ideaMaker Manual





1	Using ideal	Maker
	1.1 What	t is ideaMaker?
	1.2 Whe	re to download ideaMaker?
2	Install idea	Maker4
3	Let's Print!	
4	How to use	e ideaMaker?
	4.1 Inter	face
	4.1.1	Menu Bar
	4.1.2	Tool Bar
	4.1.3	Operation property
	4.1.4	Model list
	4.1.5	Uploading queue51
	4.1.6	Instant operation bar
	4.1.7	Model preview 53
	4.1.8	Perspective transformation
	4.1.9	Detecting Information54
	4.2 Slicir	ng settings
	4.2.1	Layer
	4.2.2	Infill
	4.2.3	Support
	4.2.4	Raft
	4.2.5	Cooling
	4.2.6	Ooze
	4.2.7	Other
	4.2.8	GCode
	4.3 Multi	ple Extruders
	4.4 Filam	nent Settings
	4.5 Print	er settings



1 Using ideaMaker

1.1 What is ideaMaker?

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

1.2 Where to download ideaMaker?

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



2 Install ideaMaker

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



Figure 2.1: Choose a language for installing ideaMaker.



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

0 ideaMaker 2.5.1	1.785 Setup	
	Choose Install Location	
\mathbf{U}	Choose the folder in which to install	lideaMaker 2.5,1.785.
Setup will install ide Browse and select	aMaker 2.5.1.785 in the following folder.To in another folder. Click Next to continue.	stall a different folder, dick
Destination Folde	r :s\Raise3D\ideaMaker\	Browse
Space required: 13	14.4MB	
Space available: 19	9.8GB	
Raise3D www.raise3d	.com - ideaMaker 2,5,1.785	
	< Back	Next > Cancel

Figure 2.2: Choose a path to install your ideaMaker.



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

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

0 ideaMaker 2.5.	L.785 Setup	• • •
Û	Choose Components Choose which features of ide install.	eaMaker 2, 5, 1, 785 you want to
Check the compon install. Click Install	ents you want to install and uncheck the to start the installation.	e components you don't want to
Select components	to install: File Association Microsoft Visual C++ Printer Driver	2008 SP 1 Redistributable
Space required: 13	14.4MB	
Raise3D www.raise3u	.com - ideaMaker 2,5,1,785	c Install Cancel

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



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

0 ideaMaker 2.5.1.7	85 Setup		
Û	Installation Complete Setup was completed sur	ccessfully.	
Completed			
Show details			
Raise3D www.raise3d.co	m - ideaMaker 2,5,1,785	Back Next	> Cancel

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



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



Figure 2.5: Complete the installation.



3 Let's Print!

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

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



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



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



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

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

	r	
Printer Type:	RAISE3D N2 - V2 Hot End	Settings
	ideaPrinter F100	*
Filament	ideafrinter FlUUL	
	ideaPrinter F2	
Primary Extruder:	RAISE3D N1	E
	RAISE3D N2	
	RAISE3D N2 Plus	
	RAISESD NI = V2 Hot End	
N. 22	RAISE3D N2 Plus - V2 Hot End	- Duplicate
High Que		
11.1		(
- A 10 1		Edit
Standard	- N2 - PLA	
A		
🦂 Speed - I	N2 - PLA	Delete
Speed - I	N2 - PLA	Delete
Speed - I Select Template	N2 - PLA	Delete
Speed - I Select Template Printer Type:	N2 - PLA RAISE3D N2 - V2 Hot End	Delete P Settings
Speed - I Select Template Printer Type: Filament	N2 - PLA RAISE3D N2 - V2 Hot End	Delete B Settings
Speed - I Select Template Printer Type: Filament Primary Extruder:	N2 - PLA RAISE3B N2 - V2 Hot End PLA 1.75mm - V2 Hot End	Delete P Settings
Speed - I Select Template Printer Type: Filament Primary Extruder:	N2 - PLA RAISE3D N2 - V2 Hot End PLA 1.75mm - V2 Hot End	Delete P Settings
Speed - I Select Template Printer Type: Filament Primary Extruder:	RAISE3D N2 - V2 Hot End PLA 1.75mm - V2 Hot End PLA 1.75mm ABS 1.75mm	Delete P Settings
Speed - I Select Template Printer Type: Filament Primary Extruder:	RAISE3D N2 - V2 Hot End PLA 1.75mm - V2 Hot End PLA 1.75mm PLA 1.75mm Pla 1.75mm PolyPC 1.75mm	Delete P Settings
Speed - I Select Template Printer Type: Filament Primary Extruder:	RAISE3D N2 - V2 Hot End PLA 1.75mm - V2 Hot End PLA 1.75mm ABS 1.75mm PolyPlex 1.75mm PolyPlex 1.75mm PolyPlex 1.75mm	Delete Tunlicate
Speed - I Select Template Printer Type: Filament Primary Extruder:	RAISE3D N2 - V2 Hot End PLA 1.75mm - V2 Hot End PLA 1.75mm PolyPC 1.75mm PolyPlex 1.75mm PolyPlex 1.75mm PolySupport 1.75mm Plastic Filement 1.75mm	Delete P Settings Duplicate
Speed - I Select Template Printer Type: Filament Primary Extruder:	RAISE3D N2 - V2 Hot End PLA 1.75mm - V2 Hot End PLA 1.75mm PolyPC 1.75mm PolyPC 1.75mm PolySupport 1.75mm Elastic Filament 1.75mm Elastic Filament 1.75mm	Delete P Settings Duplicate
Speed - I Select Template Printer Type: Filament Primary Extruder:	RAISE3D N2 - V2 Hot End PLA 1.75mm - V2 Hot End PLA 1.75mm PolyPC 1.75mm PolyPUex 1.75mm PolySupport 1.75mm Elastic Filament 1.75mm HIFS 1.75mm PETG 1.75mm	Delete P Settings Duplicate Edit
Speed - I Select Template Printer Type: Filament Primary Extruder: High Qua	RAISE3D N2 - V2 Hot End PLA 1.75mm - V2 Hot End PLA 1.75mm PolyPC 1.75mm PolyPLex 1.75mm PolySupport 1.75mm Elastic Filament 1.75mm HIPS 1.75mm PERG 1.75mm PLA 1.75mm PLA 1.75mm PLA 1.75mm PLA 1.75mm PLA 1.75mm PLA 1.75mm	Delete The settings Duplicate Edit
Speed - I Select Template Printer Type: Filament Primary Extruder: High Qua	RAISE3D N2 - V2 Hot End PLA 1.75mm - V2 Hot End PLA 1.75mm PolyPLa 1.75mm PolyPLa 1.75mm PolyPLa 1.75mm PolyPLa 1.75mm Elastic Filament 1.75mm HIPS 1.75mm PETG 1.75mm PETG 1.75mm PETG 1.75mm PTA 1.75mm - V2 Hot End ABS 1.75mm - V2 Hot End	Delete

