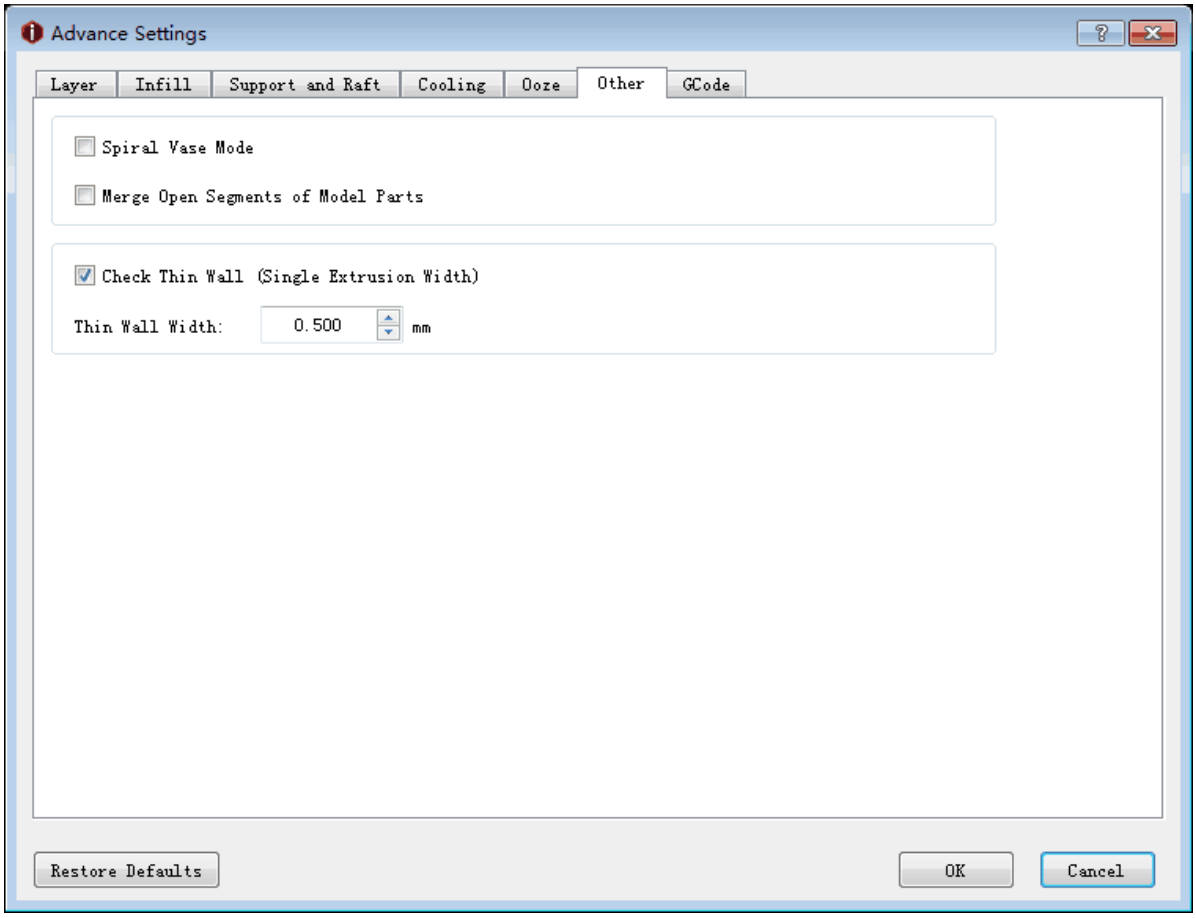


GCode

Start GCode refers to the program that printer will do or set before printing the model. Such as:

G21 refers to using metric values while printing; ***G90*** refers to using absolute positioning while printing; ***M82*** means to set extruder to absolute mode; ***M107*** means to start with the fan off; ***G28 X0 Y0*** means to move X and Y to home position at the end etc.

End GCode refers to the program that printer will do or set after finish printing the model. Such as ***M104 S0*** means to switch off the extruder heater; ***M140 S0*** means to switch off the heated bed heater, etc.

