ideaMaker Manual





 1.1 What is ideaMaker?	3
 2 Install ideaMaker	
 3 Let's Print! 3.1 Import .STL files 3.2 Slice the model 3.2.1 Select your printer type and filament type 3.2.2 Choose a slicing template 3.2.3 Edit the selected slicing template 3.2.4 Click "Slice" button to start slicing. 3.3 Estimated Print Result 	
 3.1 Import .STL files	
 3.2 Slice the model	9 9 10 11
 3.2.1 Select your printer type and filament type 3.2.2 Choose a slicing template	9 10 11
 3.2.2 Choose a slicing template	10 11
3.2.3 Edit the selected slicing template3.2.4 Click "Slice" button to start slicing3.3 Estimated Print Result	11
3.2.4 Click "Slice" button to start slicing 3.3 Estimated Print Result	
3.3 Estimated Print Result	
	12
3.3.1 Check the sliced model by layers	13
SIGIT CHECK the sheet model by haves	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	
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	
5.1.6 Import Template	131



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



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: <u>http://www.raise3d.com/pages/download</u>



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.

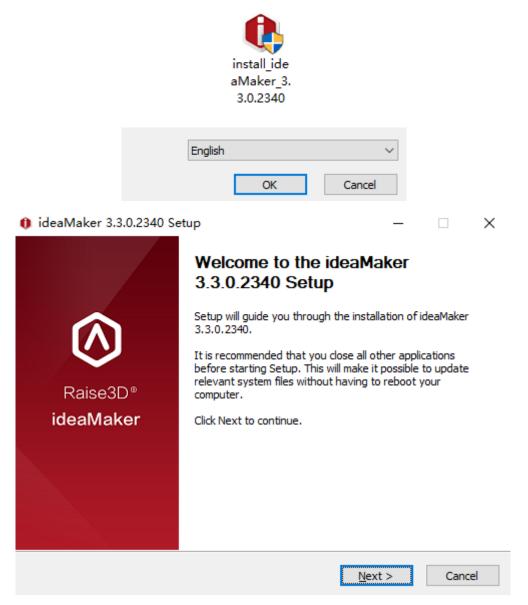


Figure 2.1: Select a language for installing ideaMaker.

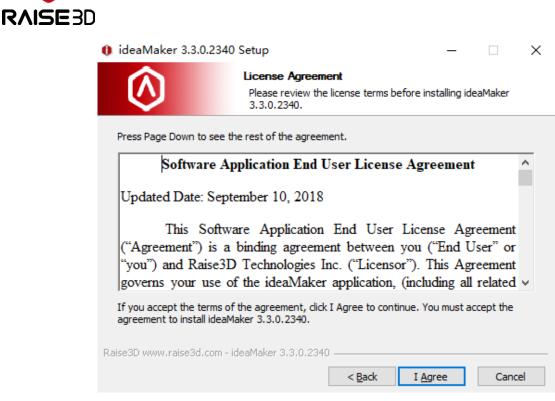


Figure 2.2: License Agreement.

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

🏮 ideaMaker 3.3.0.23	40 Setup	_		\times
\bigcirc	Choose Install Location Choose the folder in which to install i	ideaMaker 3.3	.0.2340.	
	ker 3.3.0.2340 in the following folder.To in her folder. Click Next to continue.	istall a differer	nt folder, e	dick
Destination Folder	aise 3D \ideaMaker	Brov	vse]
Space required: 147.6N Space available: 357.30				
Raise3D www.raise3d.com) - ideaMaker 3,3,0,2340	<u>N</u> ext >	Cano	cel

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.

🏮 ideaMaker 3.3.0.2340	ideaMaker 3.3.0.2340 Setup -				
\Diamond		3.0.2340 y	ou want t	0	
Check the components you want to install and uncheck the components you don't want to install. Click Install to start the installation.					
Select components to insta	II: videaMaker 3.3.0.2340 File Association Microsoft Visual C++ 2008 SP1 R Printer Driver	edistributa	able		
Space required: 147.6MB					
Raise3D www.raise3d.com - io					
	< <u>B</u> ack <u>I</u> n	stall	Cano	el	

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.

0 id	leaMaker 3.3.0.2340	Setup		_		\times
	\Diamond	Installation Compl Setup was complete				
Co	ompleted					
	Show <u>d</u> etails					
Raise	3D www.raise3d.com - id	eaMaker 3,3,0,2340 -				
			< <u>B</u> ack	<u>N</u> ext >	Cano	el

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.

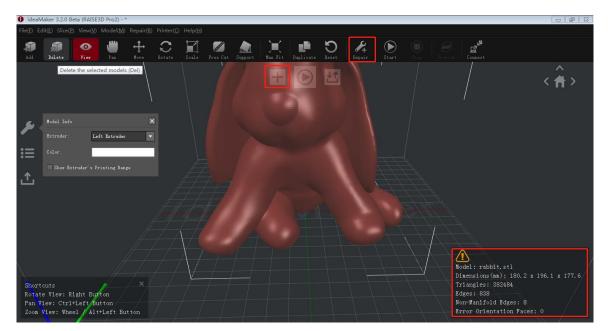


Figure 3.1: Press the "+" icon to import STL files. If it doesn't display a green tick below, press the "Repair" button to execute an auto-repair.



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.

Select Template			?
Printer Type:	RAISE3D Fro2 ideeFrinter F1	•	٢
Filament Left Extruder:	ideaFrinter F2 RAISE3D N1 RAISE3D N2 RAISE3D N2 Plus		٢
Right Extruder:	RAISE3D NI - V2 Hot End RAISE3D NZ - V2 Hot End RAISE3D NZ Plus - V2 Hot End RAISE3D Fro2 RAISE3D Fro2 RAISE3D Fro2		٩
Select Template			?
Select Template Printer Type:	RAISE3D N2 Flus - V2 Mot End	•	?
	RAISE3D N2 Flus - V2 Hot End	•	
Printer Type:	PLA 1.75mm - V2 Hot End (3 templates) ABS 3mm	•	
Printer Type: Filament Primary Extruder:	PLA 1.75mm - V2 Hot End (3 templates) ABS 3mm PLA 1.75mm yABS 1.75mm yADS 1.75mm PolyFlex 1.75mm (3 templates) PolyFlex 1.75mm (3 templates)	•	©
Printer Type: Filament Primary Extruder: Main Template Gro	PLA 1.75mm - V2 Hot End (3 templates) ABS 3mm PLA 1.75mm WABS 1.75mm PolyPC 1.75mm (3 templates)	-	\$

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.

templates)	•	\$	
templates)			
templates)			
	•	¢	
Model Extruder			
		Create	
		Duplicate	
		Edit	
		Compore	-
		compare	
		Delete	
		Import	
		Export	
Shells: 2.0			
Infill Speed: 90.0 mm/s			
	Shells: 2.0	Shells: 2.0	Create Duplicate Edit Compare Delete Import Export Shells: 2.0 Infill Speed: 90.0 mm/s

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.)

Edit Template (modified)		?)	×
Basic Template Name: Standard	- N2 - FLA		
General Infill Density: Shells:	10 • % 2.0 •		
Flatform Addition: Support: 🔊	Raft Only -		
Advanced	Restore 🔻		
Close without Saving	Save As	Save and Close	

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



3.2.4 Click "Slice" button to start slicing

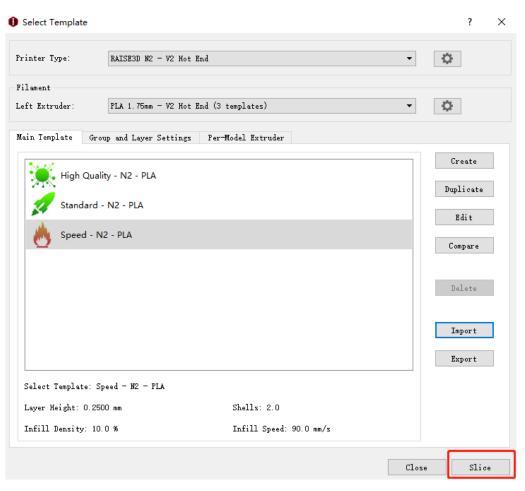


Figure 3.6: Click "Slice" to start slicing.



3.3 Estimated Print Result

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

Estimated Print Result		?	×
Prepared Print File:	rabbit.gcode		
Estimated Print Time:	26 hours, 29 min, 29 sec		
Estimated Amount:	508.8 g / 170.59 m		
Estimated Price:	¥ 15.26		
Note: Result above is or	nly for reference.		
Preview	Upload	Export	

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**.

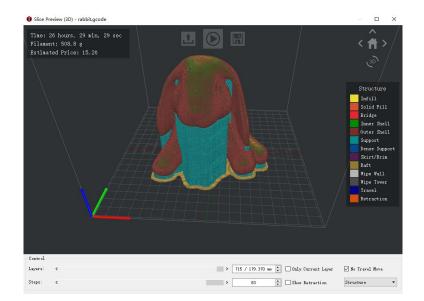


Figure 3.8: Preview how the model will be printed out.



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.

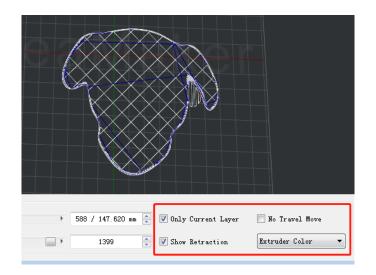


Figure 3.9: Check the retraction points with "Show Retraction". Check one layer with "Only Current Layer".

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.



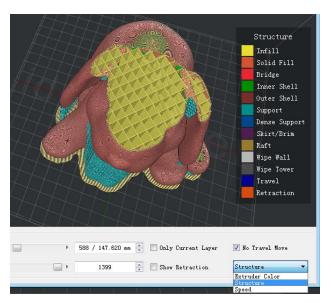


Figure 3.10: Show different structures with different colors.

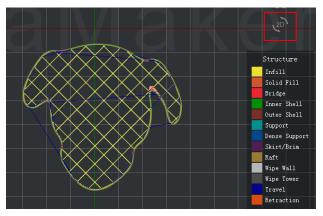


Figure 3.11: Enable "2D" to check the current layer with orthographic view.

: 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.

Estimated Print Result	? 💌
Prepared Print File:	rabbit.gcode
Estimated Print Time:	54 hours, 45 min, 18 sec
Estimated Amount:	346.4 g / 116.15 m
Estimated Price:	\$ 10.39
Note: Result above is	only for reference.
Preview	Upload Export

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.

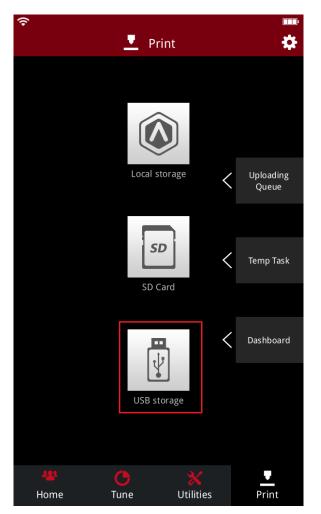


Figure 3.14: If you are using USB driver, press the "USB storage" to check the files inside. If you are using SD card, press "SD Card" to check the files inside.

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.



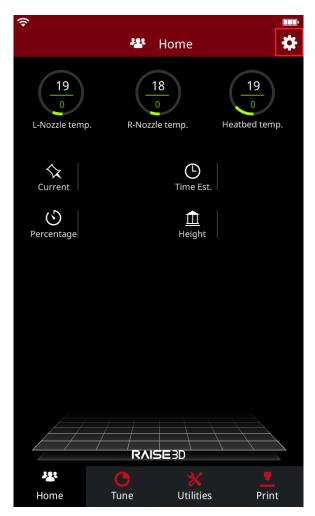


Figure 3.15: Press the gear button to enter the setting page.



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

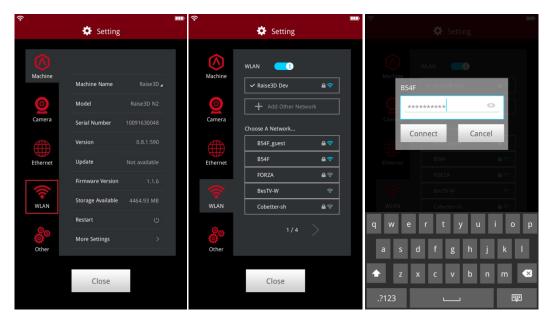


Figure 3.16: Connect the printer with your "WLAN".

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.

Estimated Print Result	? 💌
Prepared Print File:	rabbit.gcode
Estimated Print Time:	54 hours, 45 min, 18 sec
Estimated Amount:	346.4 g / 116.15 m
Estimated Price:	\$ 10.39
Note: Result above is o	nly for reference.
Preview	Upload Export



Print File				
File:	rabbit.gco	ode		
Rename to:	rabbit			
Printer:	Raise3D N2	- Raise3D@10.20.40.83 (Bookmarks)	•	Refresh
Type:	Raise3D N1	- Finn@192.168.0.117 (Bookmarks)	^	
Name:	Raise3D N2			
Address:	Raise3D N1	- J N1@10.20.40.87 (Bookmarks)	=	
	Raise3D N2	- Raise3D04@10.20.40.189 (Bookmarks)		
	Raise3D N2	- J N202@10.20.40.191		Cancel
	Raise3D N2	- J N204@10.20.40.115		$\uparrow \downarrow \uparrow$
	Raise3D N2	- JN208@10.20.40.131		$V \uparrow \uparrow$
	Raise3D N2	- J N211@10.20.40.41		7-1-1
	Raise3D N2Plu	ıs - J N203@10.20.40.161		
	Raise3D N2Plu	ıs - Raise3D@10.20.40.140		

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

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

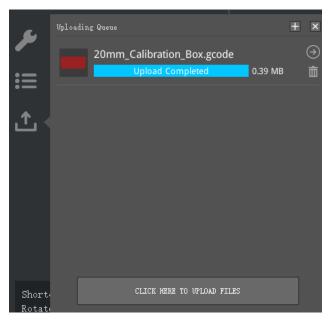


Figure 3.18: Check the uploading progress at the left side of ideaMaker.

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*.

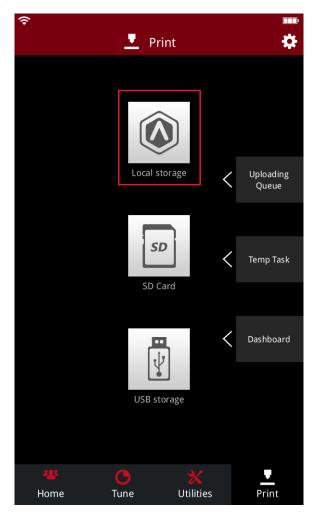


Figure 3.19: Uploaded files will be inside "Local Storage".



6. Remote connect.

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

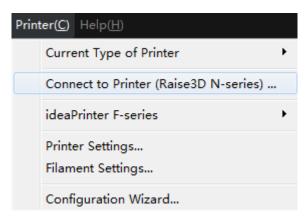


Figure 3.20: Select Printer ->Connect to Printer (Raise3D N-series).

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

0 Printer Connect					- • •
	🔁 Remot	e Connect	tion		
Bookmarks		Ð	Scanning		
Raise3D N1	Rai 192.168.0.58			×	(\rightarrow)
Raise3D N1	Raise3D 192.168.0.153			X	\Rightarrow
Raise3D N2Plus	Raise3D12345 192.168.0.55			X	(\Rightarrow)
Raise3D N2	Raise3D 192.168.0.145			X	(\Rightarrow)
	1	/ 3	>		

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



Waiting for connecting.

0 Raise3D N2 - Raise3D@192.168.0.145	
🔁 Remote Connection	
Raise3D N2 Connecting	
- Back to the Printer List	

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.

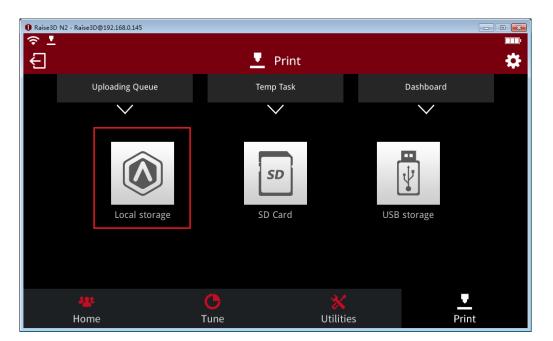


Figure 3.23: Click "Local storage" to check the uploaded files.



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



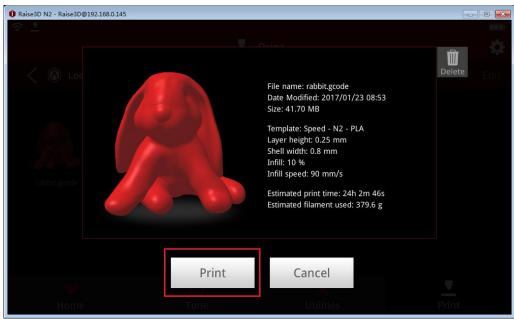


Figure 3.24: Select one file to print and check the file information.



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.

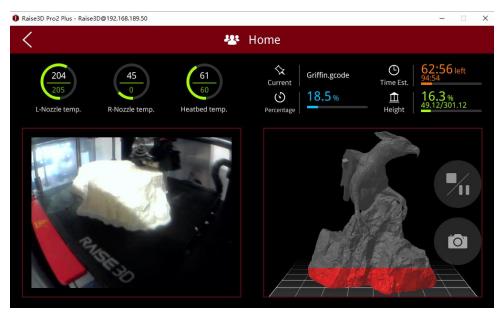


Figure 3.30: Take photos.



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.

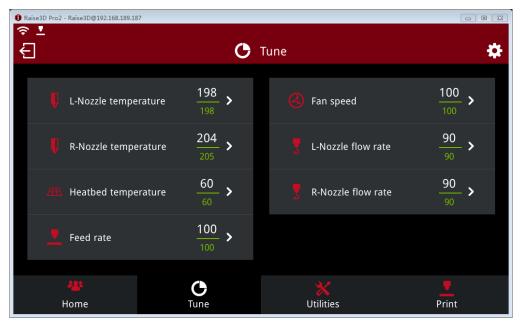


Figure 3.31: Tune.

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.