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



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

Select Template		? <mark>- ×</mark>
Printer Type: RAISE3D N2 - V2 Ho	t End	▼ Settings
Filament		
Primary Extruder: PLA 1.75mm - V2 Ho	t End 🔻	
High Quality - N2 - PLA		Duplicate
Standard - N2 - PLA		Edit
Speed - N2 - PLA		Delete
		Import
		Export
		Đ
Select Template: Speed - N2 - PLA		
Layer Height: 0.25 mm	Shells: 2	Cancel
Fill Density: 10.0 %	Infill Speed: 90.0 mm/s	Slice

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



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

lemplate Name: Spee	I = NZ = PLA	i.
General		
Fill Density:	10 🛸 %	
Shells:	2	
Platform Addition:	Raft only 🔻	
Support:	None 🔻	
Advanced	Restore Defaults	

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



6. Click *Slice* button.



Figure 3.6: Click "Slice" to start slicing.



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

Prepared Print File:	rabbit.gcode	
Estimated Print Time:	24 hours, 2 min, 46 sec	
Estimated Amount:	379.6 g / 127.28 m	
Estimated Price:	\$ 11.39	
Note: Result above is o	nly for reference.	
		122

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

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



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



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

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



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



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



Figure 3.10: Display different structures with different colors.



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



9. Close the preview dialogue box after the confirmation.

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

Option1: Export to USB storage or SD card

Export the sliced files to USB storage or SD card.

Prepared Print File:	rabbit, gcode
Estimated Print Time:	24 hours, 2 min, 46 sec
Estimated Amount:	379.6 g / 127.28 m
Estimated Price:	\$ 11.39
ote: Result above is o	nlv for reference.

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

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



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



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



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



Option2: Upload the sliced file via WLAN.

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

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



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



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

<u>ج</u>		 (•			1			180
	🔅 Setting		🔅 Setting					
Machine	Machine Name	Machine	WLAN 🚺	₽≎	Machine	WEAN	0	
0	Model	0	+ Add Other Netv	/ork	9	*****	0	
Camera	Serial Number	Camera	Choose A Network		Carrie		c 1	
Æ	Version	Æ	B54F_guest	A ?	a.	Connect	Cancel	
Ethernet	Update	Ethernet	B54F	₽ 🗢	Ethernet			
	Firmware Version		FORZA	<u>₽</u> ?				
WLAN	Storage Available	WLAN	Cobetter-sh	÷ ≜≎	WLAN			
	Restart				q w	erty	/ u i	о р
Other	More Settings	Other	174		a s	d f g	h j	k I
	Close		Close		▲ z	x c v	b n	m 🙁
					.?123			Ē

Figure 3.16: Connect the printer with your WLAN.

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

Estimated Print Result		8
Prepared Print File:	rabbit, gcode	
Estimated Print Time:	24 hours, 2 min, 46 sec	
Estimated Amount:	379.6 g / 127.28 m	
Estimated Price:	\$ 11.39	
ote: Result above is o	nly for reference.	
Preview	Imload	Export



20mm_Cali	bration_Box.gco	de		
Printer:	Raise3D N1	- Rai@192.168.0.58	•	Refresh
Type:	Raise3D N1	- Rai@192.168.0.58	^	<u>. </u>
Name:	Raise3D N2P	us - Raise3D@192.168.0.152		
Address:	Raise3D N1	- Raise3D@192.168.0.153		
	Raise3D N1	- Raise3D@192.168.0.128		
	Raise3D N1	- Raise3D@192.168.0.172	E	Cancel
1	Raise3D N2	- Raise3D@192.168.0.145		
	Raise3D N2	- Raise3D@192.168.0.109		

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

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



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



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



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



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

Pr	nter(C) Help(H)	
	Current Type of Printer	82
	Connect to Printer (Raise3D N-series)	
	ideaPrinter F-series	
	Printer Settings	
	Filament Settings	
	Configuration Wizard	

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

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

Printer Connect					
	🔁 Remot	e Connecti	on		
Bookmarks		Ð	Scanning		
Raise3D N1	Rai 192.168.0.58			X	()
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	(\mathbf{P})
	1	/ 3	>		

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



7. The connecting page.

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

Figure 3.22: Wait for connecting.

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



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



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





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



4 How to use ideaMaker?

4.1 Interface

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



Figure 4.1: An overview over the ideaMakerinterface.



4.1.1 Menu Bar

Menu bar includes all the operation commands and advanced setting.

1>File

File(E) Edit(<u>E</u>) Slice(<u>P</u>)	View(V) Mod	
	New(N)	Ctrl+N	Create a new empty workspace
5	Open Idea File(<u>O</u>)	. Ctrl+O	→Open a .idea file (which is used for old version of F-series p
	Close(W)	Ctrl+W	← Close current file
	Save Idea File(<u>S</u>)	Ctrl+S	►► Save file
	Save Idea File As	8	►►Save project file as .idea
	Import Model	•	➡ Import a .stl file
	Export Model	•	Export a .stl file
	Examples	۲	►► Export a .stl file
	Open Print File	•	→ 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 model first.