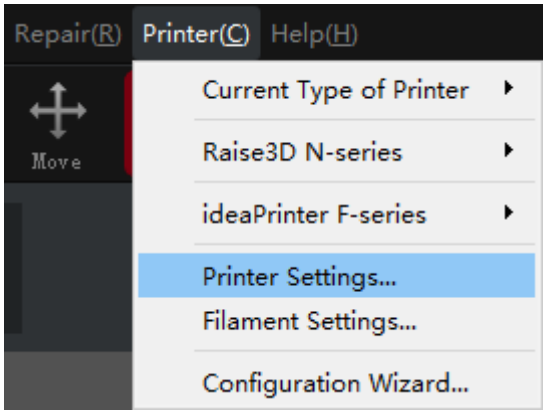


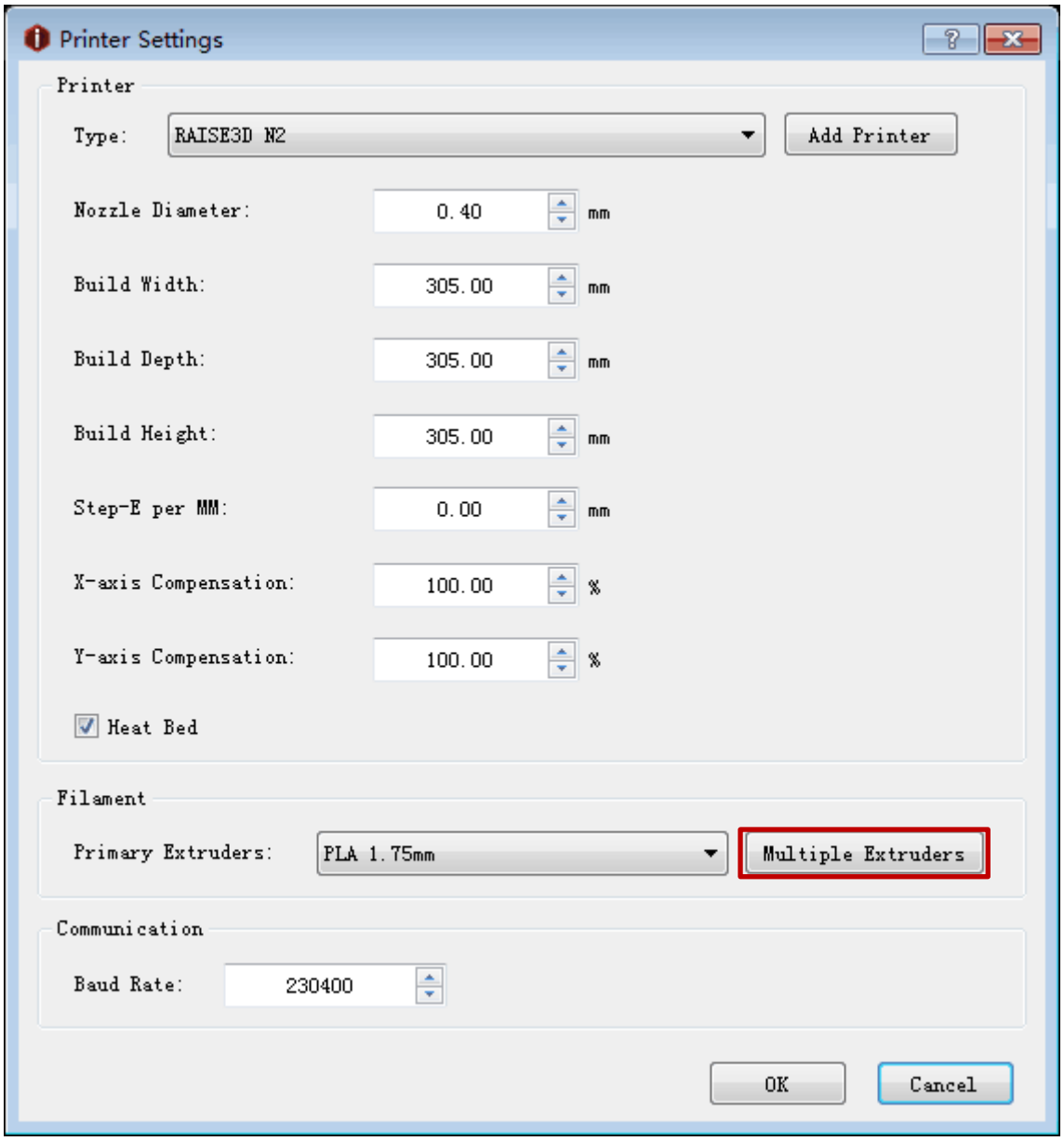
Multiple Extruders