Will back to 0 after homing or reboot.



are four motors in total.) You can push the head by hands after triggering this function.

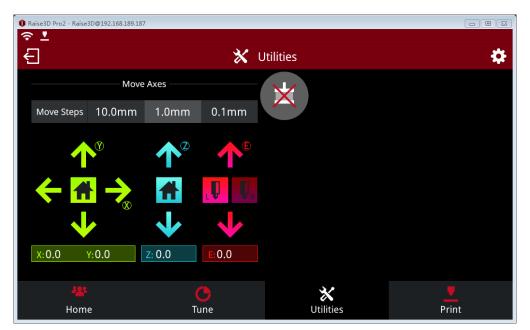


Figure 3.31: Utilities.



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**.

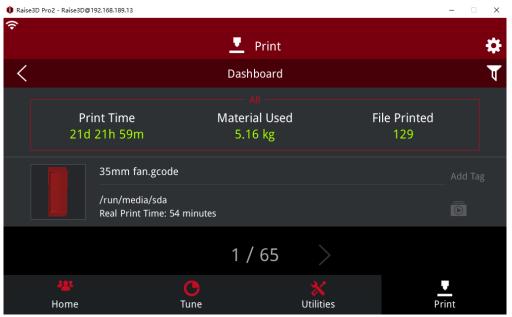


Figure 3.33 Dashboard.

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

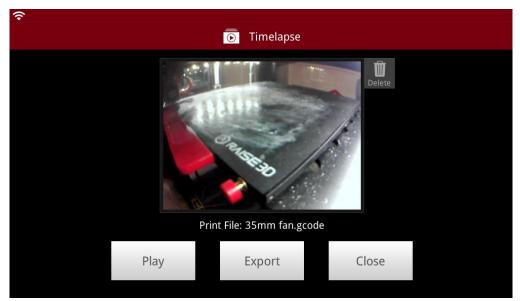


Figure 3.34: Timelapse.



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.

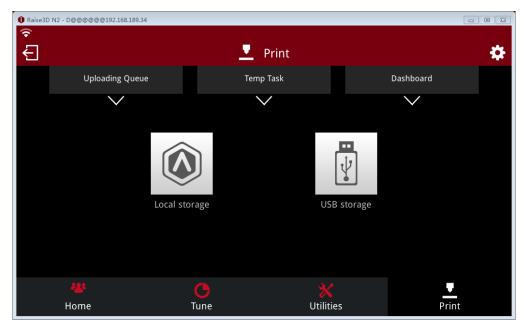
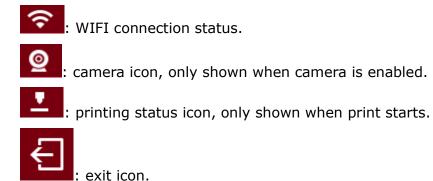


Figure 3.35: Print.



: 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.

-	se3D@192.168.189.187					23	
<u> </u>	중 ⊥ ★ Setting						
(À	Machine	O Camera	Ethernet	wlan	Other		
,	Machine Name	Raise3I	D Firr	nware Version	0-0-31		
1	Model	Raise3D Pro	2 Sto	rage Available	4.6 GB		
2	Serial Number	1010172402	2 Abo	out			
N	Version	0.9.9.50					
t	Update	Not availabl					
			Close				

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.

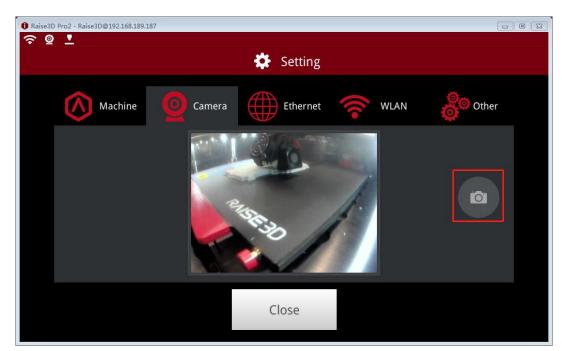


Figure 3.39: Take photo.



Raise3D Pro2 - Raise3D@192.168.189.1	187			
		🗱 Setting		
Machine	Camera	Ethernet	WLAN	Other
	Please co	onnect a camera on th	ne printer	
		Camera-2		Ø
		Close		

Figure 3.40: Camera.

Ethernet: connect printer and computer by net cable.

-	Pro2 - Raise3D@192.168.189.187					• 🗙
<u> </u>	<u>•</u>	\$	Setting			
	Machine	Camera	Ethernet	wlan	Other	
	State: Not Connected					
	IPV4 Configuration:					
	IP Address:	Subnet Mask:	Router:	DN	S Server:	
	192.168.189.187	255.255.255.0	192.168	.189.1 202	2.96.209.133	
		C	lose			

Figure 3.41: Ethernet.



WLAN: connect printer and computer by WLAN.

 ● Raise3D Pro2 - Raise3D@192.168.189.187 ○ ● <u>↓</u> 			
	🗱 Setting	5	
Machine	Camera Etherne	t 🛜 WLAN	Other
WLAN : On ✓ B54F	B ?		
	Close		

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.

I Raise3D Pro2 - Raise3D@192.168.189.187 중 @ ▲						
🗧 🖻 🔟						
Machine Q Camera	Ethe	rnet 🋜 WLAN	Cther			
Brightness		Language	English			
		Number of Nozzles	2			
	Close					

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.

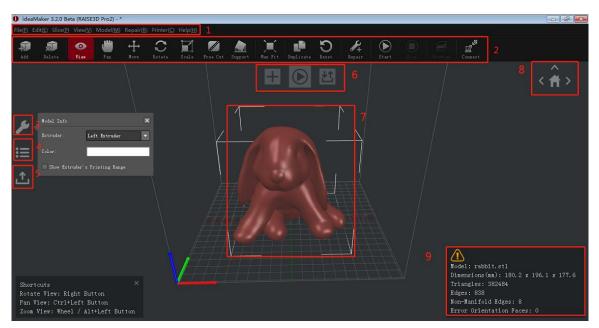


Figure 4.1: An overview over the ideaMaker interface.



4.1 Menu Bar

Menu bar includes all the operation commands and advanced setting.

1>File

File(E) Edit(E) Slice(P) Vie	w(<u>V</u>) Model(<u>M</u>	
	New(<u>N</u>)	Ctrl+N	←→Create a new empty workspace
	Open Idea File(<u>O</u>)	Ctrl+O	●→Open a .idea file
	Close(<u>W</u>)	Ctrl+W	←→Close current project
	Save Idea File(<u>S</u>)	Ctrl+S	►→ Save file
	Save Idea File As	Ctrl+Shift+S	►→Save project file as .idea
	Import Model	Ctrl+I	←→ Import a .STL, .OBJ or .3MF file
	Export Model		←→Export a .STL, .OBJ or .3MF file
	Examples	+	←→Example files
	Open Print File		$\bullet ightarrow$ Open a sliced code file, such as .idea or .gcode file
	Recent Files	+	←→ Files that have been used recently
	Exit		●→Exit ideaMaker

Note:

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



2>Edit

Edit(<u>E)</u> Slice(<u>P</u>) Vie	w(<u>V)</u> Model(<u>M</u>	
	Undo(<u>Z</u>)	Ctrl+Z	←→Revoke the last edit
	Redo(Y)	Ctrl+Y	←→Cancel the last revoke
	Cut(X)	Ctrl+X	←→Cut the selected model
	Copy(<u>C</u>)	Ctrl+C	←→Copy the selected model
	Paste(<u>V</u>)	Ctrl+V	←→Paste duplicated model
	Delete(<u>L</u>)	Del	←→Delete the selected model
	Duplicate(<u>D</u>)	Ctrl+D	←→Create a duplicate object of selected models
	Select All	Ctrl+A	←→Select all the models
	Deselect All	Ctrl+Shift+A	←→Deselect all the models
	Preferences		►Set language, shortcuts and so on

Note:

There are some special options in *preferences*.

(1) General

General	Interface	Files	Shortcuts	Confirm	and Tips	Update		
General								
Slice M	Multi-Threads	Speedup:	з		•			
Langua	ge:(*)		E	nglish	•			
Curren	cy:		¥	¢	•			
Maximum	n Duplicate Co	pies:		5	•			
🗹 Slia	ce with unsave	ed templai	es					
🗹 Save	e and restore	window st	ate					
🗹 Save	e and restore	preview s	state					
🗹 Enal	ble printer di	scovery	(*)					
🗌 Vist	ualize GCode 1	y traces	without emul	ating extr	rusion widt	h in 2D mo	de	
(*) Tal	ke effect aft	er restari	: ideaMaker.					

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*.

🏮 Edit Template		? ×
Basic Template Name: High G	Quality - Pro2 - PLA	
General Infill Density:	10 🔹 %	
Shells:	2.0	
Platform Addition:	Raft only 🔻	
Support:	None 🔻	
Advanced	Restore Restore Defaults Restore to the Last Saved	
Close without Saving	Sere As	Save and Close



		g Temperature Advanced Ooze	Other GCode
Feneral		Speed	
Layer Height:	0.1000 m m	Default Printing Speed:	50.0 🛉 mm/s
Shells:	2.0	Inner Shell Speed:	50.0 🛉 mm/s
Maximum Shells Overlap Percentage:	50 🔹 %	Outer Shell Speed:	30.0 🛉 mm/s
🗌 Print Shells in Optimal Order		X/Y Axis Movement Speed:	100.0 🔹 mm/s
first Layer Settings		Z Axis Movement Speed:	25.0 🛉 mm/s
First Layer Height:	0.3000 💼 mm	Other	
First Layer Speed:	15.0 🛉 mm/s	🗌 Spiral Vase Mode	
First Layer Flowrate:	100.0 🔦 %	🗌 Print External Shells First	
ayer Start Point		🗌 Print Parts in Sequential Order	
Layer Start Point Type:	Nearest 🔻	Minimal Segment Length:	0.012 🗼 mm
Fixed Layer Start Point X:	0.00 mm	Dimensional Compensation	
Fixed Layer Start Point V:	0.00 mm	XV Size Compensation:	0.00 🗭 mm
Seam Hiding			
Place Seam on: N	one 🔻		
Place seam on the nearest corne	r as possible		
Restore Defaults			

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

39/177



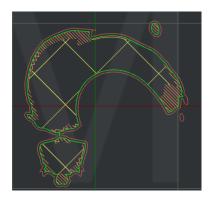


Figure 4.5: Enabled this function

(2) Interface

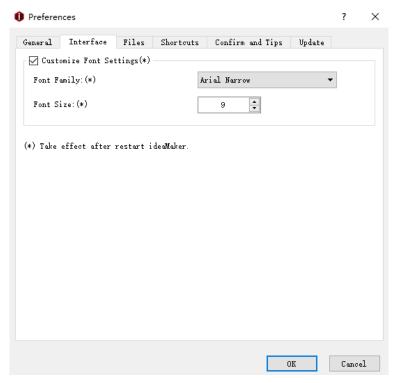


Figure 4.6: The Interface tab of Preference.

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.



Preferences				?	×
General Interface	Files Shortout	s Confirm and Tip	s Vpdate		
STL Import					
🗌 Weld Vertices					
Welding Threshold:	0.030	mm			
Import options are	only applied to newl	ly imported models.			
Models					
🗹 Place models on	platform after rotat	ting, scaling, and mi	rroring		
🗸 Automatically po	osition imported mode	els			
	-				

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.



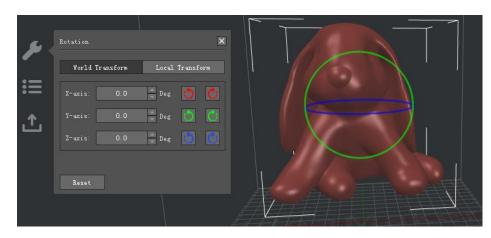


Figure 4.8: The model is floating in mid-air before rotating.

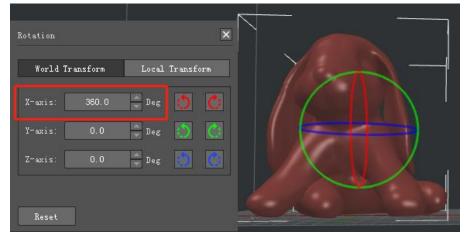


Figure 4.9: The model is placed on platform after rotating.

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

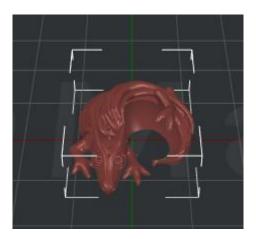


Figure 4.10: Enable the function of "Automatically position import models".



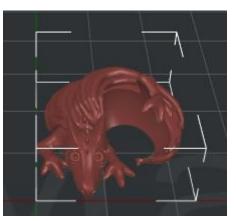


Figure 4.11: Unable the function of "Automatically position import models".

(4) Shortcuts

eral Interface	Files Shortcuts	Confi	rm and Tips 👘 Update	
rack View				
lotate View:	None	• +	Right Button 🔹	
'an View:	Ctrl	+	Left Button 🔹	
oom View:	Alt	+	Left Button 🔹	
'ine Control:	Shift	• +	Left Button 🔷 🔻	
	Shift	_		
		• +	Right Button 🔻	
Slice Navigation: ote: Left Button onl;				

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

Preferer	nces				?	×
General	Files	Shortcuts	Confirm and Tips			
— Confirm 🗹 Cont		ng with Group	o and Layer Settings			
Con:	firm model	. deletion				
Tips						
🗹 Shor	w tips for	• Model Mirro	r function			
Shor	w tips for	the visibil:	ity of shortcuts tip:	5		
🗹 Shor	w tips for	obsolete ter	nplates management			
				OK	Cance	1

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**.



🕕 ideaN	Naker 💽
1	The Group and Layer settings will override the slice settings in the main template you have selected.
	Do you want to continue?
	Do not ask me again
	Yes No

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**.

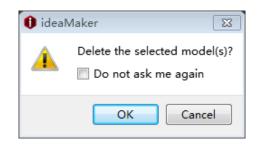


Figure 4.15: Confirm model Deletion.

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**.

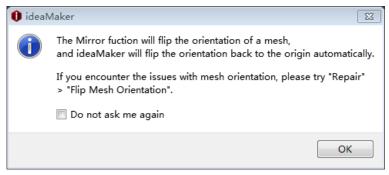


Figure 4.16: Show tips for Model Mirror Function.



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

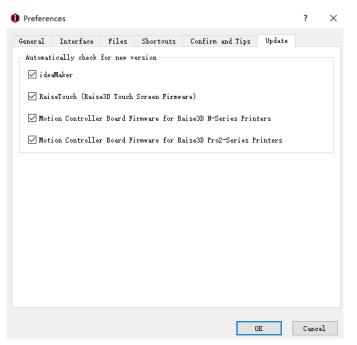


Figure 4.17: The Update tab of Preference.

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

Slice(P) View(V) Model(M) Rep	air(<u>R</u>) Printer((
Start Slice(P)	Ctrl+P	←→Slice model to get .gcode file and .data file
Stop Slice	Ctrl+Shift+P	←→ Abort slicing
Slice Report	Ctrl+R	← After slicing, you can get some estimated information such as print time, filament amount, etc.
Slice Preview	Ctrl+G	← After slicing, you can get a simulation of your print for pre-check
Upload Current Print File	Ctrl+U	←→Upload the sliced file to printer
Export Current Print File	Ctrl+E	←→Export the sliced file to computer
Manage Templates	Ctrl+M	←→Edit the slicing template even without importing a model
Compare Templates	Ctrl+T	Compare the selected slicing templates

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.

emplate 1			Template 2			
From ideaMaker	From Local Disk		From ideaMaker	From Local Disk		
Printer:	RAISE3D Pro2	•	Printer:	RAISE3D Pro2		•
Filament	ABS 1.75mm	-	Filament.	PLA 1.75mm		•
Tildinate.	700 1. Tolini		TTT dirette.	The T. Folian		
Template:	Vîtra High Quality - Pro2 - ABS 🔹 👻		Template:	Vltra High Quality - Pro2 - PLA		•
ain Template	Group and Layer Settings					
						_
Setting		Ultra High Quality - Pr	ro2 - ABS	Ultra High Quality - Pro2 - PLA		
Infill Den	sity	15 %		10 %		
 Platform Add 	ditions					
Raft Gap	from Model	0.15 mm		0.20 mm		
 Cooling 						
Maximun	n Fan Speed	50 %		100 %		
Fan Spee	d Control List	1;0;2;50		1;0;2;100		
 Temperature 	•					
Heated B	Bed Temperature	100 °C		60 °C		
Left Extru	ider Temperature	250 °C		205 °C		
Right Ext	ruder Temperature	250 °C		205 °C		
Inactive (Cooling Temperature (Left)	225 °C		180 °C		
Inactive (Cooling Temperature (Right)	225 °C		180 °C		
Inactive H	Heating Temperature (Left)	245 °C		200 °C		
Inactive H	Heating Temperature (Right)	245 °C		200 °C		
Compare Temp	lates				?	
mplate 1			Template 2			
From ideaMaker	From Local Disk		From ideaMaker	From Local Disk		
Click "Load" bu	ntton to load a new template.		Printer:	RAISE3D Pro2		•
			Filament:	PLA 1.75mm		•
Load			Template:	Vltra High Quality - Pro2 - PLA		•

Figure 4.18: Compare Templates.