2>Edit

Edit(E) Slice(P)	View(<u>V)</u> Model(<u>N</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(D)) Ctrl+D	←→Create a duplicate object of selected model
	Select All	Ctrl+A	←→Select all the models
	Deselect All	Ctrl+Shift+A	←→Deselect all the models
	Preferences		←→Set language, shortcuts and so on

27 / 93



Note:

There are some special options in *preferences.*

(1) General

General	Files	Shortcuts			
General					
Slice	Multi-Thr	eads Speedup:	3	•	
Langua	ge: (*)		English	•	
Maximu	m Duplica	te Copies:	5	*	
🗖 Sli	ce with u	nsaved templat	es		
🗖 Sav	e and res	tore window st	ate		
🗖 Sav	e and res	tore preview s	tate		
📝 Aut	omaticall	y check for ne	w version		
(*) Ta	ke effect	after restart			

Figure 4.2: The General tab of Preference.



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

assic emplate Name: High Quality - N General ill Density: 10 hells: 4 latform Addition: Raft onl upport: None Advanced Res Re lose without Saving anced Settings rr Infill Support Raft Cooling Ooz meral ayer Height: 0.40 mm seed efault Frinting Speed: 50.0 mm/ nner Shell Speed: 50.0 mm/ int and Drim kirt/Brim Extruder: 5 mm kirt/Brim Extruder: 5 mm rim Loop Lines: 1 mm rist Layer Speed: 15.0 mm/ mm			_? <mark>_</mark> 2
emplate Name: High Quality - N General ill Density: 10 hells: 4 latform Addition: Raft on upport: None Advanced Res Re lose without Saving anced Settings ar Infill Support Raft Cooling Ooz meral ayer Height: 0.01 = mm xtrusion Width: 0.40 = mm xtrusion Width: 0.40 = mm reed efault Printing Speed: 50.0 = mm fixt and Drim kirt/Brim Extruder: 1 = rim Loop Lines: 1 = rist Layer Speed: 0.30 = mm			
weneral ill Density: 10 hells: 4 latform Addition: Raft onl upport: Advanced Res Advanced Res lose without Saving ranced Settings rr Infill Support Raft Cooling Oor meral ayer Height: 0.01 mm/ marced settings retrusion Width: 0.00 mm/ firt and Brim kirt/Brim Extruder: firt Layer Seed: 10.0 mm irst Layer Seed: 0.30 mm irst Layer Speed: 15.0 mm	2 - PLA		
ill Density: 10 hells: 4 latform Addition: Raft onl upport: None Advanced Res Advanced Res Iose without Saving Res ranced Settings Res ranced Settings Res ranced Settings Res ranced Settings 0.01 ** ranced Settings 0.01 ** aver Height: 0.01 ** aver Height: 0.01 ** aver Height: 0.01 ** aver Height: 0.00 ** aver Height: 0.00 ** aver Height: 0.00 ** aver Shell Speed: 100.0 ** aver Shell Speed: 100.0 ** aver Shell Speed: 100.0 ** aver Johns ** kirt Loop Lines: 1 ** rim Loop Lines: 1 ** rist Layer Settings 15.0 ** irst Layer Speed: 15.0 **			
hells: 4 latform Addition: Raft onl upport: None Advanced Res Advanced Res re Infill Support Raft Cooling Ooz meral aver Height: 0.01 * mm xtrusion Width: 0.40 * mm reed efault Printing Speed: 50.0 * mm/ nner Shell Speed: 40.0 * mm/ /Y Axis Movement Speed: 100.0 * mm/ irt and Brim kirt/Brim Extruder: Primary Extruder kirt Loop Lines: 1 * rim Loop Lines: 5 * rst Layer Settings irst Layer Speed: 15.0 * mm/	\$		
latform Addition: Raft onl upport: None Advanced Res Advanced Res ranced Settings Ref er Infill Support Raft Cooling Ooz eneral 0.01 ÷ aver Height: 0.01 ÷ xtrusion Width: 0.40 ÷ mm 0.40 ÷ read 50.0 ÷ efault Printing Speed: 50.0 ÷ /Y Axis Movement Speed: 100.0 ÷ kirt/Brim Extruder: Primary Extruder kirt Loop Lines: 1 ÷ rim Loop Lines: 5 ÷ irst Layer Speed: 0.30 ÷ irst Layer Speed: 15.0 ÷	* *		
upport: None Advanced Res Ree Re Ree Re Iose without Saving Re anced Settings Re er Infill Support Raft Cooling Ooz anreal 0.01 * ayer Height: 0.01 * xtrusion Width: 0.40 * mm Red efault Frinting Speed: 50.0 * efault Frinting Speed: 100.0 * rinr Shell Speed: 100.0 * wirt/Brim Extruder: Primary Extruder kirt/Brim Extruder: 1 * rim Loop Lines: 1 * rist Layer Settings 0.30 * irst Layer Speed: 15.0 *	-y	•	
Advanced Res Res Res Iose without Saving Res ranced Settings Res er Infill Support Raft Cooling Ooz marcal 0.01 ** ayer Height: 0.01 ** xtrusion Width: 0.40 ** ored 50.0 ** efault Printing Speed: 50.0 ** nmr 100.0 ** virt and Brim ** kirt/Brim Extruder: Primary Extruder kirt Loop Lines: 1 ** rist Layer Settings 0.30 ** irst Layer Speed: 15.0 **	ŝ	•	