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

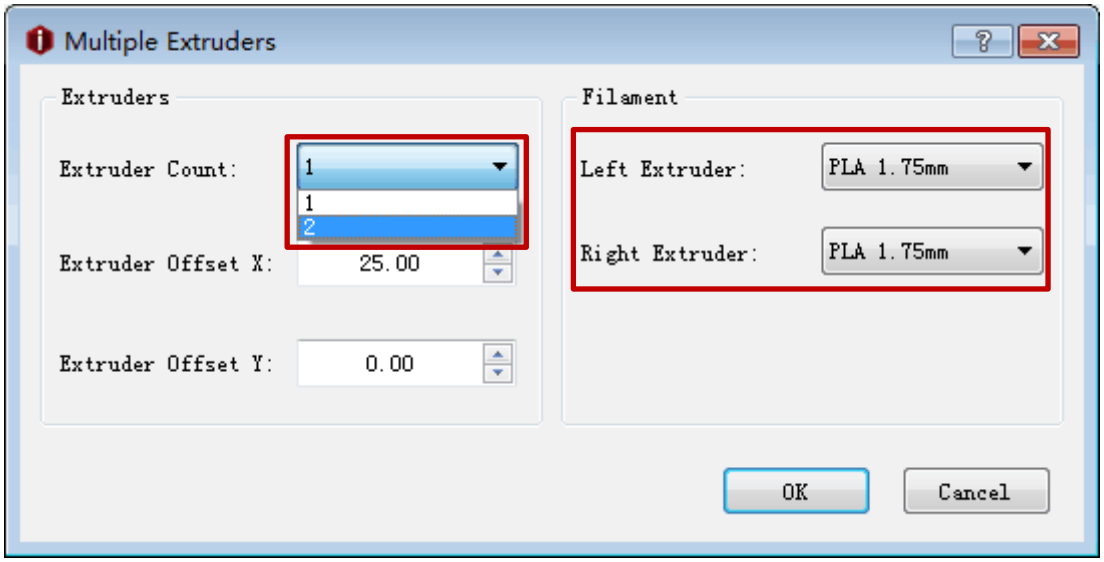
1. Set printer setting to multiple-extruder-mode.