4>View

View	Model(<u>M</u>) Repair(<u>R</u>)	Printer(C
	Home View Top View Front View Back View Right View Left View	Views of model
۲	Perspective View Orthographic View	 Check the model with perspective view Check the model with orthographic view
 Image: A start of the start of	Show Edges Show Wireframe Show Color for Invalid Me	 →Show the edges of model →Show the wireframe of model shes →Show defective model in red
\checkmark	Show Tips and Shortcuts	←→Show the tips and shortcuts in the interface

Note:

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



5>Model

View		
	Shift+V	←→ Click left button and move mouse to rotate the view
Pan	Shift+N	←→ Click left button and move mouse to pan the view
Move	Shift+M	Click left button and move mouse to move model
Rotate	Shift+R	Click left button and move mouse to rotate model
Scale	Shift+B	Click left button and move mouse to change size of model
Free Cut	Shift+C	←→Cut the model in two parts
Support Structure	Shift+U	Create your own support for the model
Cross Section	Shift+O	← Check the model's feature by layer with cross section
Mirror	•	←→Change model into a mirror shape
Center	Shift+E	←→Move model to the center of printing area
Lay Flat	Shift+L	Select a surface to lay the model flat on build platform
Auto Fit to Build Volume	Shift+F	←→Scale model into its maximum size which is able to be printed
Reset All Reset	•	\rightarrow Reset settings of the model
Put All Models on Platform	Shift+T	←→Check that what is on the build platform
Auto Ungroup Merge Selected Models Align Selected Models Arrange All Models	Shift+A	 Automatically ungroup the selected print Combine the selected models to one in order to keep their relative location to each other when rotating or scaling or moving Aligned models which have been selected Arrange models automatically

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.

s.	Cross Section		×		
	🗸 Enable			(<u> </u>	{ \
:=	○ X: 48.68	in m ▼	Reverse		
-	O Y: 44.51	mm	Reverse	H	1 also
⊥	O Z: 94.00	mm	Reverse		94.0 mm
	Control:			/// ` }	
	🗸 Show Plane	Inches			8 181
	Reset			$77N_{\pm 1}$	
		- H - F	1 1 1		

Figure 4.19: Cross Section in the Z axis direction.

			$\overline{1}$	
s.	Cross Section	×		
	🗸 Enable		-	-(
≔	○ X: 48.68	m Reverse		
	OY: 44.51 ▲	m 📃 Reverse	F	617
企	O Z: 68.00 [▲] / _→ [¬]	m 🔽 Reverse	La Ma	
	Control:		17A	68.0 mm
	✓ Show Plane Ind	ches	6,444	1/h
	Reset		AF/A	<u>74</u>

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



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

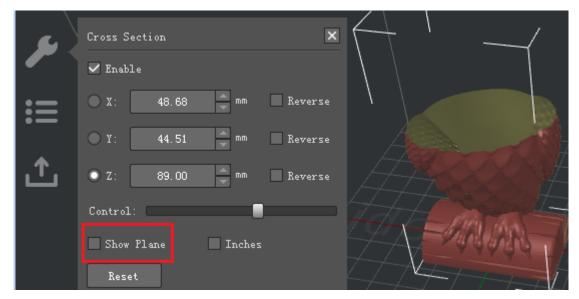


Figure 4.21: Cross Section without "Show Plane".

Inches will select the unit between inches or mm.

_	Cross Section 🗙	(
	🛩 Enable	
≔	🔿 X: 1.92 🚔 inch 🗖 Reverse	
	🔿 Y: 🚺 1.75 🚔 inch 🗖 Reverse	A
企	• Z: 3.50 🚔 inch Reverse	
	Control:	HAT FINNER
	Show Plane 🖌 Inches	LA THERE
	Reset	
		777777

Figure 4.22: Cross Section with "Inches".



Note:

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

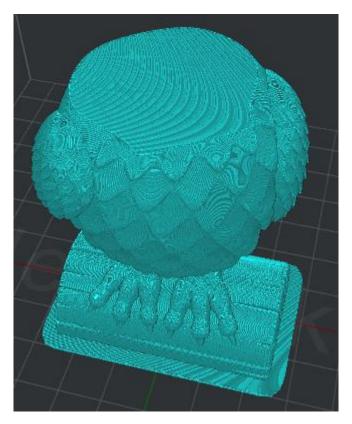


Figure 4.23: The slice preview with "Cross Section".



Lay Flat refers to selecting a surface to lay the model flat on the build platform. *Face Index* refers to the face you selected.

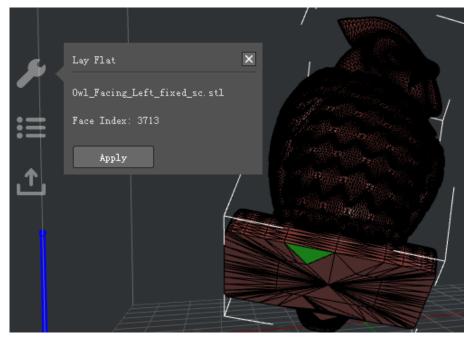


Figure 4.24: Click "Apply" to lay the model flat on the platform.

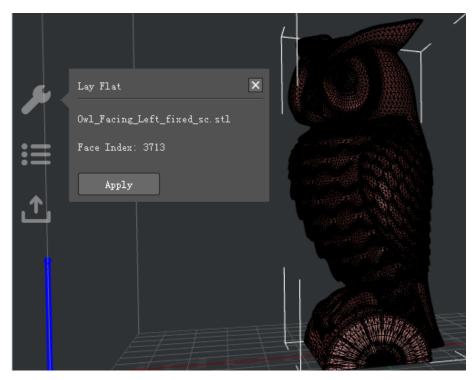


Figure 4.25: The model after "Lay Flat".



6> Repair

Rep	air(<u>R)</u> Printer(<u>C</u>) Help(<u>H</u>)	
	Remove Duplicate Faces Remove Isolated Faces Fix Face Orientation Flip Mesh Orientation Fix Holes	► Repair the selected model item by item
	Auto Repair	←→Automatically repair all the faults of the selected model(s)

Note:

You can also find auto repair icon in tool bar.

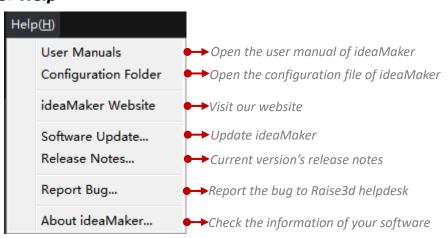
Print	ter(<u>C)</u> Help(<u>H</u>)	
	Current Type of Printer	←→Select the type of your printer
	Connect to Printer (Raise3D N-series)	Set a remote connection with your N-series printer(s)
	ideaPrinter F-series	←→Set a connection with your printer(s) via USB cable
	Printer Settings	←→ Set your printer's parameters
	Filament Settings	\bullet Set the filament that your printer is using
	Configuration Wizard	←→ Wizard for set configuration

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.





54 / 177



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.



E: Delete the selected model.



W : 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).

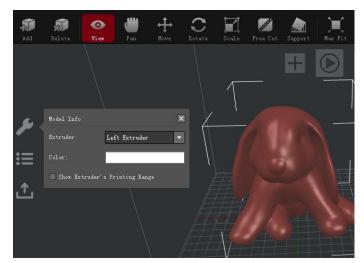


Figure 4.27: Check models with different extruders in different colors.



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



Pan : When you enable this button, you can pan the view by holding the left button and dragging your mouse.



Nove: 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.

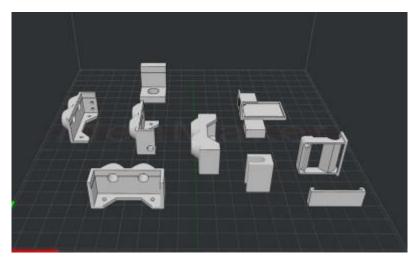


Figure 4.28: Unable the function of the "Arrange All Models".



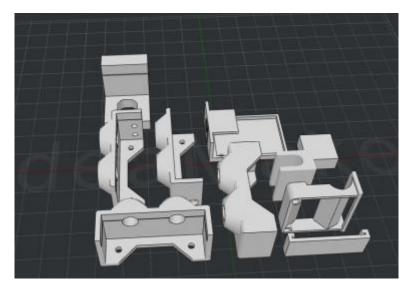


Figure 4.29: Enable the function of the "Arrange All Models".

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

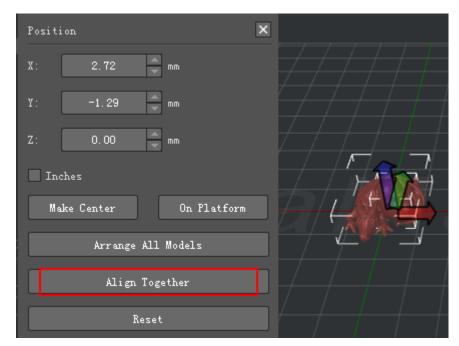


Figure 4.30: You can drag the selected model freely in X & Y directions with left-click. And click the blue arrow to drag the model in Z direction.



C Rotate

Rotate : 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.

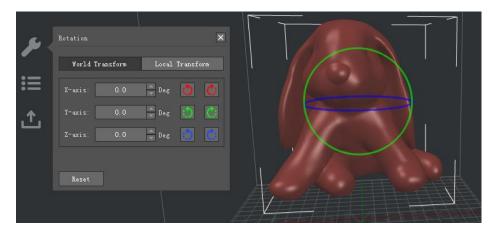


Figure 4.31: You can use both the color rings and left values to rotate the model.



: 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.



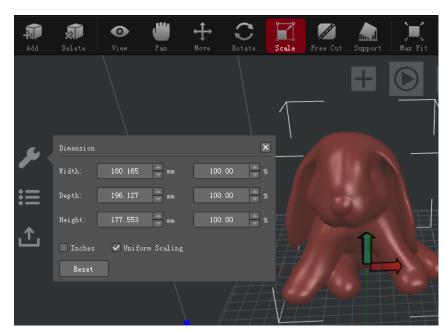


Figure 4.32: You can use both the color arrows and the spin boxes in the Dimension panel to scale the model.



Free Cut : When you enable this button, there will be a plane which you can move to cut the

model into individual pieces.

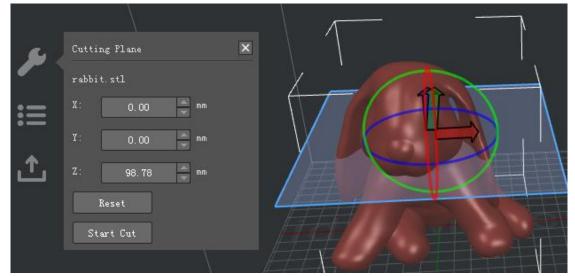


Figure 4.33: Click "Start Cut" to cut the model into individual pieces.

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



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

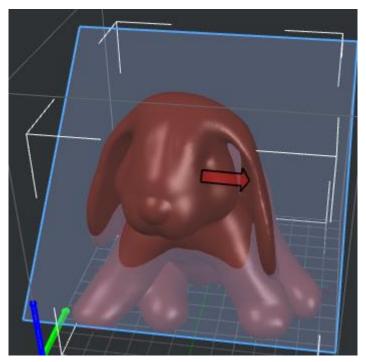


Figure 4.34: Click red arrow to move the "Cutting Plane".

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

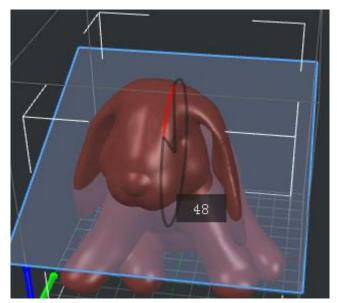


Figure 4.35: Click red circle to rotate the "Cutting Plane".



Option 3: Input the exact value.

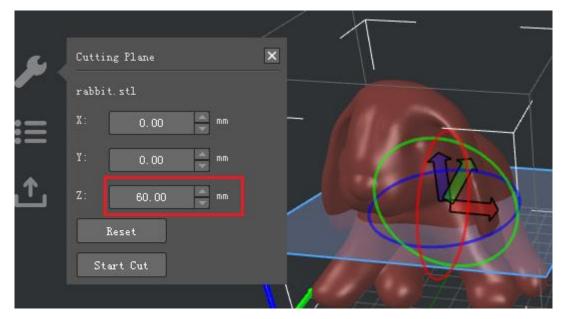


Figure 4.36: Input values to move the "Cutting Plane".

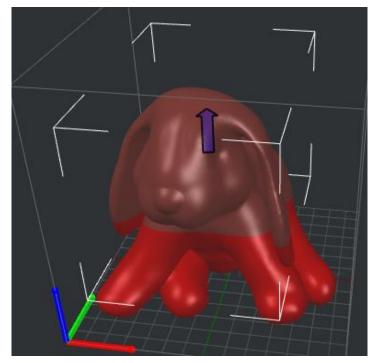


Figure 4.37: The selected model has been cut into two parts.



Support : 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**.

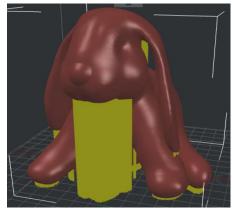


Figure 4.38: Set "Pillar Size" to be 1mm.

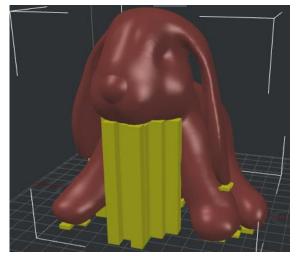


Figure 4.39: Set "Pillar Size" to be 10mm.

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.



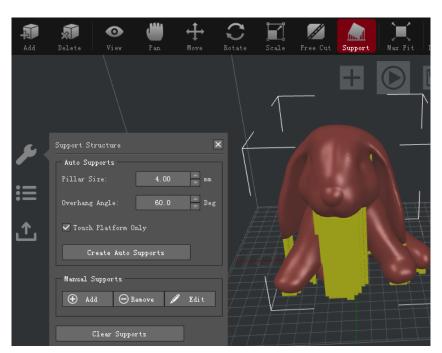


Figure 4.40: You can set the "Pillar Size" and "Overhang Angle" first. And try "Create Auto Supports". Then adjust the support with "Manual Supports".

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.

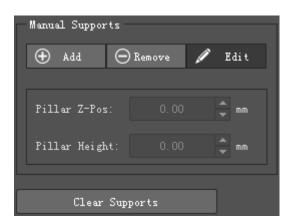


Figure 4.41: These two options will only appear after you click the "Edit" button.





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



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



: Revoke all the settings of the selected model.



E: Automatically repair model's defects.



: Start to slice the model.



: Abort slicing.



Preview : Show the sliced file in layers.



Connect : Connect to the printer with your computer via WLAN.



4.3 Operation property

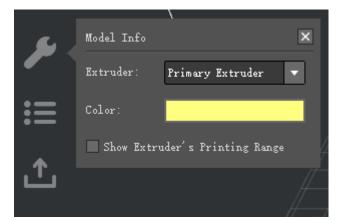


Figure 4.42: The information shown here depends on which function you are using.

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



4.4 Model list

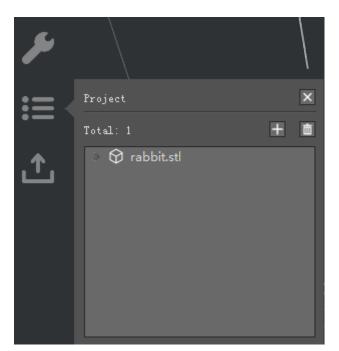


Figure 4.43: You can also add or delete models here.

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.

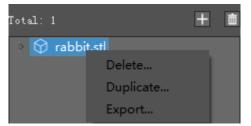


Figure 4.44: Right-click the blue section will display Delete, Duplicate, Export.

Delete refers to the model will be deleted via this button.

66 / 177



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

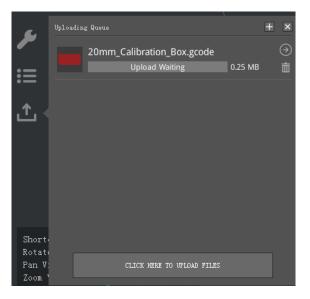


Figure 4.46: ideaMaker also can upload G-Code files sliced by other slicing software. Press "+" button to find the files you want to upload to your N-series printer from your computer.

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**.

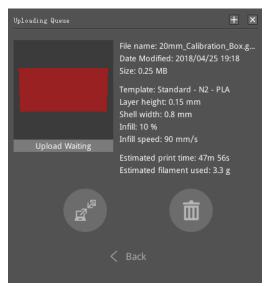


Figure 4.47: Detail information of the uploading task



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

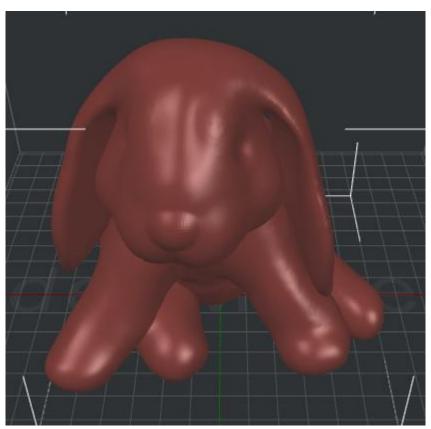


Figure 4.48: Preview your model.

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

Model: rabbit.stl Dimensions(mm): 180.2 x 196.1 x 177.6 Triangles: 382490 Edges: 834 Non-Manifold Edges: 0 Error Orientation Faces: 0

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.

Select Template			?	×
Printer Type: RAISE3D N2 - V2 Ho	t End	•	0	
7ilament Primary Extruder: PLA 1.75mm - V2 Ho	t End (3 templates)	T	¢	
Main Template Group and Layer Setting	s Per-Model Extruder			
High Quality - N2 - PLA			Create	
Standard - N2 - PLA			Duplicate Edit	
Speed - N2 - PLA			Compare	
			Delete	
			Import	
			Export	
Select Template: Standard - N2 - PLA				
Layer Height: 0.1300 mm	Shells: 2.0			
Infill Density: 10.0 %	Infill Speed: 30.0 mm/s			
		Close	Slic	e

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.

ain Template	Group and I	ayer Settings Per-Model Extru	ider	_	Create
📜 🛈 Cre	ate Template	,	?	×	
💋 Templa	ate Name:	New Template1			Duplicate
Copy	Template Fro	m			Edit
🕐 Print	er Type:	RAISE3D N2 - V2 Hot End		•	Compare
Filam	ent Type:	PLA 1.75mm = V2 Hot End		-	
Templ	.ate:	Standard - N2 - PLA		•	Delete
					Import
		OK	Cancel	·	Export

Figure 5.2: Create a new template.

Main Template	Group and Layer Settings	Per-Model Extruder	
Lieb	Quality - N2 - PLA		Create
2 TT			Duplicate
	dard - N2 - PLA		Edit
Spee	d - N2 - PLA		Compare
рето	i de la constante de		
			Delete
			Import
			Export

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.