Anced Settings aranced Settings aranced Settings aranced Settings are Infill Support Raft Cooling Ooz aranal aver Height: 0.01 * mm xtrusion Width: 0.40 * mm yeed efault Printing Speed: 50.0 * mm/ rar Shell Speed: 40.0 * mm/ /Y Axis Movement Speed: 100.0 * mm/ irst and Brim kirt/Brim Extruder: Primary Extruder kirt Loop Lines: 1 * rim Loop Lines: 5 * rst Layer Settings irst Layer Speed: 15.0 * mm/	itore 🔻		
ranced Settings r r Infill Support Raft Cooling Oor eneral aver Height: 0.01 mm xtrusion Width: 0.40 mm xtrusion Width: 0.40 mm seed efault Printing Speed: 50.0 mm/ Axis Movement Speed: 100.0 mm/ (Y Axis Movement Speed: 100.0 mm/ tirt and Brim kirt/Brim Extruder: 1 r im Loop Lines: 1 r im Loop Lines: 5 r irst Layer Speed: 15.0 mm/	store to the L	ast Saved Save As	Save and Close
er Infill Support Raft Cooling Ooz eneral aver Height: 0.01 * mm xtrusion Width: 0.40 * mm peed efault Printing Speed: 50.0 * mm/ nmer Shell Speed: 40.0 * mm/ /Y Axis Movement Speed: 100.0 * mm/ tirt and Brim kirt/Brim Extruder: Primary Extruder kirt Loop Lines: 1 * rim Loop Lines: 5 * rist Layer Sttings irst Layer Speed: 0.30 * mm/ irst Layer Speed: 15.0 * mm/			3
ayer Height: 0.01 Å mm ayer Height: 0.40 Å mm xtrusion Width: 0.40 Å mm peed efault Printing Speed: 50.0 Å mm/ Inner Shell Speed: 40.0 Å mm/ /Y Axis Movement Speed: 100.0 Å mm/ xirt and Brim kirt/Brim Extruder: Primary Extruder kirt Loop Lines: 1 Å rim Loop Lines: 5 Å rist Layer Settings irst Layer Speed: 15.0 Å mm/	e Other GCode]	
ayer Height: ayeed efault Printing Speed: efault Printing Speed: 40.0 ☆ mm/ /Y Axis Movement Speed: 100.0 ☆ mm/ kirt/Brim Extruder: kirt Loop Lines: rim Loop Lines: rim Loop Lines: rim Layer Settings irst Layer Height: 0.30 ☆ mm/ 15.0 ☆ mm/			
xtrusion Width: 0.40 mm peed efault Printing Speed: 50.0 mm/ nner Shell Speed: 40.0 mm/ /Y Axis Movement Speed: 100.0 mm/ kirt/Brim Extruder: Primary Extruder kirt/Brim Extruder: 1 m kirt/Brim Extruder: 5 m rim Loop Lines: 5 m rst Layer Settings irst Layer Height: 0.30 mm/ irst Layer Speed: 15.0 mm/			
seed efault Printing Speed: 50.0 * mm, nner Shell Speed: 40.0 * mm, /Y Axis Movement Speed: 100.0 * mm, xirt and Drim kirt/Brim Extruder: Primary Extruder kirt Loop Lines: 1 * rim Loop Lines: 5 * rist Layer Settings irst Layer Height: 0.30 * mm, irst Layer Speed: 15.0 * mm,	(The printer's n	ozzle diameter is 0.4	mm)
efault Printing Speed: 50.0 x mm, nner Shell Speed: 40.0 x mm, /Y Axis Movement Speed: 100.0 x mm, kirt and Brim kirt/Brim Extruder: Primary Extruder kirt Loop Lines: 1 x rim Loop Lines: 5 x rist Layer Settings irst Layer Height: 0.30 x mm, irst Layer Speed: 15.0 x mm,			
nner Shell Speed: 40.0 * mm, /Y Axis Movement Speed: 100.0 * mm, kirt/Brim Extruder: Primary Extruder kirt Loop Lines: 1 * rim Loop Lines: 5 * rst Layer Settings irst Layer Height: 0.30 * mm, irst Layer Speed: 15.0 * mm,	s		
/Y Axis Movement Speed: tirt and Drim kirt/Brim Extruder: Primary Extruder kirt Loop Lines: Tim Loop Lines: Tim Loop Lines: Tim Layer Settings irst Layer Height: 0.30 mm irst Layer Speed: 15.0 mm	's Oute	r Shell Speed:	25.0 🚔 mm/s
kirt and Brim kirt/Brim Extruder: Primary Extruder kirt Loop Lines: 1 * rim Loop Lines: 5 * rst Layer Settings irst Layer Height: 0.30 * mm, irst Layer Speed: 15.0 * mm,	's ZAx	is Movement Speed:	25.0 🚔 mm/s
kirt/Brim Extruder: Primary Extruder kirt Loop Lines: 1 rim Loop Lines: 5 rst Layer Settings irst Layer Keight: 0.30 irst Layer Speed: 15.0 mm,			
kirt Loop Lines: rim Loop Lines: rst Layer Settings irst Layer Height: 0.30 $\frac{1}{\sqrt{v}}$ mm, irst Layer Speed: 15.0 $\frac{1}{\sqrt{v}}$ mm,	•		
rim Loop Lines: 5 × Irst Layer Settings irst Layer Height: 0.30 × mm irst Layer Speed: 15.0 × mm	Offs	et Distance:	3.0 🎽 mm
irst Layer Settings irst Layer Height: 0.30 👘 mm irst Layer Speed: 15.0 👘 mm,			
rst Layer Settings irst Layer Height: 0.30 (*) mm irst Layer Speed: 15.0 (*) mm,			
irst Layer Speed: 0.30 w mm	7.	29 2 9 1 2	
,, · · · · · · · · · · · · · · ·	firs	ι Layer Flowrate:	100.0 😴 %
Restore 🔻			OK Cancel
Restore Defaults			

Figure 4.3: Enabled "Slice with unsaved templates".



Save and restore window state refers towindow state can be saved and restored with enabling this function

Save and restore preview state refers to preview state can be saved and restored with enabling this function.

Automatically check for new version refers tonew version can be checked with enabling this function.