Printer—Printer Settings—Multiple Extruders

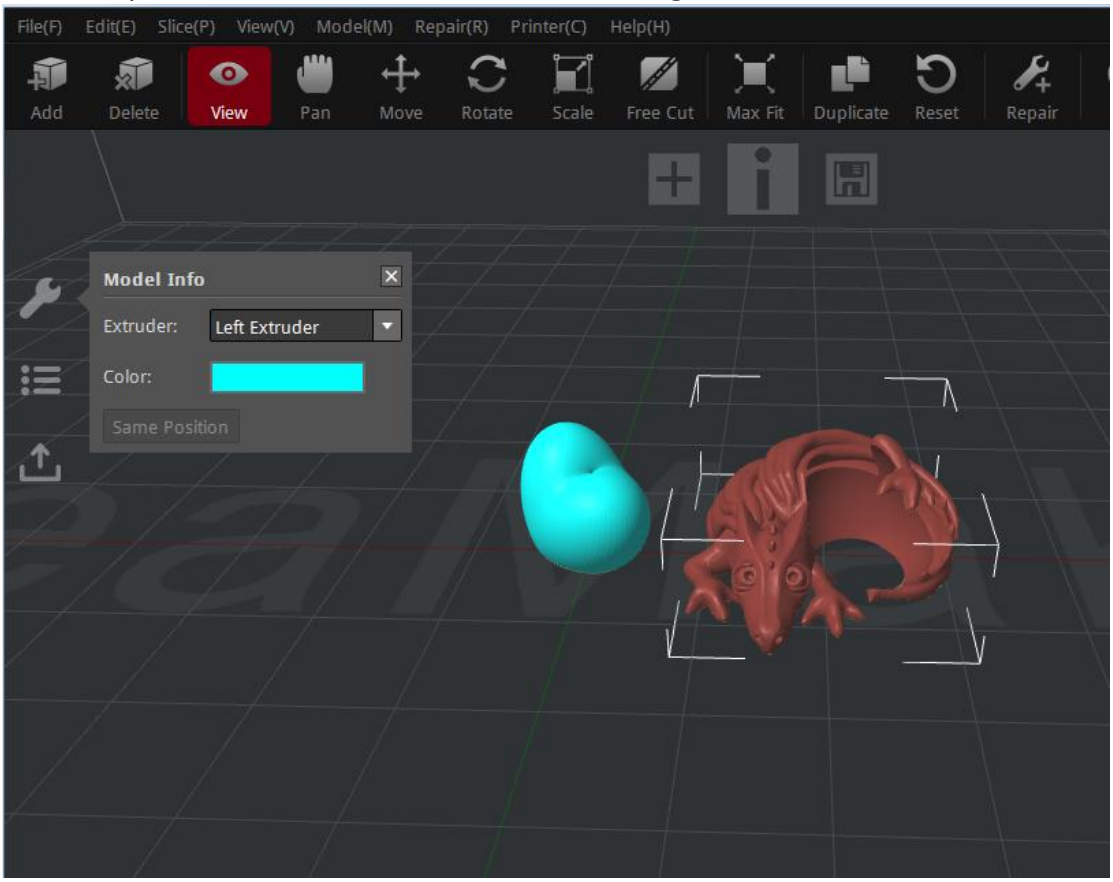


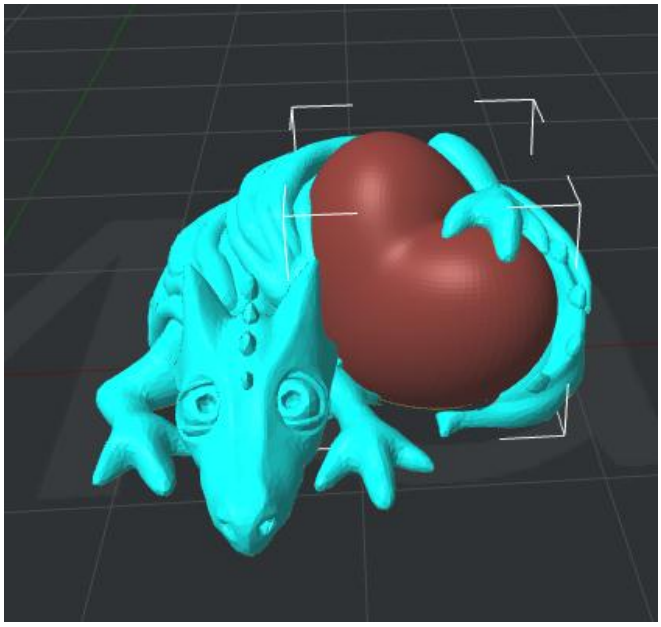


Select 2 in ***Extruder Count***, then assign filament to each nozzle in ***Filament***.