Select	Template		? ×
Printer	Type: RAISI	3D N2 - V2 Not End -	¢
Filament Primary	_	.75mm - V2 Hot End (3 templates) 🗸	٥
Main Tem	mplate Group and	Layer Settings Per-Model Extruder	
	🚺 Duplicate Templ	ate ? X	Create
	Template Name: - Copy Template Fro	New Templatei	Duplicate
Ć	Printer Type:	RAISE3D N2 - V2 Hot End -	Compare
	Filament Type:	PLA 1.75mm - V2 Hot End	Delete
	Template:	Speed - N2 - PLA 💌	Detere
		OK Cancel	Import
Select	: Template: Speed -	N2 - PLA	Export
Layer	Height: 0.2500 mm	Shells: 2.0	
Infill	. Density: 10.0 %	Infill Speed: 90.0 mm/s	
		cl	ose Slice

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.

Select Template				?	
rinter Type:	RAISE3D N2 - V2 Hot	End	-	¢	
ilament rimary Extruder:	PLA 1.75mm - V2 Hot	End (3 templates)	•	0	
ain Template Gro	oup and Layer Settings	Per-Model Extruder			
High Qual	lity - N2 - PLA			Create Duplicate	
Standard	- N2 - PLA		Г	Edit	
beed - N	N2 - PLA		L	Compare	
				Delete	
				Import	
				Export	
-	itandard - N2 - PLA	el 11 - e e			
Layer Height: 0.13 Infill Density: 10		Shells: 2.0 Infill Speed: 30.0 mm/s			
			Close	Slice	e

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



🚺 Edit Template (modified)		?	×
Basic Template Name: High Qual	ity - Pro2 - PLA		
General Infill Density: 🔊	15 🔺 %		
Shells:	2.0		
Platform Addition:	Raft Only -		
Support:	None 🔻		
Advanced	Restore 🔻		
Close without Saving	Save As	ave and Clos	se

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".

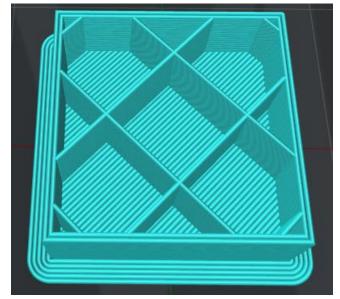


Figure 5.8: Printed with 2 shells, 10% fill density and Brim.



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.

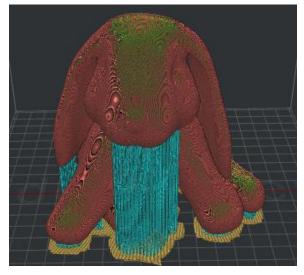


Figure 5.11: Printed with supports of "Touch Platform Only".

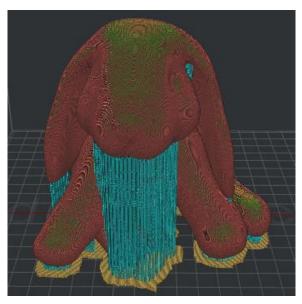


Figure 5.12: Printed with supports of "All".



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.

Edit Template (modified)	8
Basic Template Name: Standar	d - N2 Plus - PETG
General Infill Density:	15 🚔 %
Shells:	2.0
Platform Addition:	Raft only -
Support:	None 🔻
Advanced	Restore Defaults
Close without Saving	Save As Save and Close

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.

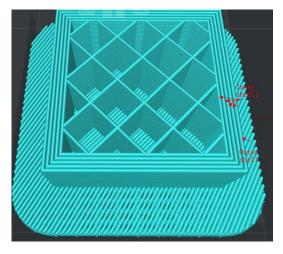


Figure 5.14: Only the shell located at outer edge is the outer shell. No matter how many shells you have set, except the outer one, the remaining shells are inner shells.

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.

Layer Start Point Type:

Fixed Layer Start Point X:

Nearest		•
Nearest		
Fixed	 	
Random		

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.

Layer Extruder Infill Suppor	t Platform Additions Cool	ing Temperature Advanced Ooze Ot	her GCode
General		Speed	
Layer Height:	0.1000 🚔 mm	Default Printing Speed:	50.0 🛉 mm/s
Shells:	2.0	Inner Shell Speed:	50.0 🔹 mm/s
Maximum Shells Overlap Percentage:	50 🔹 %	Outer Shell Speed:	30.0 🔹 mm/s
🗌 Print Shells in Optimal Order		X/Y Axis Movement Speed:	100.0 🔺 mm/s
First Layer Settings		Z Axis Movement Speed:	16.0 🔹 mm/s
First Layer Height:	0.3000 🗼 mm	Other	
First Layer Speed:	15.0 🛉 mm/s	🗌 Spiral Vase Mode	
First Layer Flowrate:	100.0 %	🗌 Print External Shells First	
Layer Start Point		🗌 Print Infill First	
Layer Start Point Type:	Nearest 👻	🗌 Print Parts in Same Order for Each	Layer
Fixed Layer Start Point X:	0.00 mm	Minimal Segment Length:	0.012 📩 mm
Fixed Layer Start Point Y:	0.00 mm	Merge Nearby Lines:	0.000 💌 mm
Seam Hiding		Dimensional Compensation	
Place Seam on:	None 🔻	XY Size Compensation for Contours:	0.00
		XY Size Compensation for Holes:	0.00 🖨 mm

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.

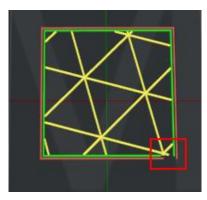


Figure 5.17: Set Coasting Distance to be 2mm.



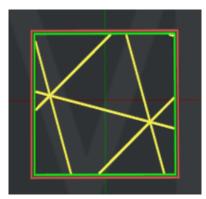


Figure 5.18: Set Coasting Distance to be 0.

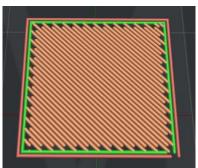


Figure 5.19: Enable the function of "Apply Coasting on Gird Pattern Infill and Lines Pattern Solid Fill".

Wipe:

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

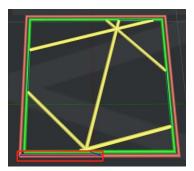


Figure 5.20: Set Outer Shell Wipe Distance to be 10mm.

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



er	Extruder	Infill	Support	Platform .	Additions	Cooling	Temperature	e Advanced	Ooze	Other	GCode		
eft I	Extruder	Right Extr	ruder										
Gene	eral						Coasting						
Ext	rusion Widt	h:			0.40	mm	Coasting Dist	ance:			0.00	▲ mm	
Retr	raction						Apply Coas	ting on Grid]	Infill an	d Lines S	Solid Fill		
\checkmark	Enable Retr	action					Wipe						
Ret	raction Spe	ed:			40.0	mm/s	Outer Shell W	/ipe Distance:			0.00	🛨 mm	
Ret	raction Mat	erial Amou	nt:		1.5	mm	Outer Shell W	lipe Speed:			50.0	🔹 mm/s	
Min	uimal Travel	of Retrac	tion:		1.0	mm							
Min	imal Amount	of Retrac	tion:		0.02	mm							
Ext	ra Restart	Amount:			0.00	mm							
Res	tart Speed:				40.0	mm/s							
zн	lop at Retra	ction:		0). 000 🗘	m							
	Z Hop Only	at Solid P	arts										

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.



Extruder Infill Support	t Platform Additions	Cooling	Temperature	Advanced	Ooze	Other	GCode	
ft Extruder Right Extruder								
Enable Right Extruder Configurati								
	ion							
eneral			oasting					
Extrusion Width:	0.40	mm	Coasting Distan	ce:		L	0.00	÷ mm
etraction			Apply Coasti	ng on Grid F	attern I	nfill and	Lines Patt	ern Solid Fi
✓ Enable Retraction		- 8	ipe					
			Juter Shell Wip	e Distance:		Г	0.00	÷ mm
Retraction Speed:	40.0	1000/ 5	_					
Retraction Material Amount:	1.5	mm	Duter Shell Wip	e Speed:		L	50.0	🔹 mm/s
Minimal Travel of Retraction:	1.0	mm						
linimal Amount of Retraction:	0.02	mm						
xtra Restart Amount:	0.00	mm						
lestart Speed:	40.0	mm/s						
Hop at Retraction:	0.000	mm						
Z Hop Only at Solid Parts								



yer Extruder	Infill	Support	Platform Additions	Cooling	Temperature	Advanced	Ooze	Other	GCode		
eft Extruder.	Right Extr	ruder									
General					Coasting						
Extrusion Wi	dth:		0.40	mm	Coasting Distan	ce:			0.00	nm	
Retraction					Apply Coasti	ng on Grid F	attern I	nfill and	l Lines Pa	ttern Solia	l Fil
🗹 Enable Re	traction				Wipe						
Retraction S	peed:		40.0	mm/s	Outer Shell Wip	e Distance:			0.00	÷ mm	
Retraction M	aterial Amou	nt:	1.5	mm	Outer Shell Wip	e Speed:			50.0	🔹 mm/s	
Minimal Trav	el of Retrac	tion:	1.0	mm							
Minimal Amou	nt of Retrac	tion:	0.02	mm							
Extra Restar	t Amount:		0.00	mm							
Restart Spee	d:		40.0	mm/s							
Z Hop at Ret	raction:		0.000	mm							
🗌 Z Hop Onl	y at Solid P	arts									

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).

Infill Extruder:	All Extruder 🔷 🔻
	All Extruder
	Left Extruder
Infill Density:	Right Extruder

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.

Infill Pattern Type:

Infill Extrusion Width Percentage:

Grid	•
Grid	
Rectilinear	
Honeycomb	
Triangles	
Cubic	

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



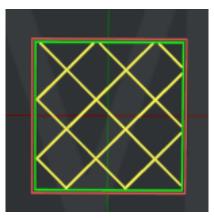


Figure 5.26: Set "Infill Pattern Type" to be "Grid".

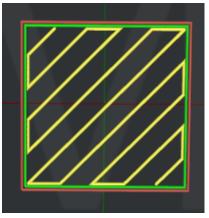


Figure 5.27: Set "Infill Pattern Type" to be "Rectilinear".

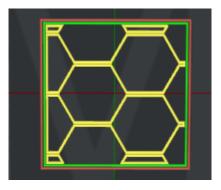


Figure 5.28: Set "Infill Pattern Type" to be "Honeycomb".

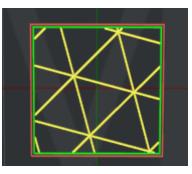


Figure 5.29: Set "Infill Pattern Type" to be "Triangles".



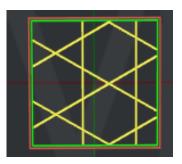


Figure 5.30: Set "Infill Pattern Type" to be "Cubic".

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.

Bottom Solid Fill Pattern Type:

Lines -Lines Rectilinear Concentric

Top Solid Fill Pattern Type:

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

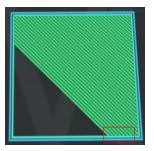


Figure 5.32: Set "Top Solid Fill Patter Type" to be "Lines".

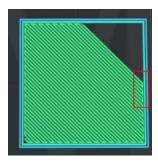


Figure 5.33: Set "Top Solid Fill Pattern Type" to be "Rectilinear".



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.

Advanc	ed Settings	;									?	\times
Layer	Extruder	Infill	Support	Platform Additions	Cooling	Temperature	Advanced	Ooze	Other GCo	de		
Infill						Top and Down S	olid Part —					
Infill	. Extruder:			All Extruder 🔹		Bottom Solid I	Fill Layers:		10	•		
Infill	. Density:			10 🔹 %		Top Solid Fill	l Layers:		10	*		
Infill	. Speed:			60.0 🛉 mm/s		Bottom Solid 1	Fill Speed:		60.0	t mm/s		
Infill	. Overlap:			15 🔹 %		Top Solid Fill	l Speed:		60.0	nm/s		
Infill	. Flowrate:			100.0 🔹 %		Bottom Solid 1	Fill Flowrate		100.0	÷ %		
Infill	Pattern T	уре:		Grid 🔫	·	DOLLOW DOLLG	IIII IIOWI die		100.0	v 70		
Infill	. Extrusion	Width Pero	entage:	100 🔺 %		Top Solid Fill	l Flowrate:		100.0	\$		
Infill	. Offset X:			0.00 🛉 mm		Bottom Solid 1	Fill Pattern	Туре:	Lines	•		
Infill	. Offset ¥:			0.00 🛉 mm		Top Solid Fill	l Pattern Typ	e:	Lines	•		
Combin	e Infill L	ayers:		1 *		🗹 Use Gap Fil	lling for Fai	ld Solid F	ill Parts			
Infill	Angle					Top and Down S	olid Fill An	gle				
Angle:		0	Deg	45 135		Angle:	0	Deg	45 135			
A	dd Infill A	ingle				Add Infil	l Angle					
Rem	ove Infill	Angle				Remove Inf	ill Angle					
Resto	re 🔻									OK	Cance	1

Figure 5.35: The Infill tab.



5.1.3.4 Support

Generate Support refers to selecting which structure for overhang part.

Generate Support:	None 🔻
Support	None Touch Platform Only All

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.

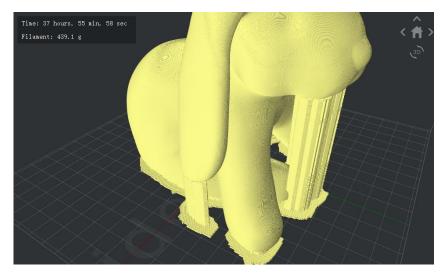


Figure 5.37: Printed with supports of "Touch Platform Only".

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



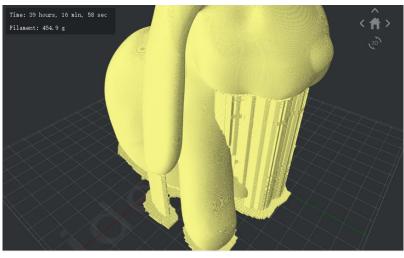


Figure 5.38: Printed with supports of "All".

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.

Support Type:	Normal 🔻
	Normal
Support Infill Type:	Pillar

Figure 5.39: Two types of support structure.

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

Support	Infill	Туре:
Support	Speed:	



Figure 5.40:	Two types	of Support Inf	ill pattern.
--------------	-----------	----------------	--------------



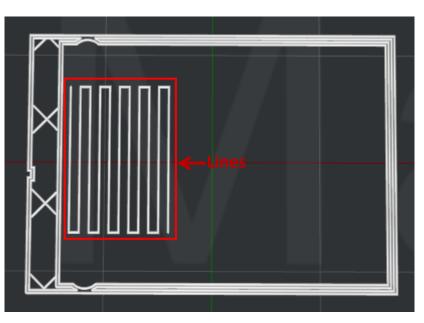


Figure 5.41: Set "Support Infill Type" to be "Lines".

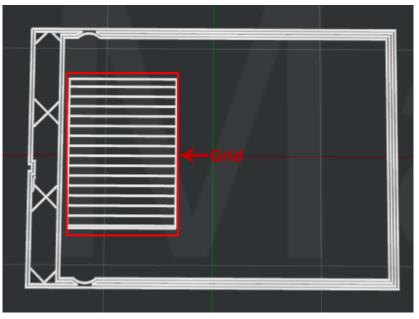


Figure 5.42: Set "Support Infill Type" to be "Grid".

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).



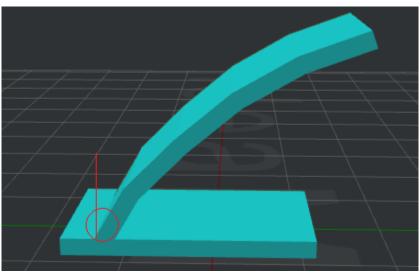


Figure 5.43: Overhang angle.

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.

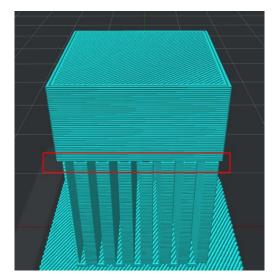


Figure 5.44: Set "Horizontal Expansion" to be 0.



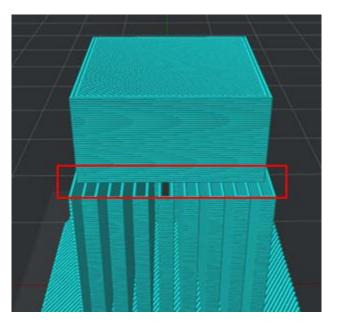


Figure 5.45: Set "Horizontal Expansion" to be 2.

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

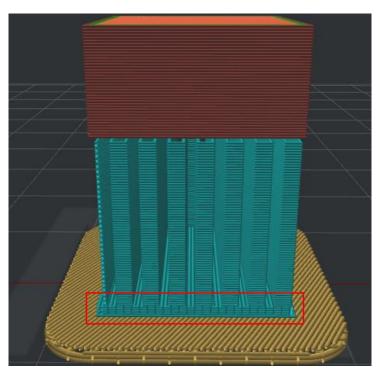


Figure 5.46: Set "Support Base Layer" to be 5.



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**).

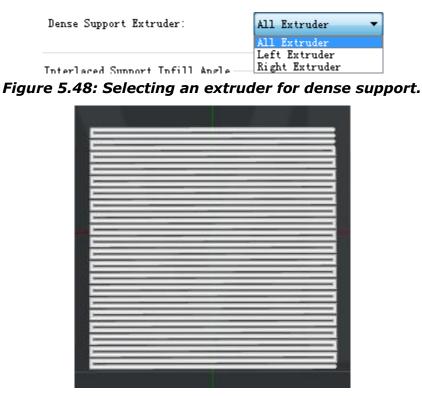


Figure 5.49: Set "Dense Support Infill Type" to be "Lines".





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.

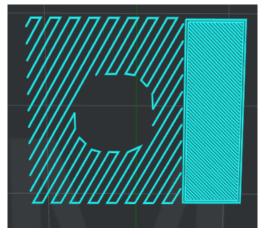


Figure 5.51: Set "Interlaced Support Infill Angle" to be 30 degrees.



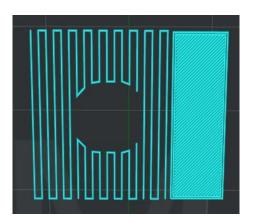


Figure 5.52: Set "Interlaced Support Infill Angle" to be 0 degree.

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.