(2) Files

Commel	Filer	Chautanta		
General	TITES	Shortcuts		
STL Imp	ort			
🔲 Wel	d Vertice	5		
Weldin	g Thresho	Ld: 0.030 mm		
Import	options	are only applied to newly impor	ted models.	
-				
Models				
🔽 Pla	ce models	on platform after rotating, sc	aling, and mirroring	
is frace models on practoum arcer rocacing, scaling, and milloring				
[ma]				
📝 Aut	omaticall	y position import models		
🔽 Aut	omaticall	y position import models		
🔽 Aut	omaticall;	y position import models		
📝 Aut	omaticall	y position import models		
📝 Aut	omaticall;	y position import models		
<table-cell> Aut</table-cell>	omaticall	y position import models		
<table-cell> Aut</table-cell>	omaticall;	y position import models		
<table-cell> Aut</table-cell>	omaticall;	y position import models		
<table-cell> Aut</table-cell>	omaticall;	y position import models		
<table-cell> Aut</table-cell>	omaticall	y position import models		

Figure 4.4: The Files tab of Preference.

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



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

Rotation World Transform	X Local Transform	
X-axis: 0.0 Y-axis: 0.0	Deg 🖒 🖒	
Z-axis: 0.0	Deg 🖒 🕐	
Reset		

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



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



(3) Shortcuts



Figure 4.7: The Shortcuts tab of Preference.

3>Slice





4>view



Note:

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



5>Model

Mod	del(<u>M)</u> Repair(<u>R</u>) Printer(<u>C</u>)	Help(<u>H</u>)	
۲	View(V) Pan(<u>P</u>)	Shift+V Shift+P	 →Click left button and move mouse to rotate the view →Click left button and move mouse to pan the view
	Move(<u>M</u>) Rotate(<u>R</u>) Scale(<u>S</u>) Free Cut Support Structure Cross Section	Shift+M Shift+R Shift+S	 Click left button and move mouse to move model Click left button and move mouse to rotate model Click left button and move mouse to change size of model Cut the model in two parts Create your own support for the model Check the model's feature by layer with cross section
	Mirror Center Lay Flat Auto Fit to Build Volumn	•	 →Change model into a mirror shape →Move model to the center of printing area →Select a surface to lay the model flat on build platform →Scale model into its maximum size which is able to be printed
	Reset All Reset	•	\blacktriangleright Reset settings of the model
	Put All Models on Platform Auto Ungroup Merge Selected Models Align Selected Models		 Check that what is on the build platform Automatically ungroupthe 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

Note:

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



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

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

		λ		
<u>_</u>	Cross Section	×		1
-	🛩 Enable		\leftarrow	{
:=	O X: 48.68	mm 📃 Reverse		
-	O Y: 44.51	mm 📃 Reverse	4 y	1 al
⊥	• Z: 94.00	mm 🔲 Reverse	4	94.0 mm
	Control:		1// 1/10	
	🛩 Show Plane 📃 I	nches		3 33 1
	Reset		$//\lambda_{-1}$	- II

Figure 4.8: Cross Section in the Z axis direction.

:=		
1	Cross Section	
	🗹 Enable	
:=	○ X: 48.68 — nm	Reverse
	OY: 44.51	Reverse
企	O Z: 68.00 mm	Reverse
	Control:	58. U mm
	🗹 Show Plane 📃 Inche:	5
	Reset	

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



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



Figure 4.10: Cross Section without "Show Plane".

Inches will choose the unit between inches or mm.

_	Cross Section
-	- Enable
i≡	X: 1.92 inch Reverse
	OY: 1.75 inch Reverse
ſ	⊙ Z: 3.50 → inch Reverse
	Control:
	Show Plane Inches
	Reset

Figure 4.11: Cross Section with "Inches".


Note:

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



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



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

Face Index refers to the face you selected.



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



Figure 4.14: The model with "Lay Flat".



6> Repair Repair(R) Printer(C) Help(H) Remove Duplicate Faces Remove Isolated Faces Fix Face Orientation Flip Mesh Orientation Fix Holes Auto Repair... Automatically repair all the faults of the selected model(s)

Note:

You can also find auto repair icon in tool bar.

7>F	Printer		
Prin	ter(<u>C)</u> Help(<u>H</u>)		
	Current Type of Printer	۲	←→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 wire
	Printer Settings		←→Set your printer's parameters
	Filament Settings		←→Set the filament that your printer is using
	Configuration Wizard		←→ Wizard for set configuration

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.





4.1.2 Tool Bar

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



: Delete the selected model.



: When you enable this button, you can set model's color and which nozzle to print it

(if you have your printer as a dual-extruder printer).



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



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



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



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

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

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



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





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







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



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

You can also set the exact amplify rate value to change model in the operation property zone. *Inch* refers to changing size display from mm to inch.

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

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





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



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

model apart.



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



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

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



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

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



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



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



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



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





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

property zone.

Auto Supports:

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

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

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



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



Manual Supports:

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

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

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

Manual Supports								
🕀 Add	Θ	Remove	ø	Edit				
Pillar Z-Pos		0.00		≜ ▼ mm				
Pillar Heigh	ıt:	0.00		÷ mm				
Clear	Sup	ports						

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

Max Fit : When you click this button, the selected model will be amplified to max 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.



ir :Automatically repair model's defects.





: Start to slice the model.



: Abort slicing.



": Show the sliced file in layers.



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

4.1.3 Operation property



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

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



4.1.4 Model list



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

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



: Delete the model you have selected.



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

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





Figure 4.30: Please export model one by one.

4.1.5 Uploading queue



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

This section shows the uploading list to your printer.



4.1.6 Instant operation bar



: Add models



: Start slicing directly



: Save as another printing file



4.1.7 Model preview



Figure 4.32: Preview your model.

This section is designed for previewing the current model.



4.1.8 Perspective transformation



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

Check the model with the default views.

4.1.9 Detecting Information



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

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

Dimensions refers to the size of the model.