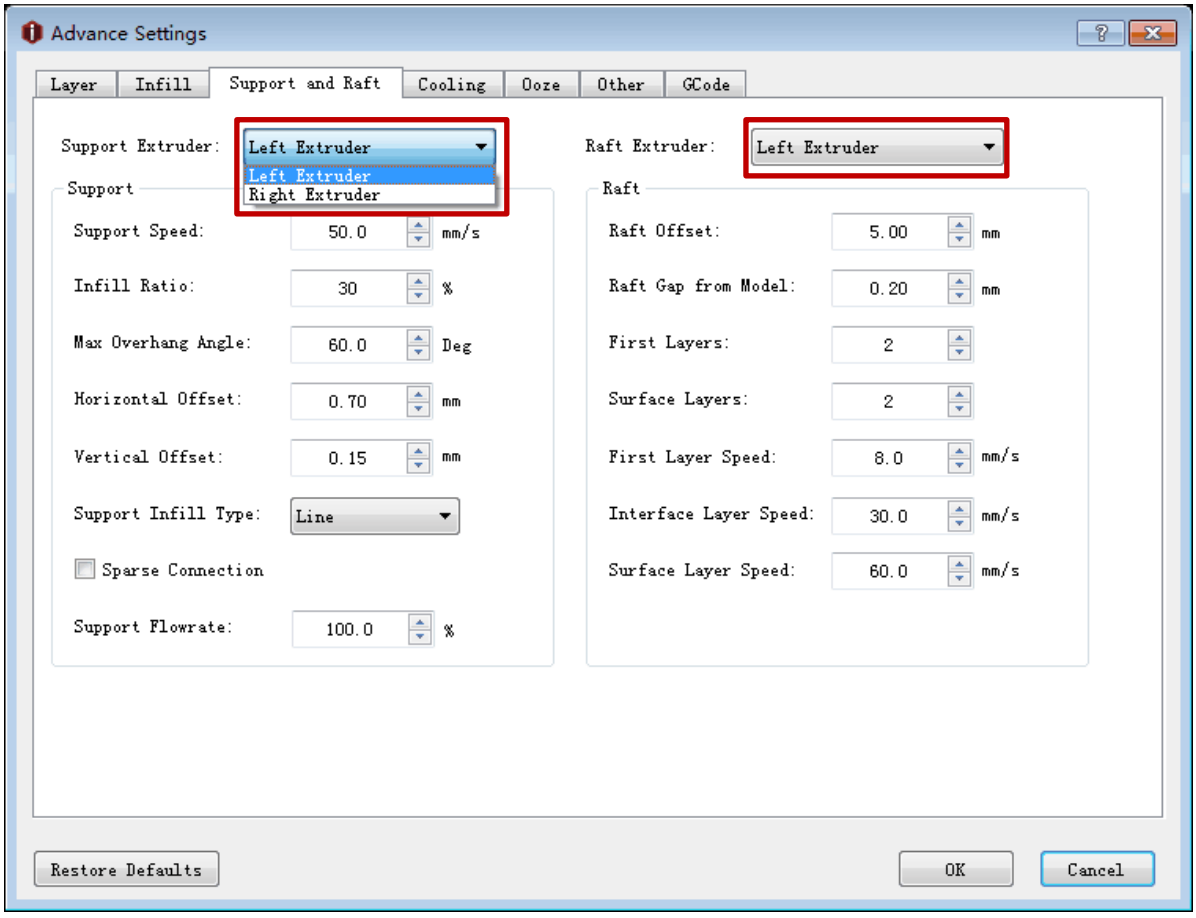


2. If you want to print a multiple-filament model, you need to set the extruders print different part of the model, and then assembling the model.