Advanced Settings			? ×
Layer Extruder Infill Support	Platform Additions Cooling	g Temperature Advanced Ooze	Other GCode
Generate Support: 💍 All	•		
Support		Dense Support	
Support Extruder:	Left Extruder 🛛 🔻	Dense Support Layers:	0
Support Type:	Normal 🗸	Dense Support Infill Ratio:	80 🔶 %
Support Infill Type:	Lines 🔻	Dense Support Infill Type:	Line 👻
Support Speed:	50.0 🛉 mm/s	Dense Support Extruder:	All Extruder 🔹
Infill Ratio:	30 🐳 %	Interlaced Support Infill Angle	
Max Overhang Angle:	45 🔺 Deg	Angle: 0 🛉 Deg	-45
Horizontal Offset:	0.70 🛉 mm	Add Infill Angle	
Vertical Offset Top Layers:	1		
Vertical Offset Down Layers:	1	Remove Infill Angle	
Support Flowrate:	100.0 🔹 %		
Horizontal Expansion:	0.00 m	Other	
Solid Base Layers:	0	Pillar Size:	4.00 🛖 mm
		Add Sparse Connection	
Restore 🔻			OK Cancel

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.

Raft Lines Type:



Figure 5.55: Two types of Raft Lines.



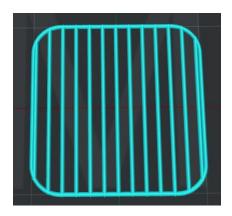


Figure 5.56: Set "Raft Lines Type" to be "Lines".

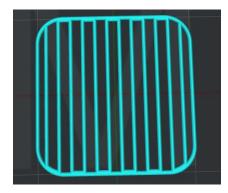


Figure 5.57: Set "Raft Lines Type" to be "Rectilinear".

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**.

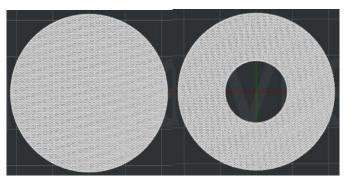


Figure 5.58: Normal Raft. 5.59: Keep Holes in raft structure.

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.

First Layer Middle Layer	Surface Layer
Middle Layers:	1
Middle Layer Speed:	30.0 🚔 mm/s
Extrusion Width Percentage:	150 🔹 %
Layer Height:	0.3800 🗭 mm
Middle Layer Infill Ratio:	50 🗘 %
Middle Layer Infill Angle:	90 📮 Deg

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.

First Layer Middle Layer	Surface Layer
Surface Layers:	2
Surface Layer Speed:	60.0 🗭 mm/s
Extrusion Width Percentage:	100 🔹 %
Layer Height:	0.2200 💼 mm
Surface Layer Infill Ratio:	85 🔹 %
Surface Layer Infill Angle:	30 🗘 Deg
Surface Layer Flowrate:	173.0 🔶 %

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.



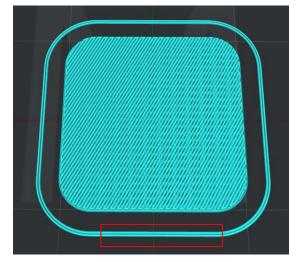


Figure 5.62: Set "Skirt Loop Lines" to be 2.

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

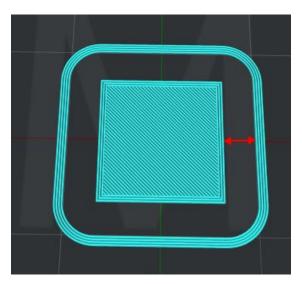


Figure 5.63: Set "Skirt Offset Distance" to be 5mm.

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.



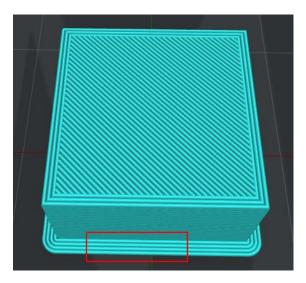


Figure 5.64: Set "Brim Loop Lines" to be 4.

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*.

	Platform Additions Cooling	Temperature Advanced Ooze O	ther GCode
Platform Addition: Raft Onl	ty 🔻		
Raft		Skirt and Brim	
Raft Extruder:	Left Extruder 🔹 🔻	Skirt/Brim Extruder:	Left Extruder 💎 👻
Raft Offset:	5.00 m	Skirt Loop Lines:	1
Raft Gap from Model:	0.20 m	Skirt Offset Distance:	3.0 mm
Raft Lines Type:	Lines 🔻	Brim Loop Lines:	5
🗌 Keep Holes in Raft Structure		Print Outer Shell Before Inner S	hell in the First Layer
First Layer Middle Layer Surf	face Layer		
First Layers:	2		
First Layer Speed:	8.0 🗭 mm/s		
Extrusion Width Percentage:	200 🔹 %		
Layer Height:	0.5000 🗭 mm		
Layer Height: First Layer Infill Ratio:	0.5000 nm		
First Layer Infill Ratio:	33 🔹 %		

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.

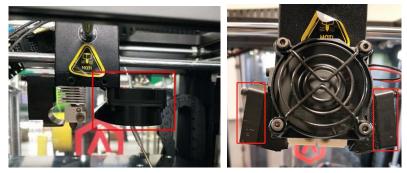


Figure 5.66: Controllable cooling fan.



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.

Printer Settings						?	×
Printer Type:	RAISE3D N2 - V2 Hot	End	•	Remove	Export	Reset	
General Advance	ced						
-Printer Setting	gs		Extruders				
Nozzle Diamete	r:	0.40 m	Extruder Count:		1 🔻		
Build Width:		305.00 🔹 mm	Primary Extruder:		[Raise3D] PLA 1.75m	m - V2 Hot End	•
Build Depth:		305.00 🛉 mm					
Build Height:		305.00 🔹 mm					
🗹 Vse Heated	Bed						
🗹 Enable Fan	Speed Control (Use PWM	-Controllable Fans)					
Distance From	Border of Build Plate:	2.50 🔹 mm					
Build Plate Sh	ape:	Rectangle 🔹 🔻					
Firmware:		Marlin 💌					
Add / Duplicate	Import				Save	Clos	se

Figure 5.67: Use PWM-Controllable fans



Layer	Fan Speed
1	0
2	100
	Layer 1

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.

1 Advanced Settings	?	×
Layer Extruder Infill Support Platform Additions Cooling Temperature Advanced Ooze Other GCode		
Cooling Blip fan speed to 100% when starting from low speed		
Minimal Layer Print Time: 15.0 🌲 see Low Fan Speed Threshold: 30	* 96	
Slow Down Frinting Speed 500 Elip Fan Speed Pause Duration: 500	msec	
Minimal Printing Speed: 10.0 🛊 mm/s		
🔽 Increase Fan Speed		
Maximum Fan Speed: 0 🔹 %		
Fan Control		
Layer: 1 🗘 Layer Fan Speed		
Fan Speed: 0 🔶 %		
Add Fan Point		
Remove Fan Foint		
Restore • 0K	Cano	el

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.

yer Extruder Infill Support	Platform Additions Cooling	Temperature	Advanced Ooze Other	GCode	
Temperature		Cool Do	wn Inactive Extruder		
Heated Bed Temperature:	60 🗘 ° C	Park Posi	tion X:	30.00	mm
Primary Extruder:	205 🔹 °C	Park Posi	tion Y:	295.00	mm
		Inactive	Cooling Temperature (Left):	170	°C
🗌 Use Temperature Control List		Inactive	Cooling Temperature (Right):	170	°C
Heated Bed Primary Extruder		Heat v	p Inactive Extruder in Advance	P	
Acato bea Trimary Extruder	Layer Temperature	Heat up A	head of Time:	40.0	sec
Layer: 1	Layer remperature	Inactive	Heating Temperature (Left):	200	°C
Temperature: 40 * °C		Inactive	Heating Temperature (Right):	200	°C
Add Temperature Remove 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%.

ayer Extruder Infill Support	Platform Additions Cool	ing Temperature Advanced Ooze	e Other GCode
Enable Acceleration Control		🗌 Enable Jerk Control	
Acceleration		Jerk	
Printing Acceleration:	800.00 🛉 mm/s2	Printing Jerk:	12.00 mm/s
Inner Shell Acceleration:	600.00 mm/s2	Inner Shell Jerk:	12.00 mm/s
Outer Shell Acceleration:	500.00 🛉 mm/s2	Outer Shell Jerk:	12.00 mm/s
Infill Acceleration:	1000.00 🛉 mm/s2	Infill Jerk:	12.00 mm/s
Bottom Solid Fill Acceleration:	800.00 🛉 mm/s2	Bottom Solid Fill Jerk:	12.00 mm/s
Top Solid Fill Acceleration:	800.00 m m/s2	Top Solid Fill Jerk:	12.00 mm/s
Travel Acceleration:	1000.00 🛉 mm/s2	Travel Jerk:	12.00 mm/s
🗸 Override Filament Settings (Left)	Extruder)		
Left Filament Flowrate:	90.0 🗘 %		
🖉 Override Filament Settings (Right	Extruder)		
Right Filament Flowrate:	90.0 🔹 %		

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).

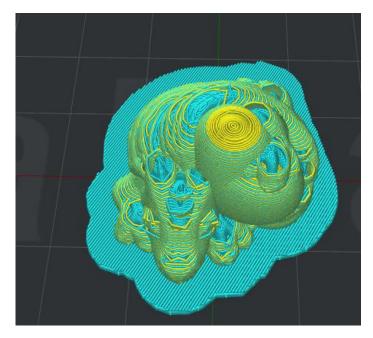


Figure 5.72: Set "Wipe Wall Angle" to be 90.

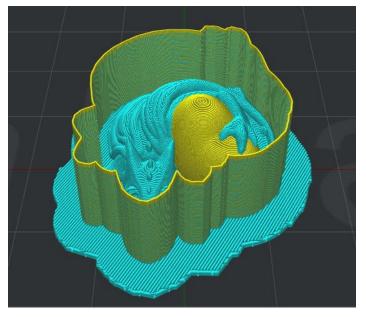


Figure 5.73: Set "Wipe Wall Angle" to be 0.



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

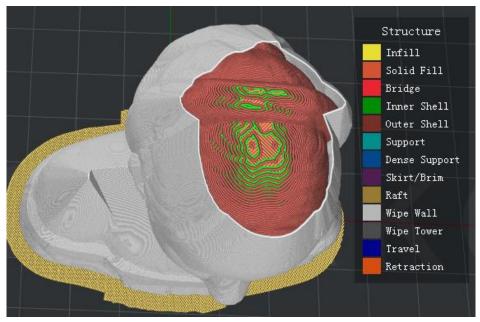


Figure 5.74: Set "Wipe Wall Loop Lines" to be one.

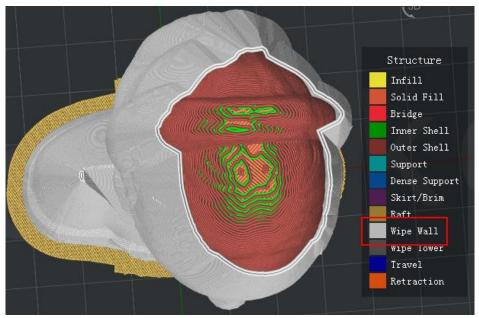


Figure 5.75: Set "Wipe Wall Loop Lines" to be two.

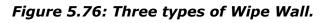
Wipe Wall Type refers to the shape of the wipe wall shells. The primary difference is the distance of model and wipe wall.

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.



WipeWall Type:

Contoured	
Contoured	
WaterFall	
Vertical	



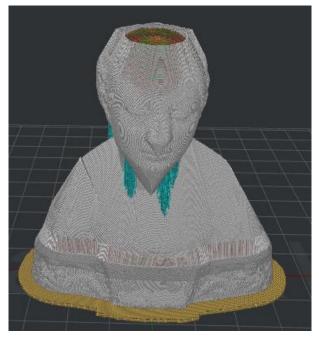


Figure 5.77: Set "Wipe Wall Type" to be "Contoured".

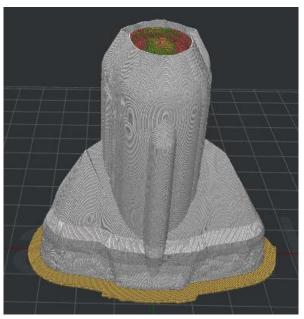


Figure 5.78: Set "Wipe Wall Type" to be "WaterFall".



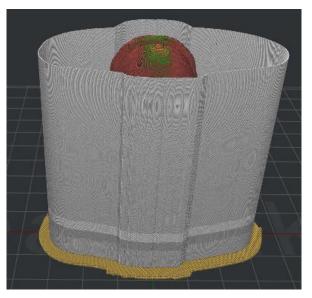


Figure 5.79: Set "Wipe Wall Type" to be "Vertical".

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.

🕤 🗹 Placed at the Fixed Position of the	Platform		
Wipe Tower Offset X:	-2.00	* *	۱M
Wipe Tower Offset Y:	2.00	<u>к</u>	۱M
Wipe Tower Fixed Position X:	27.50	• •	וח
Wipe Tower Fixed Position Y:	280.00	• •	וח

Figure 5.80: Wipe Tower Fixed Position.



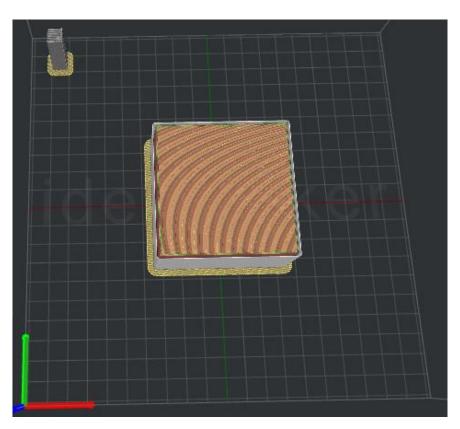


Figure 5.81: Enable the function of Placed at the Fixed of the Platform.

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.

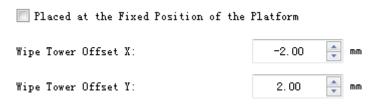


Figure 5.82: Wipe Tower X/Y offset.



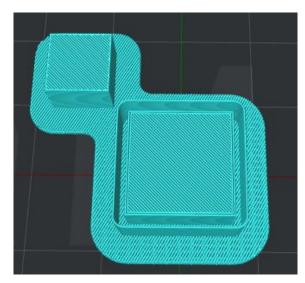


Figure 5.83: Wipe Tower.

Advanced Settings		?)
Layer Extruder Infill Support Platform Additions Cooli	ing Temperature Advanced Ooze Othe	er GCode
Other	☑ Wipe Wall	
🗹 Force Retraction On Layer Change	Wipe Wall Offset:	2.00 🗭 mm
☐ Force Retraction Before Traveling to Outer Shell	Wipe Wall Angle:	30 📮 Deg
Avoid Retraction Inside Models	Wipe Wall Loop Lines:	1
🗹 Enable Retraction at Bottom and Top of Models	Wipe Wall Type:	Contoured 👻
Bottom Layers: 0	Wipe Wall Speed:	50.0 🗭 mm/s
Top Layers: 2	🕤 🔽 Wipe Tower	
Extruder Switch Ooze Control	Wipe Tower Width:	12.00 m
Retraction Speed of Extruder-switch: 20.0 💌 mm/s	Wipe Tower Infill Ratio:	100 🐳 %
Retraction Amount of Extruder-switch: 3.50 🔭 mm	Wipe Tower Speed:	50.0 🛉 mm/s
Restart Speed of Extruder-switch: 20.0 💌 mm/s	Placed at the Fixed Position of	the Platform
Extra Restart Amount of Extruder-switch: 0.00 💌 mm	Wipe Tower Offset X:	-2.00 🛉 mm
Travel	Wipe Tower Offset Y:	2.00 🗭 mm
Avoid Traveling Through Holes	Wipe Tower Fixed Position X:	27.50 mm
Maximum Travel Path Length: 200 🔹 %	Wipe Tower Fixed Position Y:	280.00 👘 mm
Restore 🔻		OK Cancel

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.

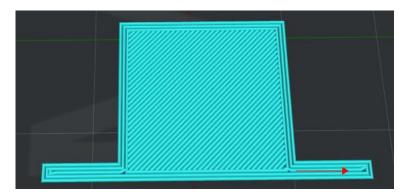


Figure 5.85: Enable the "Fill Gaps in shells".

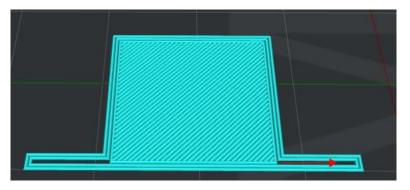


Figure 5.86: Unable the "Fill Gaps in shells".

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**.

Advanced Settings		? ×
Layer Extruder Infill Support Platform Additions Cooling	Temperature Advanced Ooze Other	GCode
Repair	Pause at Height	
Merge Open Segments of Model Parts	Height: 5.000 🖨 mm	
Merge Internal Overlapping Parts	Add Pause Point	
Thin Wall	Add Fause Foint	
🗹 Check Thin Wall (Single Extrusion Width)	Remove Pause Point	
Minimal Extrusion Width Percentage: 25 🔺 %		
Maximum Extrusion Width Percentage: 200 🔹 %	The height property does not include raft.	
Gap Filling	Bridging	
✓ Fill Gaps in Shells	Extrusion Width Percentage:	100 🗘 %
Gap Filling Speed: 60.0 🖨 mm/s	Bridging Speed:	60.0 🔹 mm/s
Allow Filling Gaps using Single Extrusion Filling	🗌 Enable Bridging Fan Speed	100 🔹 %
Minimal Single Extrusion Width Percentage: 33 🚖 %	Enable Fixed Bridging Angle	0 📥 Deg
Maximum Single Extrusion Width Percentage: 200 🔹 %	Minimal Allowed Briding Area:	20.00 🔹 mm2
Single Extrusion Filling Speed: 40.0 🔹 mm/s	Global Offset	
Travel	X-Offset:	0.00 🛊 mm
Avoid Traveling Through Holes	Y-Offset:	0.00 🗭 mm
	Z-Offset:	0.00 🗭 mm
Restore 🔻		OK Cancel

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.