Triangles refers to the number of triangles in the model.

Edges refers to the number of edges in the model.

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

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



4.2 Slicing settings

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

Select Template		8 💌
Printer Type: RAISE3D N2 - V2 Ho	t End	▼ Settings
Filament Primary Extruder: PLA 1.75mm - V2 Ho	t End 🔻	
High Quality - N2 - PLA		Duplicate
Standard - N2 - PLA		Delete
		Import
		Export
Select Template: Speed - N2 - FLA		
Layer Height: 0.25 mm	Shells: 2	Cancel
Fill Jensity: IU.U %	intill Speed: 90.0 mm/s	Slice

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



Template Name: Spe	ed - N2 - PLA	
General		
Fill Density:	10 🔦 🗴	
Shells:	2 *	
Platform Addition:	Raft only 🔻	
Support:	None 🔻	
Advanced	Restore Defaults	

Figure 4.36: Common parameters.

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

Shells refer to the thickness of model's wall.

Platform Addition refers to the type of bottom layer.

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

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

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

Platform Addition:	Raft only 🔻
Support:	None Skirt only Brim only
	Raft only Raft with Skirt Raft with Brim

Figure 4.37: There are 3 types of Raft.





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



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

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

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

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

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





Figure 4.40: There are 3 types of supports.



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



Figure 4.42: Printed with supports of "All".



Note: Advance Setting.

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

Tennlete Nene:			
remprace Name. Spe	ed - NZ - FLA		
General			
Fill Density:	10 🔹 9	\$	
Shells	2		
Platform Addition:	Raft only	•	
Support:	None	•	
Advanced	Restore Defa	ults	

Figure 4.43: There are more settings in Advanced.

4.2.1 Layer

General:

Layer Height refers to the thickness of every single layer. Someone call it resolution as well. *Extrusion width* refers to width of extruded line. The default extrusion width will be the same as origin 0.4mm nozzle's diameter. If you have changed your nozzle to be other size in *Printer Settings*, please remember also to edit here.

Speed:

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

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

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





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

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

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

Skirt and Brim:

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

Force Addition of Skirt refers to enabling a skirt.

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

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

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



First Layer Settings:

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

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

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

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

ayer	Infill	Support	Raft	Cooling	Ooze	Other	GCode				
Gener	.al										
Laye	r Height:		0.25	mm							
Extr	usion Widtl	1 :	0.40	mm	(The	printer's	nozzle (liameter is O.	4 mm)		
Speed	i										
Defa	ult Printin	ng Speed:	50.0	😭 mm/ :	5						
Inne	r Shell Sp	eed:	40.0	🛬 mm/s	5	0	iter Shel	l Speed:	25.0	🔿 mm/s	
X/Y	Axis Movem	ent Speed:	150.0	🔹 mm/s	5	Z	Axis Mov	ement Speed:	25.0	🔹 mm/s	
Skirt	. and Brim										
Skir	t/Brim Ext	ruder:	Left Extr	ruder	•]					
Skir	t Loop Lin	25:	1	*		0	Efset Dis	tance:	3.0	mm	
Brim	Loop Line	6	5	*							
First	. Layer Set	tings									
Firs	t Layer He	ight:	0.30			F	irst Laye	r Flowrate:	100.0	* *	
Firs	t Layer Sp	eed:	30.0		S						

Figure 4.45: The Layer tab.



4.2.2 Infill

Infill:

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

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

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

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

Infill Pattern Type refers to infill structure.

Grid refers to the mesh structure.

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

Infill Pattern Type:



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

Top and Down Solid Part:

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

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

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

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

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

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

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

Lines refer to the segments at the same direction.

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



Solid Fill	Pattern	Туре:	L
			T

Lines	-
Lines	
Rectilinear	

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

ayer Infill Support	Raft Cooling Ooze	Other GCode		
Infill		Top and Down Solid Part		
Infill Extruder:	Primary Extruder 🔻	Bottom Solid Fill Layers:	50	
Infill Speed:	30.0 🚔 mm/s	Top Solid Fill Layers:	50	
Infill Overlap:	25 💽 %	Bottom Solid Fill Speed:	60.0	mm/s
Infill Flowrate:	100.0 🔭 %	Top Solid Fill Speed:	60.0	mm/s
Infill Pattern Type:	Grid	Bottom Solid Fill Flowrate:	100.0	\$
		Top Solid Fill Flowrate:	100.0	* *
		Solid Fill Pattern Type:	Lines	•

Figure 4.48: The Infill tab.

4.2.3 Support

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

Support:

Support Type refers to the structure of support.

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

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



Support Type: Nor Support Infill Type: Pil

Normal -Normal Pillar

Figure 4.49: Two types support structure.

Support Infill Type refers to the infill structure of support.

Lines refer to the segments at the same direction.

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

Support Infill Type:	Line 💌
Surgeout Served:	Grid
Support Speed:	······································

Figure 4.50: Two types support infill.

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

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

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

Overhang Angle refers to the angle between the overhang surface and Z axis.

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

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

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

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

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

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

Dense Support:

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



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

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

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

Dense	Support	Infill	Type:	Line
				Grid
				Line

Figure 4.51: Two types of dense support infill.

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

Angle refers to the direction of support.

Add Infill Angle refers to adding a value.

Remove Infill Angle refers to removing the selected value.



sayer infill Support K	art cooring	Oore	other ocode	
Support Extruder: Left Extru	der 🔹	·		
Support			Dense Support	
Support Type:	Normal	•	Dense Support Layers:	0
Support Infill Type:	Line	•	Dense Support Infill Ratio:	80 🐥 %
Support Speed:	50.0	mm/s	Dense Support Infill Type:	Line 🔻
Infill Ratio:	30	8	Interlaced Support Infill Ang	le
Max Overhang Angle:	60.0	Deg	Angle: 0.0 🚔 Deg	0
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	8		
Pillar Size:	4.00	mm		-
🔲 Sparse Connection				

Figure 4.52: The Support tab.