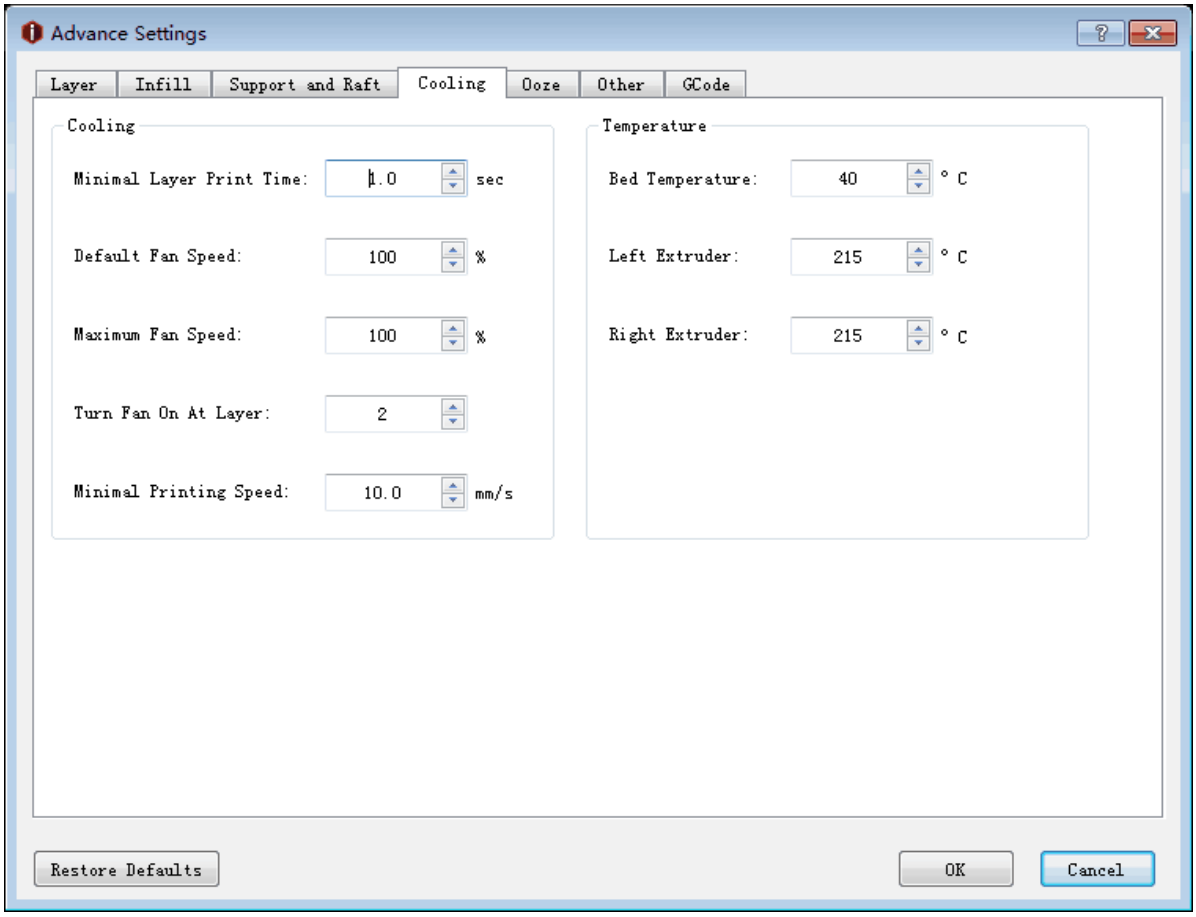




3. When slicing the model, if you use specific filament to print support (such as soluble filament) then you need to set support extruder.



4. If you use different filament while printing in multiple-extruder-mode, you need to set the extruders' temperature separately.



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

These settings as follows:

Multiple Extruders Ooze Control:

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

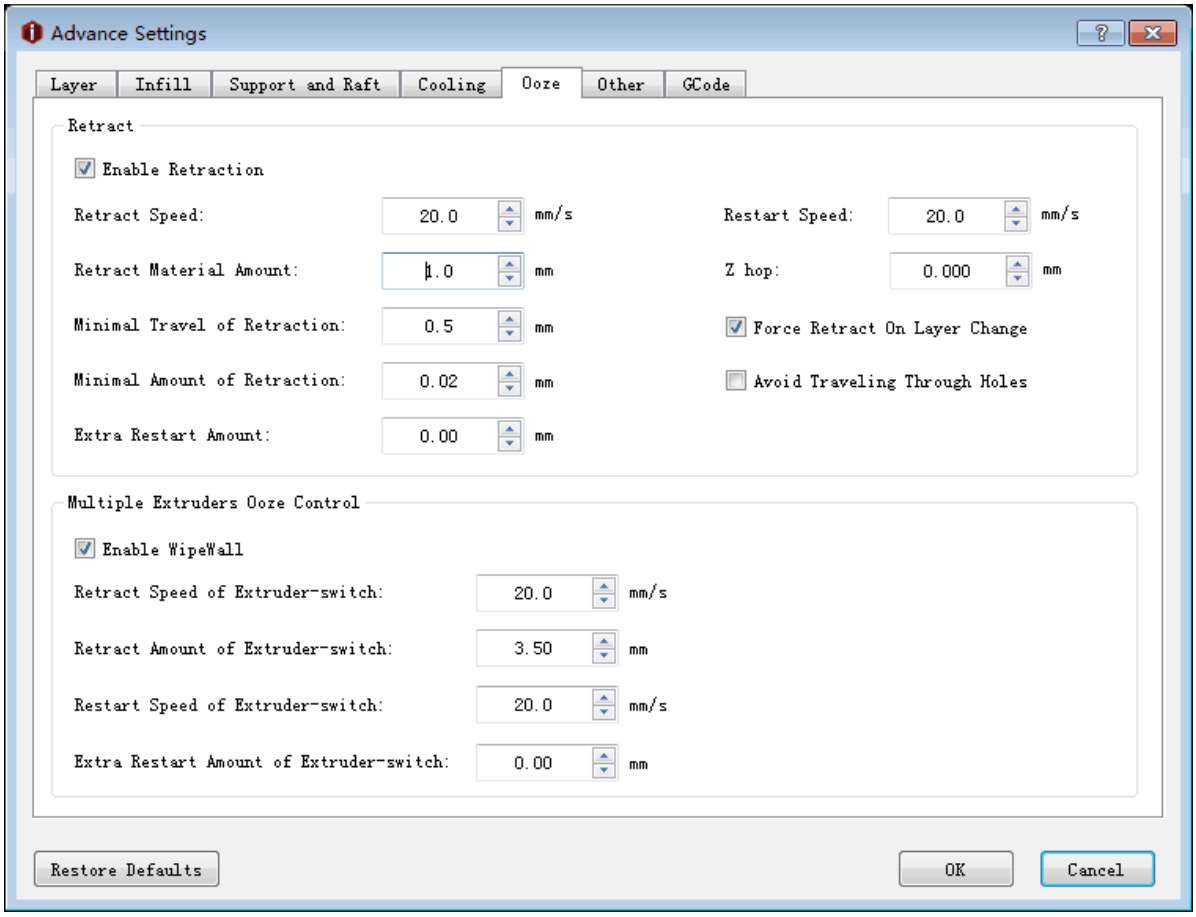
Retract Speed of Extruder-switch refers to the speed which the filament is retracted at when switching nozzle between dual-extrusio.

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

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

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

After set these five steps, you can handle multiple extruders well.