Start GCode when the number of extruders is

Insert Heatup GCode Automatically in Start²

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).

🗹 Insert Heatup GCo	de Automatically in Start GCode
Heatup Sequence:	Heatup heated bed and nozzle(s) simultaneously 🔹 🔻
	Heatup heated bed and nozzle(s) simultaneously
-Post-process Commands	Heatup nozzle(s) first Heatup heated bed first

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.

tart GCode End GCod	e Extruder Switch GCode Layer Change GCod	le Retraction	GCode
Start GCode when the G21 G30 M107 G28 X0 V0 G28 X0 V0 G28 Z0 G1 Z15.0 F{travel_x; T1 G32 Z0 G1 F200 E10 G32 Z0 G1 F200 E10 G32 Z0 G1 F200 E10 G32 Z0 G1 F200 E10 G32 D0 G1 F200 E10 G1 F20	 Add Settings Add settings you want to modify. Layer (2 selected) Left Extruder (1 selected) Right Extruder Infill Support Platform Additions Cooling Temperature (3 selected) Advanced Doze (1 selected) 	? ×	Built-In Flaceholders: (Double click item to add to goode) Slice Settings {travel_xy_speed} {layer_default_speed} {retraction_speed1} {temperature_extruder1} {temperature_extruder2} {temperature_heatbed} {extruder_switch_retaction_amount}
GI F{travel_xy_spee M117 Printing Insert Meatup GCo Meatup Sequence:	>		Add More Remove Selected
ost-process Commands	OK	Cancel	Add Remove (Double click item to edit settings)

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*.



Command Type:	Replace SOURCE with	TARGET		•
Source			Target	
G92 E0			G1 X20 YO F140 E30	

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

Post-process Commands		
{"source":["G92 E0"],"target":["G1 X20 Y0 F140 E30"],"type":"replace"}	Add Remove	
	(Double click item to edit settings)	

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*.

Command Type: Remov	e all occurrences of SOURCE	•
Source	Target	
692 EO	GI X20 YO F140 E30	
	OK	Cancel

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



Post-process Commands	
{"source":["G92 E0"],"type":"remove"}	Add Remove
	(Double click item to edit settings)

Figure 5.94: The Post-process Command tab.

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

tart GCode End GCode Extruder Switch GCode Layer Change GCode Retractio	on GCode
Start GCode when the number of extruders is 2 G21 G30 M107 G28 X0 Y0 G28 X0 Y0 G28 Z0 G1 Z15.0 F{travel_xy_speed} 1 T1 G32 E0 G1 F200 E10 G32 E0 G1 F200 E-[extruder_switch_retaction_amount] 0 G2 E0 G1 F200 E10 G32 E0 G1 F200 F10 G1 F200 F10 G32 E0 G1	Built-In Placeholders: (Double click item to add to goode) V Slice Settings {travel_xy_speed} {layer_default_speed} {retraction_speed1} {temperature_extruder1} {temperature_extruder2} {temperature_heatbed} {extruder_switch_retaction_amount}
✓ Insert Heatup GCode Automatically in Start GCode Heatup Sequence: Heatup heated bed and nozzle(s) simultaneously ▼	Add More Remove Selected
ost-process Commands	
("source":["G92 E0"],"target":[],"type":"replace"}	Add Remove (Double click item to edit settings)

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.

art GCode End GCode Extruder Switch GCode Layer Change GCode Retra	otion GCode
Start GCode when the number of extruders is $\hfill 2$ $\hfill \neg$	Built-In Placeholders: (Double click item to add to gcode)
<pre>G21 G30 MU07 G28 X0 Y0 G28 X0 Y0 G32 Z0 G1 Z15.0 F{travel_xy_speed} T1 G32 E0 G1 F200 E10 G32 E0 G1 F200 E-{extruder_switch_retaction_amount} T0 G32 E0 G1 F200 E10 G32 E0 G1 F[travel_xy_speed] M117 Printing</pre>	 Slice Settings {travel_xy_speed} {layer_default_speed} {retraction_speed1} {temperature_extruder1} {temperature_extruder2} {temperature_heatbed} {extruder_switch_retaction_amount}
☑ Insert Meatup GCode Automatically in Start GCode featup Sequence: Heatup heated bed and nozzle(s) simultaneously ▼	Add More Remove Selected
st-process Commands	Add Remove (Double click item to edit settings)

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.

emplate 1			Template 2		
From ideaMaker	From Local Disk		From ideaMak	er From Local Disk	
Printer: RAISE3D N2 - V2 Hot End		End 👻	Printer:	RAISE3D N2 - V2 Hot End	. –
Filament:	PLA 1.75mm - V2 Hot	End 💌	Filament:	PLA 1.75mm - V2 Hot End	
Template:	High Quality - N2 -	PLA 👻	Template:	Standard - M2 - PLA	•
ain Template	Group and Layer Set	tings			
Setting		High Quality - N2 -	PLA	Standard - N2 - PLA	
✓ Layer					
Layer He	-	0.1000 mm		0.1300 mm	
	Movement Speed	100.0 mm/s		30.0 mm/s	
✓ Left Extruder					
	t Retraction	0.000 mm		0.130 mm	
✓ Infill					
Infill Spe		60.0 mm/s		30.0 mm/s	
	Solid Fill Layers	7		5	
	d Fill Layers	10		7	
	Solid Fill Speed	60.0 mm/s		20.0 mm/s	
	d Fill Speed	60.0 mm/s		20.0 mm/s	
 Support 					
Generate	e Support	None		All	
Generate Support		None 50.0 mm/s		All 30.0 mm/s	?
Generate Support		Solo mm/s Select Template	82 - V2 Hot End		?
Generate Support	Speed ₩ ↔	50.0 mm/s Select Template Frinter Type: RATSE3D 1	82 - V2 Hot End		
Generate Support	Speed ₩ ↔	50.0 mm/s Select Template Printer Type: RAISE20 1 Filamant	82 - W2 Hot End nn - W2 Hot End (3 temp)	30.0 mm/s	
Generate Support	Speed ₩ ↔	50.0 mm/s Select Template Printer Type: RAISE20 J Filamat Drinary Extruder: PLA 1.75	nm - V2 Hot End (3 temp]	30.0 mm/s	•
Generate Support	Speed	50.0 mm/s Select Template Printer Type: RAISE20 1 Filamant	nm - V2 Hot End (3 temp]	30.0 mm/s	•
Generate Support	Speed	50.0 mm/s Select Template Printer Type: RAISE20 1 Filment Prinary Extruder: PLA 1.75 Main Template Group and Lay	nm — V2 Hot End (3 temp] er Settings Per-Mode]	30.0 mm/s .etcs) Extruder	▼ Ø
Generate Support	Speed	50.0 mm/s Select Template Printer Type: RAISE20 Filsent Prinary Extruder: PLA 1.755 Main Template Group and Lay	nm - V2 Hot End (3 temp]	30.0 mm/s .etcs) Extruder ?	• 0
Generate Support	Speed Pan Hove tes	50.0 mm/s Select Template Printer Type: BAIS220 1 Filmant Prinary Extruder: PLA 1.75 Hain Template Group and Lay Template	nn - V2 Hot End (3 temp) er Settings Per-Mode] ate 2 n ideaMaker From Loc	30.0 mm/s .etcs) Extruder ?	
Generate Support	Speed Fan Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore K	50.0 mm/s	nn - V2 Hot End (3 teng) er Settings – Per-Model ste 2 n ideaWaker – From Loc nter: – RAISE3D N2 ancht: – FLA 1.75am	30.0 mm/s .tes) .Extruder ? al Disk - V2 Kot End - V2 Kot End	 Create Nuplicate Edit Compare
Generate Support	Speed Fan Hove tes From Local Disk MISE3D #2 - V2 Hot End	50.0 mm/s	nn - V2 Hot End (3 tenp) er Settings Per-Model ate 2 n ideaMaker Pron Loc nter: RAISE3D K2	30.0 mm/s .tes) .Extruder ? al Disk - V2 Kot End - V2 Kot End	Create Záit Congare
Generate Support	Speed Fan Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore K	50.0 mm/s	nn - V2 Hot End (3 teng) er Settings Per-Model ate 2 n ideaMaker From Loc nter: RAISE3D H2 anent: FLA 1.75am plate: Standard -	30.0 mm/s .tes) .Extruder ? al Disk - V2 Kot End - V2 Kot End	Create Multicate Kdit Compare
Generate Support	Speed Fan Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore K	50.0 mm/s Select Template Frinter Type: EMISED Filamont Frinary Extruder: FEA 1.756 Main Template Group and Lay Fro Fri Template Template Template Template Template Template Template Template Template Template	nn - V2 Hot End (3 teng) er Settings Per-Model ate 2 n ideaMaker From Loc nter: RAISE3D H2 anent: FLA 1.75am plate: Standard -	30.0 mm/s .tes) .Extruder ? al Disk - V2 Kot End - V2 Kot End	 Create Muplicate Edit Conpare Delete
Generate Support	Speed Fan Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore K	50.0 mm/s Select Template Frinter Type: EMISED Filamont Frinary Extruder: FEA 1.756 Main Template Group and Lay Fro Fri Template Template Template Template Template Template Template Template Template Template	nn - V2 Hot End (3 temp) er Settings Per-Model ate 2 n ideaMaker Pron Loc nter: RAISE3D H2 anent: PLA 1.75mm plate: Standard - Found!	30.0 mm/s .tes) .Extruder ? al Disk - V2 Kot End - V2 Kot End	 Create Replicate Zdit Conpare Delete Import
Generate Support	Speed Fan Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore K	50.0 mm/s Select Template Printer Type: EXISED 1 Filmant Prinary Extruder: FLA 1.75 Main Template Group and Lay Filmant From Fri Fri Fri Fri Fri Fri Fri Fri	nn - V2 Hot End (3 teny) er Settings Per-Model ate 2 n ideaMaker From Loc nter: RAISE3D H2 anent: PLA 1.75mm plate: Standard - found! H2 - PLA	30.0 mm/s .tes) .Extruder ? al Disk - V2 Kot End - V2 Kot End	 Create Rajicate Zajit Caspare Delete Taport
Generate Support	Speed Fan Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore Kore K	50.0 mm/s Select Template Frinter Type: EMISED 1 Filmant Frinary Extruder: FLA 1.75 Main Template Group and Lay Fri Fri Fri Fri Fri Fri Fri Fri	nn - V2 Hot End (3 teny) er Settings Per-Model ate 2 n ideaMaker Pron Loc nter: RAISE3D H2 anent: PEA 1.75mn plate: Standard - found! E2 - PEA	30.0 mm/s .etez) Extrudor ? al Disk - V2 Kot End - V2 Kot End KZ - FLA	 Create Rajicate Zajit Caspare Delete Taport

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.

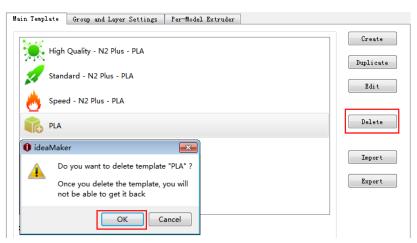


Figure 5.98: Delete slice template.



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.

Import Slicing Templa	te		? ×
Basic			
Slicing Template:	0.8 nozzle-N2(1).bin		
Template Name:	0.8 nozzle-export		
		OK	Cancel

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.

Import Slicing Templat	te	? 💌
Basic		
Slicing Template:	R12-export.bin	
Template Name:	R12-export	
Import to the followin	g printer	
Printer Type: RAIS	E3D N2 - V2 Hot End	
🔲 No, I want to impor	t the new printer included in the template	Details
Import to the followin	g filament	
Filament Type: Poly	Support 1.75mm	
🔲 No, I want to impor	t the new filament included in the template	Details
	OK	Cancel

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**.

1 Import Slicing Template	•	? 💌
Basic		
Slicing Template:	R12-export. bin	
Template Name:	R12-export	
- Import to the following	printer	
Printer Type: RAISE	3D N2 - V2 Hot End	
📝 No, I want to import	the new printer included in the template	Details
Import to the following	filament	
Filament Type: PolyS	upport 1.75mm	
V No, I want to import	the new filament included in the template	Details
	OK	Cancel

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*.

🕕 Edit - "RAISE3D Pro2-copy"				? 🗙
Printer		Extruder		
Type: RAISE3D Pro2-copy		Extruder Count:	2 🔹	
Nozzle Diameter:	0.40 mm	Left Extruder:	PLA 1.75mm	-
Build Width:	305.00 💼 mm	Right Extruder:	PLA 1.75mm	-
Build Depth:	305.00 💼 mm	Extruder Offset X:	0.00 💼 mm	
Build Height:	300.00 💌 mm	Extruder Offset Y:	0.00 mm	
Step-E per MM:	0.00	Temperature Limit		
X-axis Compensation:	100.00 🔹 %	Heated Bed Maximum Temperature	Limit: 110	÷ ° C
Y-axis Compensation:	100.00 💌 %	Primary Extruder Maximum Tempe	arature Limit: 310	🗘 ° C
Baud Rate:	230400	Right Extruder Maximum Tempera	ature Limit: 310	÷ C
🔽 Use Heated Bed				
📝 Enable Fan Speed Control (Use	PWM-Controllable Fans)			
			Save	Close

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*.

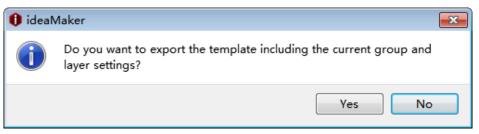


Figure 5.103: Export the slicing template.



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.

Select Template		8 23
Printer Type:	RAISE3D Pro2 Plus	•
Filament		
Left Extruder:	PLA 1.75mm (4 templates)	• \$
Right Extruder:	PVA 1.75mm	- ÷
Main Template Gr	oup and Layer Settings Per-Model Extruder	
Setting Groups	Setting Group 1	
Setting Group	Per-Group Settings Per-Layer Settings	
+ -		
Import an	d Export	
		Close Slice

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.

Add New Setting Group	
Setting Group Name: Setting Group 2	
Setting Groups	New Setting Group
▲ Setting Group 1	
20mm_Calibration_Box.stl	
35mm fan bottom cover.stl	
Leveling Block.stl	
	OK Cancel

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

· refe

1

: 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.

Select Template				? 🛛
Printer Type: RAISE3D N	🕽 Edit Setting Group -	"Setting Group 3"		? 🗙
Filament	Setting Group Name:	Setting Group 3		
Primary Extruder: PLA 1.75m Main Template Group and Laye Setting Groups D Setting Group 1 D Setting Group 2 A Setting Group 3 Leveling Block.STL	Setting Groups Setting Group 1 20mm_Calibra Setting Group 2	ation_Box.stl	Setting Group 3 Leveling Block.STL	
+ - /		- /	ОК	Cancel
			Close	Slice

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.

1 Add Settings	? 💌
Add the settings you want to modify.	
▶ 🔲 Layer	A
Left Extruder	
Right Extruder	
▷ 🔲 Infill	
Support	
Cooling	
Ooze	E
4 🔲 Other	
Merge Open Segments of Model Parts	
Merge Internal Overlapping Parts	
Check Thin Wall	
Thin Wall Width	
Bridging Extrusion Width Percentage	
Bridging Speed	
Global Offset X	-
OK	Cancel

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.



ting Groups Setting Group 1	Setting Group 1 Per-Layer Settings		
35mm fan bottom cover.stl	▲ Layer		
Setting Group 2	Layer Height	0.15 mm	
20mm_Calibration_Box.stl	Shells	2	
Setting Group 3	First Layer Height	0.30 mm	
Leveling Block.stl	First Layer Speed	8.0 mm/s	
	First Layer Flowrate	100.0 %	
	Layer Start Point Type	Nearest	
	Fixed Layer Start Point X	0.00 mm	
	Fixed Layer Start Point Y	0.00 mm	
	Default Printing Speed	40.0 mm/s	
	Outer Shell Speed	15.0 mm/s	
	Inner Shell Speed	40.0 mm/s	
	X/Y Axis Movement Speed	100.0 mm/s	
	Z Axis Movement Speed	25.0 mm/s	
	Print External Shells First	Disabled	

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.

Setting Group 2 Per-La	yer Settings
▲ Setting1	0.00 - 0.00, 0.15 mm, 100%
Start Z	0.00 mm
End Z	0.00 mm
Layer Height	0.15 mm
Speed Multiplier	100 %

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.



Edit - "Setting1"	? 🔀
▲ Layer	
Layer Height	0.15 mm
Add Settings	Remove Selected
Add Settings	Vemove Serected
	OK Cancel
	OK Cancel

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.

1 Edit - "Setting1"	1 Add Settings
Layer Height 0.15 mm	Add the settings you want to modify. Add the settings you want to mod
Add Settings	OK Cancel
	OK Cancel

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

_	Import and Export
	Export to Local Disk
	Clear Group and Layer Settings
	Import from Local Disk
	Import from 20mm_Calibration_Box.idea
	Import from Last Used Group and Layer Settings

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.

Template Group and Layer Setting	s (3 Setting Groups) Per-Model Extruder
20mm_Calibration_Box.stl	Left Extruder
35mm fan bottom cover.stl	Left Extruder
Leveling Block.stl	Right Extruder

Figure 5.113: Choose an extruder for your model.



6 Multiple Extruders

6.1 Calibrate XY offset

Printer->Printer Settings-> Extruders Count

Print	ter(<u>C)</u> Help(<u>H</u>)
	Current Type of Printer
	Connect to Printer (Raise3D N-series)
	ideaPrinter F-series
	Printer Settings
	Filament Settings
	Configuration Wizard

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

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

neral Adva	nced			
Printer Settin	ngs		Extruders	
Left Nozzle I	liameter:	0.40 🛉 mm	Extruder Count:	2 💌
Right Nozzle	Diameter:	0.40 * mm	Left Extruder:	[Raise3D] PLA 1.75mm = V2 Hot End
Build Width:		305.00 m	Right Extruder:	[Raise3D] PLA 1.75mm = V2 Hot End
Build Depth:		305.00 🛉 mm	Extruder Offset X:	25.00 🔹 nm
Build Height:		305.00 🗘 mm	Extruder Offset Y:	0.00 🔹 mm
	n Speed Control (Use PWM			Y above are set to O, the following hether models are placed outside the
Distance From	Border of Build Plate:	2.50 🛉 mm	Virtual Extruder Offset X:	25.00 mm
Build Plate S	Shape:	Rectangle 🔹	Virtual Extruder Offset Y:	0.00 🐥 mm
Firmware:		Marlin 👻		

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*:

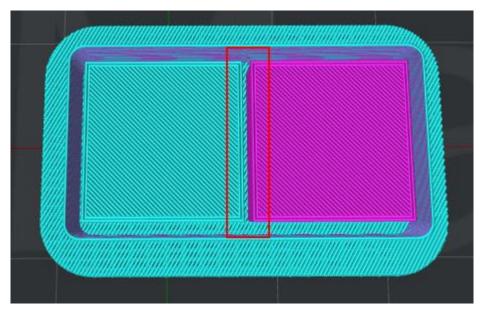


Figure 6.3: There is a gap between two calibration boxes.