4.2.4 Raft

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

Raft:

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

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

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

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

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

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



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

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

Raft Lines Type refers to the fill structure of Raft.

Lines refer to the segments at the same direction.

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

Raft Lines Type:	Lines 💌
	Lines
	Rectilinear

Figure 4.53: Two types of Raft.

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

Layer Infill Su	upport Raf	t Cool	ing	Ooze	Other	GCode		
Raft Extruder:	Left Extrude	r	•					
Raft								
Raft Offset:	I	5.00	*	mm				
Raft Gap from Mode	1:	0.20		mm				
First Layers:		2	*					
Surface Layers:		2	*					
First Layer Speed:		8.0	*	mm/s				
Interface Layer Sp	eed:	30.0	*	mm/s				
Surface Layer Spee	d:	60.0	*	mm/s				
Surface Layer Infi	ll Ratio:	85	*	*				
Raft Lines Type:	(Lines		•				
🔲 Keep Holes in R	aft Structur	•						

Figure 4.54: The Raft tab.



4.2.5 Cooling

Cooling:

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

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

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

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

Temperature:

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

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

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

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

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

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

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

Fan Speed refers speed of the extra cooling fan.

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



Temperature:

Bed Temperature: refers temperature of build plate.

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

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

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

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

Layer Infi	11 Support	Raft	Cooling	g Ooze	Other	GCode			
Cooling						Temperature			
Minimal Lay	er Print Time:	1	5.0	sec		Bed Temperature:	40	🔹 ° C	
Default Far	Speed:	1	00 🍦	%		Left Extruder:	215	÷ C	
Maximum Far	. Speed:	1	00 🌻	*		Right Extruder:	215	e v c	
Minimal Pri	nting Speed:	1). 0 🍂	mm/s					
Fan Control									
Layer:	1		Layer	Fan Speed					
Fan Speed:	0	×	1	0 100					
Add Roy	Point								
Addition	, i i orac								
Remove F	an Point								

Figure 4.55: The Cooling tab.



4.2.6 Ooze

Retract:

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

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

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

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

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

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

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

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

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

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

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

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

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



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

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

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

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

WipeWall Offset refers to the distance between WipeWall and model.

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



Figure 4.56: Set Angle of WipeWall to be 90.





Figure 4.57:Set Angle of WipeWall to be 0.

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

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

WipeWall Type:



Figure 4.58: Three type of WipeWall.




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



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





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

Layer	Infill	Support	Raft	Cooling	Ooze	Oth	er GCode			
Retra	ct									
📝 E:	nable Retr	action								
Retr	act Speed:		ĺ	20.0		s		Restart Speed:	20.0	mm/s
Retr	act Materi	al Amount:		1.0	mm			Z hop:	0.000	mm
Mini	nal Travel	of Retract:	ion:	0.5	mm			📝 Force Retract O	In Layer Change	
Mini	nal Amount	of Retract:	ion:	0.02	🔹 mm			🔲 Avoid Traveling	; Through Holes	
Extr	a Restart	Amount:	1	0.00	mm			📝 Avoid Retract I	nside Models	
📝 E: Retr	nable Wipe act Speed	Wall of Extruder	-switch:		20.0	(A) (V)	mm/s	WipeWall Offset:	2.00	mm
Retr	act Amount	of Extrude	r-switch	:	3.50	*	mm	WipeWall Angle:	30.0	Deg
Rest	art Speed	of Extruder	-switch:		20.0	*	mm/s	WipeWall Type:	Contoured	•]
Extr	a Restart	Amount of E:	xtruder-	switch:	0.00	*	mm			

Figure 4.62: The Ooze tab.



4.2.7 Other

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

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

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

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

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

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

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

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

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

Random refers to separate position.

Layer Start Point Type: Nearest Fixed Layer Start Point X: Fixed Random

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

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

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

75 / 93



You can define the height by clicking the *Add Pause Point* button. And delete the defined height by choosing the height in right box and clicking the *Remove Pause Point* button. *Note*: This height doesn't include *Raft*.

Global Offset means that you can set offset to the location of your model. You can separately set **X-Offset**, **Y-Offset** and **Z-Offset**.

layer Infil	l Support	Raft	Cooling	Ooze	Other	GCode	<u>a</u>				
Spiral Ve Merge Ope Merge Int Print Ext	use Mode en Segments of ernal Overlapp ernal Shells F	Model Pa ing Part irst	rts s		- Pause Heig	at Heig ht: Add Pau Remove P	5.000 15e Point ause Point	mm			
📝 Check Thi Thin Wall Wi	n Wall (Single dth: 0.5	Extrusi	on Width)] mm								
Layer Start Fixed Layer Fixed Layer	Point Type: Start Point X: Start Point Y:	Neares 0.(0.(t · ·	mm	The	height j	property do	es not i	nclude raft	20	
-Global Offse	Č-										
X-Offset:	0.00	mm	Y-Offset	: 0.	00 🌻	mm	Z-Offse	t: 0	00	mm	

Figure 4.64: The Other tab



4.2.8 GCode

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

ayer Infill S.	mport Raft Cooling Ooze Other GCode	
Start GCode		
Number of Extruder	(2)	
	621 G90	
1	M82	
S3	M107	
	G28 X0 YO	100
	G28 Z0	=
	G1 Z15.U F{travel_speed}	
	G1 F140 F29	
	G1 X20 Y0 F140 E30	
	G92 E0	
	G1 F{travel_speed}	-
1 -	M140 S0 G91 G1 E-1 F300 G1 Z+0.5 E-5 X-20 Y-20 F9000.00 G28 X0 Y0 M84 G90	

Figure 4.65: The GCode tab.