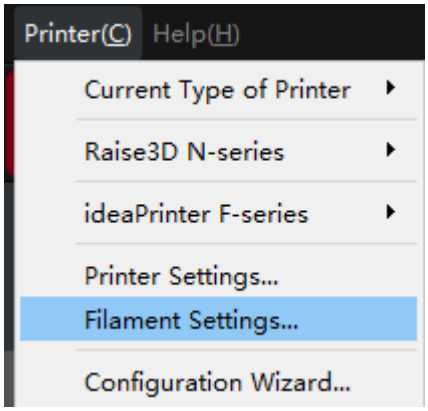


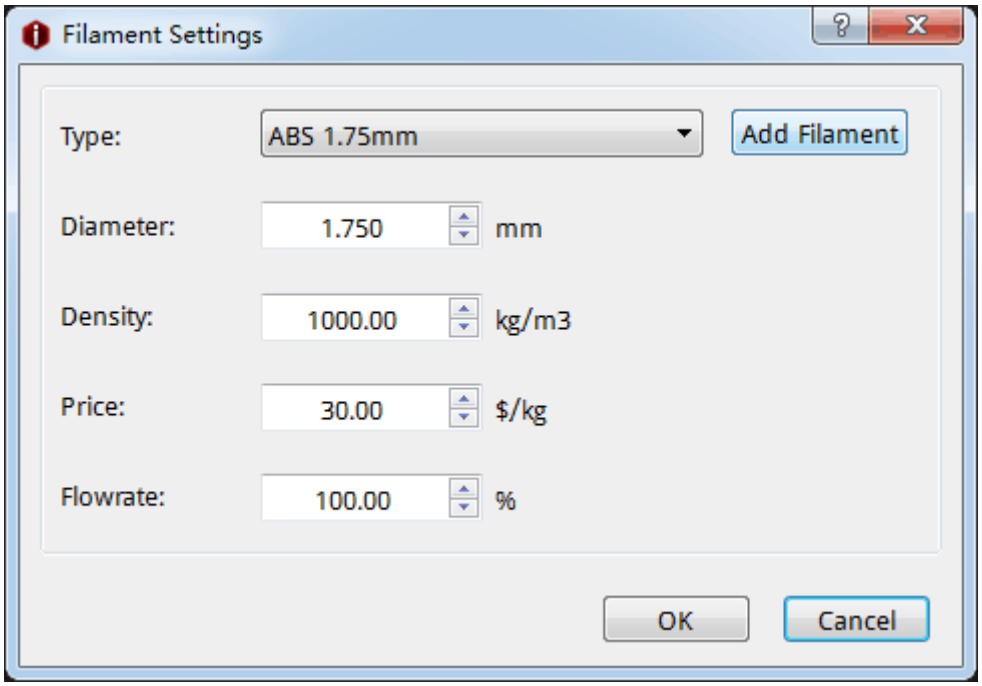
Filament Setting

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

1. Edit the existing template.

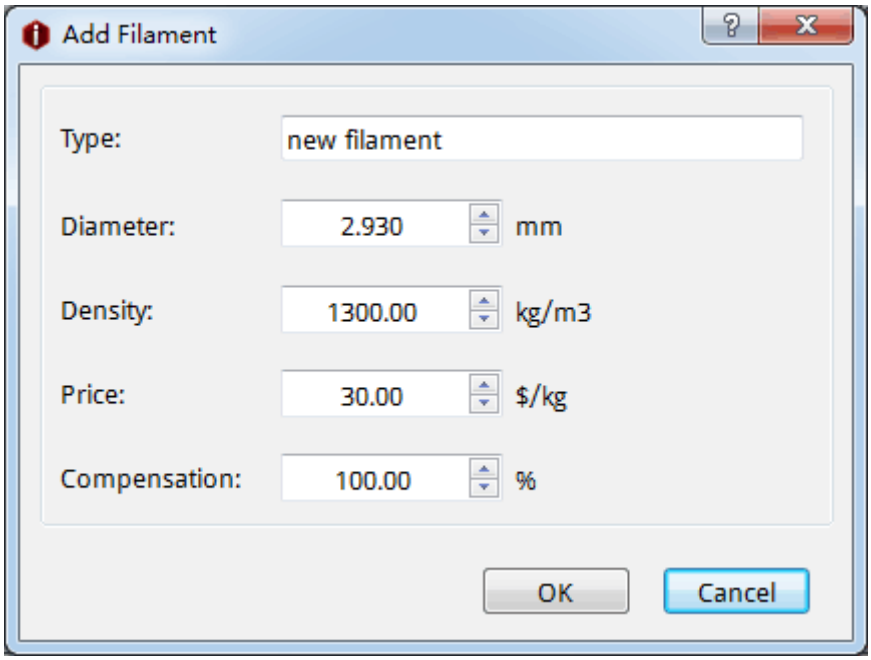
Printer—Filament Settings





In this page, you can edit the parameter of your filament. The default setting is for Raise3D own filament.

2. Add a new template.



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

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

RAISE3D

www.raise3d.com

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