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.



nter Type:	RAISE3D N2 - V2 Hot	End	 Remov 	e Export Reset
neral Advanc	ed			
Printer Setting	5		Extruders	
Left Nozzle Di	ameter:	0.40 mm	Extruder Count:	2 👻
Right Nozzle D	iameter:	0.40 mm	Left Extruder:	[Raise3D] FLA 1.75mm = V2 Hot End
Build Width:		305.00 🗭 mm	Right Extruder:	[Raise3D] PLA 1.75mm = V2 Hot End
Build Depth:		305.00 🔹 mm	Extruder Offset X:	25.00 🛉 mm
Build Height:		305.00 📫 mm	Extruder Offset V:	0.00 m m
🗹 Vse Heated I	Bed		Extruder offset 1.	
🗹 Enable Fan S	Speed Control (Use PW	M-Controllable Fans)		Y above are set to O, the following whether models are placed outside the
Distance From 1	Border of Build Plate	: 2.50 🛉 mm	Virtual Extruder Offset X:	25.00 m
Build Plate Sh	ape:	Rectangle 🔻	Virtual Extruder Offset V:	0.00 mm
Firmware:		Marlin 🔻		

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

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

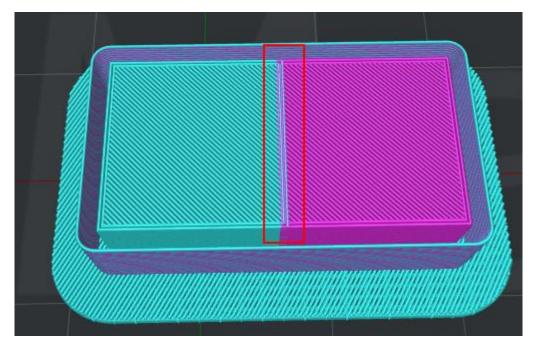


Figure 6.5: There is a superposition between two calibration boxes.

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.

Printer Settings	? ×
Printer Type: RAISE3D N2 - V2 Hot End	▼ Remove Export Reset
General Advanced	
Printer Settings	Extruders
Left Nozzle Diameter: 0.40 💌 mm	Extruder Count: 2
Right Nozzle Diameter: 0.40 💌 mm	Left Extruder: [Raise3D] PLA 1.75mm - V2 Hot End 💌
Build Width: 305.00 🔹 mm	Right Extruder: [Raise3D] FLA 1.75mm - V2 Hot End 🔻
Build Depth: 305.00 🖨 mm	Extruder Offset X: 25.00 💂 mm
Build Height: 305.00 🔹 mm	Extruder Offset Y: 0.00 🗭 mm
🗹 Use Heated Bed	
🗹 Enable Fan Speed Control (Use PWM-Controllable Fans)	If both Extruder Offset X and Y above are set to 0, the following values will be used to check whether models are placed outside the extruder's printing range:
Distance From Border of Build Flate: 2.50 🗼 mm	Virtual Extruder Offset X: 25.00 👘 mm
Build Plate Shape: Rectangle 💌	Virtual Extruder Offset Y: 0.00 🔺 mm
Firmware: Marlin 💌	
Add / Duplicate Import	Save Close

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*:

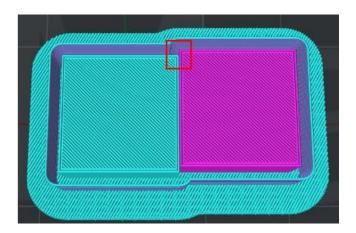


Figure 6.7: The right calibration box is backward than the left calibration box.

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.



Printer Settings						? ×
Printer Type:	RAISE3D N2 - V2 Hot	End	•	Remove	Export	Reset
General Advanc	ed					
Printer Setting	5		Extruders			
Left Nozzle Di	ameter:	0.40 🛉 mm	Extruder Count:	2	•	
Right Nozzle D	iameter:	0.40 🛖 mm	Left Extruder:		Raise3D] PLA 1.75mm	n - V2 Hot End 🔻
Build Width:		305.00 후 mm	Right Extruder:	[]	Raise3D] PLA 1.75mm	n - V2 Hot End 🔻
Build Depth:		305.00 🗭 mm	Extruder Offset X:		25.00 🖨 mm	
Build Height:		305.00 🔹 mm	Extruder Offset V:		0.00 🗘 mm	
🗹 Use Heated :	Bed					
🗹 Enable Fan :	Speed Control (Use PWM	-Controllable Fans)	If both Extruder Off values will be used extruder's printing	to check whether		
Distance From 3	Border of Build Plate:	2.50 mm	Virtual Extruder Off	ŝset X:	25.00 * mm	
Build Plate Sh	ape:	Rectangle 🔻	Virtual Extruder Off	iset Y:	0.00 🔺 mm	
Firmware:		Marlin 🔻				
Add / Duplicate	Import				Save	Close

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*:

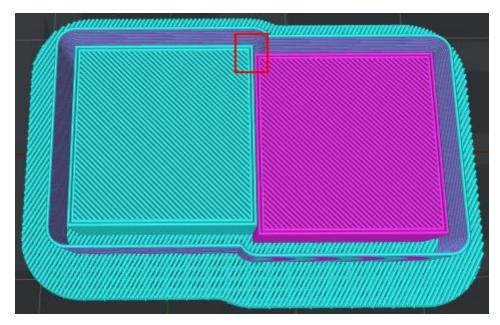


Figure 6.9: The right calibration box is more forward than the left calibration box.



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.

Printer Settings		? ×
Printer Type: RAISE3D N2 - V2 Hot End	▼ Remove	Export Reset
General Advanced		
Printer Settings	Extruders	
Left Nozzle Diameter: 0.40 👘 mm	Extruder Count:	2 👻
Right Nozzle Diameter: 0.40 👘 mm	Left Extruder:	[Raise3D] PLA 1.75mm - V2 Hot End 💌
Build Width: 305.00 👘 mm	Right Extruder:	[Raise3D] PLA 1.75mm = V2 Hot End 💌
Build Depth: 305.00 mm	Extruder Offset X:	25.00 💼 mm
Build Height: 305.00	Extruder Offset V:	0.00
🗹 Use Heated Bed	If both Extruder Offset X and Y a	have any set to 0, the full mine
☑ Enable Fan Speed Control (Use PWM-Controllable Fans)	values will be used to check whet extruder's printing range:	
Distance From Border of Build Plate: 2.50 👘 mm	Virtual Extruder Offset X:	25.00 mm
Build Plate Shape: Restangle 🔻	Virtual Extruder Offset V:	0.00 🛖 mm
Firmware: Marlin 💌		
Add / Duplicate Import		Save Close

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.

Select Template			?
Printer Type:	RAISE3D N2 Plus - V2	2 Hot End 🔹	¢
Filament			
Left Extruder:	PLA 1.75mm - V2 Hot	End (3 templates)	¢
Right Extruder:	PLA 1.75mm - V2 Hot	End 👻	\$
Right Extruder: Main Template G	PLA 1.75mm - V2 Hot roup and Layer Settings		\$
	roup and Layer Settings		\$
Main Template G	roup and Layer Settings ation_Box.stl	Per-Model Extruder	۵

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.

Add	XII Delete	⊙ View	W Pan	↔ Move	Rotate	L Scale	Fr ee Cut	Support
\ //								
//								
	Model Info			×				
	Extruder:	Left Ex	truder		-	_7\		
í E	Color:				+	-(2
	Show Ex	truder's Pi	inting Ran	ge		1		
<u>ئ</u>					/ V_ /	V		

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



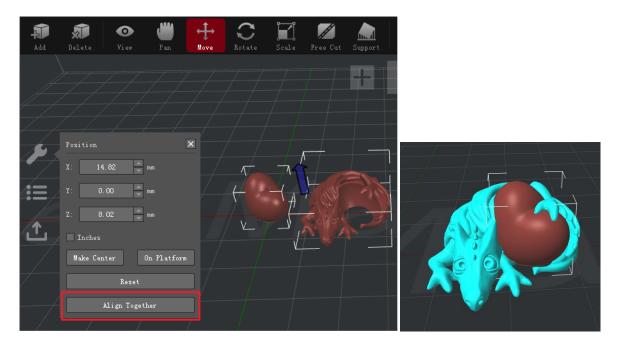


Figure 6.13: Try "Align Together" to align the models with "Move" enabled. If it doesn't work well, try to move the models manually.

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

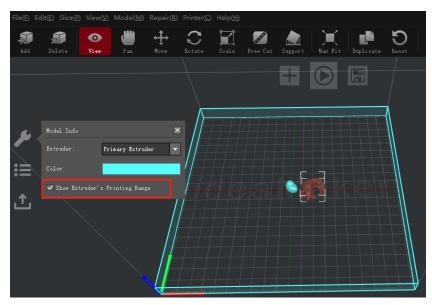


Figure 6.14: Check Right Extruder's printing range.



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.

Layer Extra	ıder Infill	Support	Platform Addition	s Coolin	g Temperature	Advanced	Ooze	Other	GCode
Generate Su	pport:	All	•						
Support				_	Dense Support				
Support Ext	ruder:		Left Extruder Left Extruder	-	Dense Suppor	t Layers:			0
Support Typ	e:		Right Extruder		Dense Suppor	t Infill Rati	o:		80 🗘 %
Figu Layer Extrude		L 5: S Support	Select an Platform Additions	extr Cooling		o prir dvanced 002			
Platform Addi	tion:	Raft only	•						
Raft				B	aft Interface Lay	rer			
Raft Extruder	:		Right Extruder Left Extruder	•	Interface Layers:			1	•
Raft Offset:			Right Extruder		Interface Layer S	peed:		30.	0 🔶 mm/s

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).

Lay	/er	Extruder	Infill	Support	Platform	Additions	Cooling	Temperature	Advanced	Ooze	Other	GCode
_ 1	empe	rature										
	Heate	ed Bed Temper	rature:		þ0	🗘 o c						
ſ	Left	Extruder:			205	÷°c						
	Ri ght	t Extruder:			205	÷°C						

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).

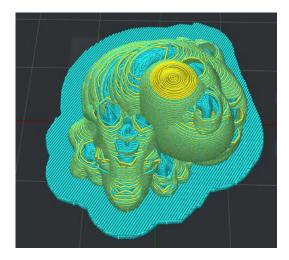


Figure 6.18: Set "Wipe Wall Angle" to be 90.



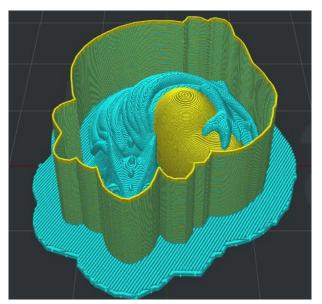


Figure 6.19: Set "Wipe Wall Angle" to be 0.

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

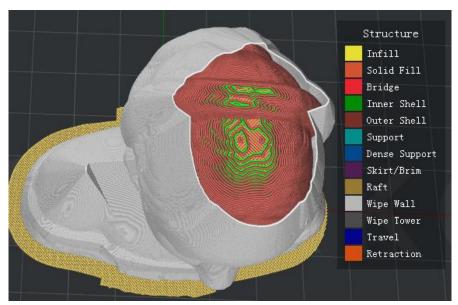


Figure 6.20: Set "Wipe Wall Loop Lines" to be one.



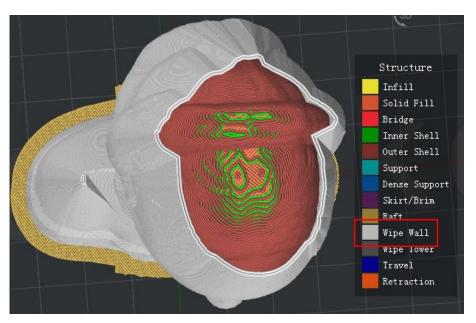


Figure 6.21: Set "Wipe Wall Loop Lines" to be two.

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.

WipeWall Type:

Contoured	-
Contoured	
WaterFall	
Vertical	

Figure 6.22: Three types of Wipe Wall.



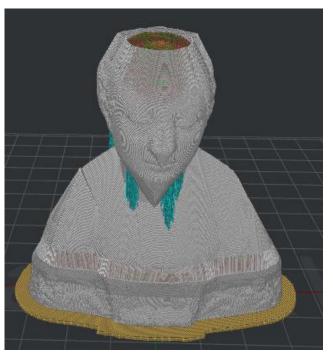


Figure 6.23: Set "Wipe Wall Type" to be "Contoured".

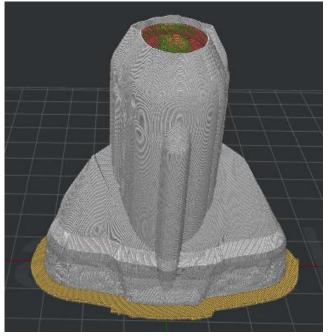


Figure 6.24: Set "Wipe Wall Type" to be "WaterFall".



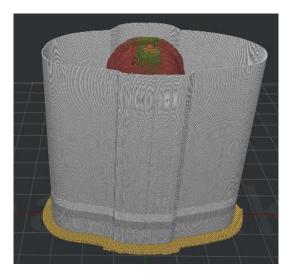


Figure 6.25: Set "Wipe Wall Type" to be "Vertical".

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.

🕤 🗹 Placed at the Fixed Position of the	Platform		
Wipe Tower Offset X:	-2.00	*	mm
Wipe Tower Offset Y:	2.00	-	mп
Wipe Tower Fixed Position X:	27.50	•	mп
Wipe Tower Fixed Position Y:	280.00	•	mm

Figure 6.26: Wipe Tower Fixed Position.



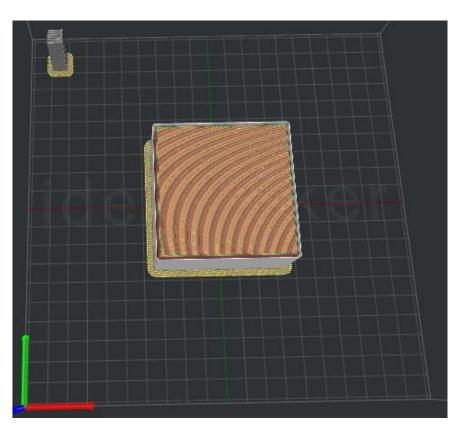


Figure 6.27: Enable the function of Placed at the Fixed of the Platform.

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.

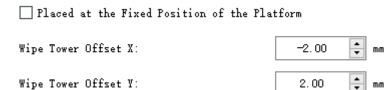
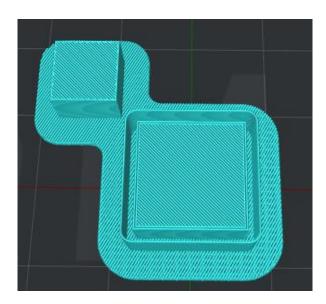
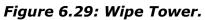


Figure 6.28: Wipe Tower X/Y offset.







ver Extruder Infill Support Platform Additions Cool	ing Temperature Advanced Ooze	Other GCode
Other	🖂 Wipe Wall	
Force Retraction On Layer Change	Wipe Wall Offset:	2.00 mm
☐ Force Retraction Before Traveling to Outer Shell	Wipe Wall Angle:	30 🛉 Deg
🗹 Avoid Retraction Inside Models	Wipe Wall Loop Lines:	1
☑ Enable Retraction at Bottom and Top of Models	Wipe Wall Type:	Contoured 💌
Bottom Layers: O	Wipe Wall Speed:	50.0 🛉 mm/s
Top Layers: 2	🕤 🗹 Wipe Tower	
xtruder Switch Ooze Control	Wipe Tower Width:	12.00 m
Retraction Speed of Extruder-switch: 20.0 💌 mm/	Wipe Tower Infill Ratio:	100 🔹 %
Retraction Amount of Extruder-switch: 3.50 💌 mm	Wipe Tower Speed:	50.0 🛉 mm/s
Restart Speed of Extruder-switch: 20.0 💌 mm/	Belaced at the Fixed Position	a of the Platform
Extra Restart Amount of Extruder-switch: 0.00 💌 mm	Wipe Tower Offset X:	-2.00 mm
ravel	Wipe Tower Offset Y:	2.00 mm
Avoid Traveling Through Holes	Wipe Tower Fixed Position X:	27.50 mm
Maximum Travel Path Length: 200 🔹 %	Wipe Tower Fixed Position Y:	280.00 🛉 mm

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

Print	ter(<u>C)</u> Help(<u>H</u>)	
	Current Type of Printer	۲
	Connect to Printer (Raise3D N-series)	
	ideaPrinter F-series	F
	Printer Settings	
	Filament Settings	
	Configuration Wizard	

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

Filament:	PLA 1.75mm	▼ Remove	Export Reset
Filament Setti	ngs	Override Slicing Settings in Templ	ates Linked to the Filament
Filament Type	PLA 1.75mm	✓ Temperature	
		Extruder Temperature	225 °C
Br and :		Use Temperature Control List	🗹 Enabled
Display Name:	PLA 1.75mm		
Diameter:			
Diameter:	1.750 🛉 mm		
Density:	1240.00 * kg/m3		
Price:	30.00 ¥/kg		
Flowrate:	94.0 🔹 %		
Notes			
		+ -	

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*.

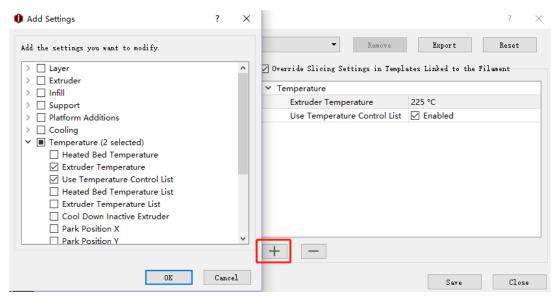


Figure 7.3: Override Slicing Settings.



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

	Platform Additions Co	-		Code	Preview Connect
Tenperature		_	Down Inactive Extruder	يغو	
Heated Bed Temperature:	60 🕂 ° C	Edit - "PLA 1.75r	nm"		? ×
Left Extruder:	💉 205 🐳 ° C	Filament Type:	PLA 1.75mm		
Right Extruder:	N 205 🗘 ° C	Filament Settings		🗸 Override Slicing Settings in Templates	Linked to the Filament
🖋 🗌 Use Temperature Control List		Br and :		✓ Temperature	
		Display Name:	PLA 1.75mm	Extruder Temperature 22 Use Temperature Control List	5 °C Enabled
Heated Bed Left Extruder Rig	ht Extruder	Diameter:	1.750 🗘 nm		Lindbed
Layer: 1 🔺	Layer Temperati	Density:	1240.00 * kg/m3		
Temperature: 40 * °C		Price:	30.00 🗘 ¥/kg		
		Flowrate:	94.0 * %		
Add Temperature		Notes			
Remove Temperature					
				+ -	
					Save Cancel
					s: 12 2 fold Edges: 0

Figure 7.4: Edit the override settings in Filament Settings.

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

Moc 🌖 Sele	ct Template					?	×	
Printe Par	er Type:	RAISE3D Pro2			-	ž		
-Filane	ent							
Left H	lxtruder:	PLA 1.75mm (25 templates)			-	⊁		
Right	Extruder:	PLA 1.75mm			- 4	⊁		
🟮 Filament Sett	ings						?	×
Filament:	PLA 1.75mm		•	Remove	Export	Re	set	
Filament Setti	ngs		🗸 Override Slicing	Settings in Templ	ates Linked to	the Filar	ment	
Filament Type	: PLA 1	. 75mm	✓ Temperature					
Brand:			Extruder Ten	perature	225 °C			
brand.			Use Tempera	ature Control List	🗹 Enabled			
Display Name:	PLA 1	. 75mm						
Diameter:		1. 750 🔹 mm						
Density:	1	240.00 🛊 kg/m3						
Price:		30.00 🔹 ¥/kg						
Flowrate:		94.0 🗘 %						
Notes								
			+ -					
	_							
Add / Duplica	te Im;	port			Save		Clos	e

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



7.2 Add/Duplicate Filament

/ilament: PL/	A 1.75mm		•	Remove	Export	Reset
Filament Settings			🗹 Override Slicing	Settings in Te	nplates Linked to the	e Filament —
Filament Type:	PLA 1.75mm		✓ Temperature			
Brand:			Extruder Ter		225 °C	
Di alla.				ature Control Li	st 🔽 Enabled	
Display Name:	PLA 1.75mm	🚺 Add Fila	ament		? ×	
Diameter:	1. 750	mm 💿 Duplica	ate the selected filame	ent		
Density:	1240.00	kg				
Price:	30.00	○ Add a r ¥	new filament			
Flowrate:	94.0	96		Next	Cancel	
Notes						
			+ -			

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**.

🚺 Add Filament			? ×
Filament Type:	PLA 1.75mm=copy		
-Filament Settin	gs	🔽 Override Slicing Settings in Templ	ates Linked to the Filament
Br and :		✓ Temperature	
Display Name:	PLA 1.75mm-copy	Extruder Temperature	225 °C
Display Mame.	TLA I. /Dmm=copy	Use Temperature Control List	✓ Enabled
Diameter:	1.750 🔹 mm		
Density:	1240.00 k g/m3		
Price:	30.00 ¥/kg		
Flowrate:	94.0 🗳 %		
Notes			
		+ -	
			OK Cancel

Figure 7.7: Duplicate the selected filament.



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

🚺 Add Filament		? >	<
Filament Type:	new filament		
-Filament Settings		Override Slicing Settings in Templates Linked to the Filament	
Br and :			
Display Name:			
Diameter:	1.750 mm		
Density:	1240.00 🔹 kg/m3		
Price:	30.00 🗘 ¥/kg		
Flowrate:	100.0 🔹 %		
Notes			
		+ -	
		0K Cancel	

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.

🚺 Filament Setti	ngs	? ×
Filament:	PLA 1.75	Remove Export Reset
-Filament Setti	ngs	Override Slicing Settings in Templates Linked to the Filament
Filament Type:	PLA 1.75	
Br and :		
Display Name:	PLA 1.75	
Diameter:	1.750 mm	
Density:	1240.00 🔹 kg/m3	
Price:	30.00 ¥/kg	
Flowrate:	94.0 🔹 %	
Notes		
		+ -
Add / Duplicat	te Import	Save Close

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.

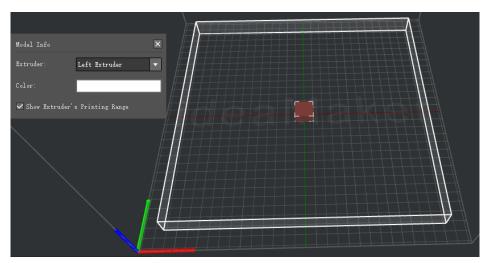


Figure 8.1: Set Distance From Border of Build Plate 20 mm.

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.

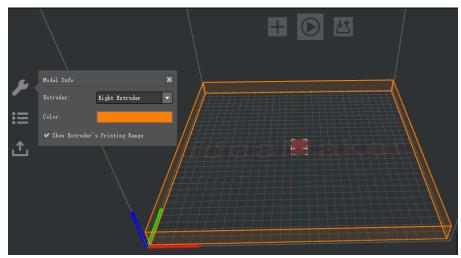
Virtual Extruder Offset Y refers to the right extruder's offset in Y axis.

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*.







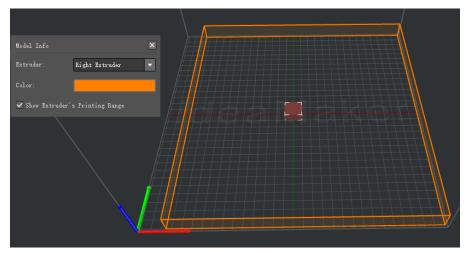


Figure 8.3: Set Virtual Extruder Offset Y to be 25 mm.

ter Type:	RAISE3D Pro2 Plu	15 (modified)	▼ Remo	Export Reset
eral Adva	nced			
rinter Setti	ngs		Extruders	
left Nozzle I	liameter:	0.40 🔹 mm	Extruder Count:	2 🔻
Right Nozzle	Diameter:	0.40 💼 mm	Left Extruder:	PLA 1.75mm
Build Width:		305.00 m m	Right Extruder:	PLA 1.75mm
Build Depth:		305.00 🗭 mm	Extruder Offset X:	0.00 💼 mm
Build Height:		605.00 🗭 mm	Extruder Offset V:	0.00 mm
🗹 Use Heated	l Bed		Tf both Butunder Officet V en	d Y above are set to 0, the following
🗹 Enable Far	a Speed Control (Use	e PWM-Controllable Fans)		whether models are placed outside the
listance From	Border of Build Pl	.ate: 2.50 🛉 mm	Virtual Extruder Offset X:	25.00 🗭 mm
Build Plate S	hape:	Ellipse 🝷	Virtual Extruder Offset V:	0.00 🜩 mm
'irmware:		Raise3D Speedy 💌		

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.

nter Type:	RAISE3D Pro2 Plus (modified)	 Remove Export Re 	set
			and the second s	
eneral Advan	iced			
Other Settings			Custom Command	
Step-E per MM	:	0.00	🗹 Omit RaiseTouch Specified GCode	
X-axis Compens	sation:	100.00 🔹 %	End of Start GCode:	
Y-axis Compens	sation:	10þ.00 🔦 %	Beginning of End GCode:	
Baud Rate:		230400	Pause at Meight GCode:	
Temperature Li	mit			
Heated Bed Ma	ximum Temperature Limit:	110 🔹 ° C		
Left Extruder	Maximum Temperature Limit:	310 🔹 ° C		
Right Extruder	r Maximum Temperature Limit:	310 🔹 ° C		
			,•	

Figure 8.4: Custom Command.



8.1 Edit Printer Settings

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

Print	ter(<u>C</u>)	Help(<u>H</u>)	
	Curre	ent Type of Printer	۲
	Raise	3D N-series	۲
	idea	Printer F-series	۲
	Printe	er Settings	
	Filam	ent Settings	
	Confi	guration Wizard	

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

Printer Settings						?	×
rinter Type:	RAISE3D Pro2 (mod	ified)	-	Remove	Export	Reset	
General Advan	iced						
-Printer Settin	gs		Extruders				
Nozzle Diamet	er:	0.40 🖍 mm	Extruder Count:	1	•		
Build Width:		305.00 🔹 mm	Primary Extruder:	PI	"A 1.75mm		•
Build Depth:		305.00 🛉 mm					
Build Height:		300.00 🛉 mm					
🗹 Vse Heated	Bed						
🗹 Enable Fan	Speed Control (Use H	WM-Controllable Fans)					
Distance From	Border of Build Plat	te: 2.50 🛉 mm					
Build Plate Sl	hape:	Rectangle 🔹					
Firmware:		Raise3D Speedy 🔻					
Add / Duplicate	Import				Save	Clos	se

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



Option2: Enter Select template tab and click setting button.

ew Moc 🛈 Sele	ect Template				?	×	
Print	er Type:	RAISE3D Pro2			•		
Printer Settings						?	×
Printer Type:	RAISE3D Pro2	(modified)	•	Remove	Export	Reset]
General Advar	rced						
Printer Settin	ıgs		Extruders				
Nozzle Diamet	er:	0.40 🛉 mm	Extruder Count:	1	•		
Build Width:		305.00 m	Primary Extruder:	PLA	1.75mm		•
Build Depth:		305.00 🛉 mm					
Build Height:		300.00 🗭 mm					
🗹 Use Heated	Bed						
🗹 Enable Fan	Speed Control ((Use PWM-Controllable Fans)					
Distance From	Border of Build	l Plate: 2.50 🛉 mm					
Build Plate S	hape:	Rectangle 🔹					
Firmware:		Raise3D Speedy 💌					
Add / Duplicate	Import				Save	Clo	se

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*

Printer Settings		? ×
Printer Type: RAISE3D Pro2	•	Remove Export Reset
General Advanced		
Printer Settings	Extruders	
Nozzle Diameter:	0.40 🔹 mm Extruder Count:	1 -
Build Width:	305.00 💼 mm Primary Extruder:	PLA 1.75mm 👻
Build Depth:	305.00 🗭 mm	
Build Height:	300.00 🛊 🖿 mm	? ×
✓ Use Heated Bed	Duplicate the sel	ected printer
🗹 Enable Fan Speed Control (Use PWM-Cont	trollable Fans) 🔿 Add a new printer	
Distance From Border of Build Plate:	2.50 mm	
Build Plate Shape: Rec	ctangle 🔻	Next Cancel
Firmware: Rai	ise3D Speedy 🔻	
Add / Duplicate Import		Save Close

Figure 8.8: Add/Duplicate Printer.

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

Printer Type: RAISE3D Prof	0	Extruder Count:	1 👻	
Trinter Type. RAISESD FFG.	2-copy	Extrater count.	•	
Nozzle Diameter:	0.40 🗭 mm	Primary Extruder:	PLA 1.75mm	•
Build Width:	305.00 👘 mm			
Build Depth:	305.00 🔹 mm			
Build Height:	300.00 👘 mm			
🗹 Vse Heated Bed				
🖉 Enable Fan Speed Contro	l (Use PWM-Controllable Fans)			
Distance From Border of Bu	ild Plate: 2.50 👘 mm			
Build Flate Shape:	Rectangle 🔹			
Firmware:	Raise3D Speedy 💌			

Figure 8.9: Duplicate the selected printer.



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

🚯 Add Printer				?	×
General Advanced					
Printer		Extruder			
Printer Type: new printer		Extruder Count:	1 -		
Nozzle Diameter:	0.40 🔹 mm	Primary Extruder:	PLA 1.75mm	•	
Build Width:	305.00 m				
Build Depth:	305.00 🗘 mm				
Build Height:	305.00 🛉 mm				
🗹 Use Heated Bed					
🗹 Enable Fan Speed Control (Use PWM	-Controllable Fans)				
Distance From Border of Build Plate:	2.50 🔹 mm				
Build Flate Shape:	Rectangle 🔹				
Firmware:	Marlin 🔻				
			OK	Cancel	1
			- OII	Janoe	-

Figure 8.10: Add a new printer.

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

w Moc		?	×
Printer Type: RAISE3D Fro2		-	Provide Provide
Printer Settings			? ×
Printer Type: RAISE3D Pro2 (modified)	▼ Remove	Export	Reset
General Advanced			
Printer Settings	Extruders		
Nozzle Diameter: 0.40 💼 mm	Extruder Count:	1 👻	
Build Width: 305.00 🔺 mm	Primary Extruder:	PLA 1.75mm	•
Build Depth: 305.00 🛊 mm			
Build Height: 300.00 🚖 mm			
✓ Use Heated Bed			
🗹 Enable Fan Speed Control (Use PWM-Controllable Fans)			
Distance From Border of Build Plate: 2.50 💌 mm			
Build Flate Shape:			
Firmware: Raise3D Speedy 🔻			
Add / Duplicate Import		Save	Close

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.

Printer Settings					?	×
Printer Type: RAISE3D Pro2		•	Remove	Export	Reset	
General Advanced						
Printer Settings		Extruders				
Nozzle Diameter:	0.40 m	Extruder Count:		1 👻		
Build Width:	305.00 🗭 mm	Primary Extruder:	1	PLA 1.75mm		•
Build Depth:	305.00 🛉 mm					
Build Height:	300.00 🛉 mm					
🗹 Use Heated Bed						
🗹 Enable Fan Speed Control (Use PWM	-Controllable Fans)					
Distance From Border of Build Plate:	2.50 🗭 mm					
Build Plate Shape:	Rectangle 🔻					
Firmware:	Raise3D Speedy 🔻					
Add / Duplicate Import				Save	Clos	e

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.

neral Advar	aced			
Printer Settin	igs		Extruders	
Left Nozzle D	iameter:	0.40 🛉 mm	Extruder Count:	2 🗸
Right Nozzle :	Diameter:	0.40 🛉 mm	Left Extruder:	PLA 1.75mm -
Build Width:		305.00 🛉 mm	Right Extruder:	PLA 1.75mm -
Build Depth:		305.00 🛉 mm	Extruder Offset X:	0.00 🛉 mm
Build Height:		605.00 🗭 mm	Extruder Offset Y:	0.00 mm
🗹 Use Heated	Bed		If both Bytruder Offret V and	Y above are set to 0. the following
🗹 Enable Fan	Speed Control (Use PW	M-Controllable Fans)		hether models are placed outside the
Distance From	Border of Build Plate	: 2.50 🗭 mm	Virtual Extruder Offset X:	25.00
Build Plate S	hape:	Rectangle 🔻	Virtual Extruder Offset V:	0.00 🛉 mm
Firmware:		Raise3D Speedy 💌		

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.



Remove Printer - "RAISE3D N2 Plus - V2 H	Hot End-copy"				
The following templates associated with the printer will be removed.					
Flease check the templates you want to rem	107e.				
(Double click template to view settings)					
▲ 🔽 PLA 1.75mm - V2 Hot End	Move to the following printer and filament				
✓ New Template12	Template: New Template12				
ABS 1.75mm - V2 Hot End					
New Template11	Printer: T3 💌				
	Filament: PLA 1.75mm - V2 Hot End -				
Remove All Templates 1 temp					
Kemove ALL Templates I temp	plate(s) will be removed.				
	OK Cancel				

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.

Remove Printer - "RAISE3D N2 Plus - V2 Hot End-copy"	1 Remove Printer - "RAISE3D N2 Plus - V2 Hot End-copy"
The following templates associated with the printer will be removed. Please check the templates you want to remove.	The following templates associated with the printer will be removed. Flease check the templates you want to remove.
Oruble click template to view settings) PLA 1.75mm - V2 Hot End New Template12 ABS 1.75mm - V2 Hot End New Template11 New Template1 Remove All Templates	Would click tesplate to view settings) Image: Setting of the
OK Cancel	OK Cancel

Figure 8.15: Remove All Templates.

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



🚯 Remove Printer - "RAISE3D N2 Plus - V2 Hot	Remove Printer - "RAISE3D N2 Plus - V2 Hot End-copy"					
The following templates associated with the printer will be removed.						
Please check the templates you want to remove						
(Double click template to view settings)						
▲ ▼ PLA 1.75mm - V2 Hot End	Move to the	following printer and filament —				
Vew Template12	Template:	New Template12				
ABS 1.75mm - V2 Hot End						
New Template11	Printer:	ТЗ	•			
	Filament:	PLA 1.75mm - V2 Hot End	•			
Remove All Templates 1 templa	te(s) will be re	moved.				
		OK	Cancel			

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.

1 Remove Printer - "RAISE3D N2 Plus - V2 Hot End-copy"	0 Select Template
The following templates associated with the printer will be removed.	Printer Type: T3
Flease check the templates you want to remove.	Filment Left Extruder: ASS 1.75mm - V2 Not End (1 template)
(Double click template to view settings) Image: PLA 1.75mm - V2 Hot End Image: New Template12 Image: New Template11 Image: Plant 1.75mm - V2 Hot End Image: Plant 1.75mm - V2 Hot End	Ref Lick upper Ref Lick upper Right Extruder: FE A 1 75me - V2 Not End Right Extruder: FE A 1 75me - V2 Not End Main Template: Group and Layer Settings File New Template11 Duplicate Zait Salect Template: Her Template11 Layer Kright: 10 m Salect Template: Ker Template11 Layer Kright: 0.10 m Shelt: 2 Infill Density: 15 0 %
OK Cancel	Close Slice

Figure 8.17: Remove the selected Templates.





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

177 / 177