4.3 Multiple Extruders

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

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

Printer->Printer Settings->Multiple Extruders



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

igs			
AISE3D N2 - V2	Hot End		▼ Add Printer
ater:	0, 40	mm	Extrusion Width NOI be overwritten
	305.00	mm	
	305.00	mm	
t:	305.00	mm 🔹	
AM :	0.00	mm	
ensation:	100.00	* %	
ensation:	100.25	* *	
ruder: PL	A 1.75mm - V2 H	ot End	• Multiple Extruders
n			
	230400	-	
			OK Concel
	ngs MAISE3D N2 - V2 eter: : : t: MM: ensation: ensation: ruder: PLi	ngs HAISE3D N2 - V2 Hot End eter: 0.40 : 0.40 : 305.00 : 305.00 t: 305.00 MM: 0.00 ensation: 100.00 ensation: 100.25 ruder: PLA 1.75mm - V2 H n 230400	ngs HAISE3D N2 - V2 Hot End eter: 0.40 * mm : 0.40 * mm 305.00 * mm 100.00 * mm MM: 0.00 * mm MM: 0.00 * mm ensation: 100.00 * % ensation: 100.25 * %

Figure 4.67: Click "Multiple Extruders".



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

Extruder Count:	2 •		
Left Extruder:	2 ABS 1. (5mm - V2 Hot E: o	•	
Right Extruder:	PLA 1.75mm - V2 Hot End	1 🔹	
Extruder Offset X:	25.00 🗼 mm		
Extruder Offset Y:	0.00 🔹 nm		
Temperature Limit			
Heat Bed Maximum Tem	perature Limit:	110	🔹 ° C
Left Extruder Maximu	m Temperature Limit:	310	* C
Right Extruder Maxim	um Temperature Limit:	310	▲ C

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

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



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



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



Add	Delete View	W Pan	₩ove	C Rotate	Scale	n Free Cut	Support
Å							
4							
Je .	Position		×				-
	X: 14.82 Y: 0.00	mm mm		F		HP	325
.↑.	Z: 8.02	mm		/v_			2
	Inches Make Center	On Platfor	n /		V.	-	
1	Rese	6					
-/	Align To;	ether					



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



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



Figure 4.71: Check Right Extruder's printing range.



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

Layer Infill S	upport	Raft	Cooling	Ooze	Other	GCode
Support Extruder:	Left Ext	ruder	+			
Support	Right Ex	ruder truder			Dens	e Support
Figure 4.72	: Choos	se an e	xtruder t	to print	t suppo	rt.
-				-		
I	C	Raft	Culture	0	011	CC.1.
Layer Infill	Support	nare	Looling	Uoze	Uther	GLODE
Raft Extruder:	Left Ex	truder	•	•		
Raft	Left Ex Right B	truder xtruder		<u> </u>		

Figure 4.73: Choose an extruder to print raft.

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

Raft Cooling	Ooze	Other	GCode	
			- Temperature	
15.0 🚔 se	ec.		Bed Temperature: 40 🚔 ° C	
100 💉 %			Left Extruder: 215 🚔 °C	
100 🔹 %			Right Extruder: 215 🚔 °C	

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



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

These settings as follows:

Multiple Extruders Ooze Control:

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

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

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

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

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

WipeWall Offset refers to the distance between WipeWall and model.

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





Figure 4.75: Set Angle of WipeWall to be 90.



Figure 4.76:Set Angle of WipeWall to be 0.



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

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

WipeWall Type:



Figure 4.77: Three type of WipeWall.



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





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



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



Layer	Infill	Support	Raft	Cooling	Ooze	Other	GCode			
Retra	ict									
V E	nable Retr	action								
Retr	act Speed:		[20.0	mm/s			Restart Speed:	20.0	mm/s
Retr	act Materi	al Amount:	Ī	1.0	mm			Z hop:	0.000	mm
Mini	mal Travel	of Retracti	ion:	0.5	mm			📝 Force Retract O:	n Layer Change	
Mini	mal Amount	of Retracti	ion:	0.02	🔹 mm			📄 Avoid Traveling	Through Holes	
Extr	a Restart	Amount:	- I	0.00	mm			📝 Avoid Retract I:	nside Models	
📝 E Retr	nable Wipe act Speed	∦all of Extruder	-switch:		20.0	mm	/s	WipeWall Offset:	2.00	mm
Retr	act Amount	of Extruder	r-switch		3. 50	mm		WipeWall Angle:	30.0	Deg
Rest	art Speed	of Extruder-	-switch:		20.0	mm	/ s	WipeWall Type:	Contoured	•
Extr	a Restart .	Amount of Ex	ctruder-	switch:	0.00	mm				
-										

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

4.4 Filament Settings

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

1. Edit the existing template.

Printer->Filament Settings

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

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



Туре:	ABS 1.75mm - V2 Hot En	d 🔻 🖌 🖌
Diameter:	1.750 🚔 mm	
Density:	1120.00 🚔 kg/r	n3
Price:	30.00 🌲 \$/k;	g .
Flowrate:	100.00 💉 %	

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

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

2. Add a new template.

Type:	new filament	
Diameter:	2.930	mm nm
Density:	1300.00	🔹 kg/m3
Price:	30.00	* \$/kg
Flowrate:	100.00	* %

Figure 4.84: Set parameters for the new filament.



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

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

4.5 Printer settings

When you want edit the parameters for your printer, follow check the steps as below: 1. Edit the existing template in **Printer->Printer Settings**



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



Printer			-	
Type: RAISE3D N2		▼ Ad	d Printer	
Nozzle Diameter:	0.40	🔹 mm		
Build Width:	305.00	mm		
Build Depth:	305.00	mm		
Build Height:	305.00	mm		
Step-E per MM:	0.00	🔹 mm		
X-axis Compensation:	100.00	÷ %		
Y-axis Compensation:	100.25	\$		
🔽 Heat Bed				
Filament				
Primary Extruders: PLA	. 1. 75mm		• Multip	le Extruders
Communication				
Baud Rate: 2304	100			

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

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



2. Add a new printer.

1 Add Printer					? X
Printer			Extruder		
Type: new printer			Extruder Count:		•
Nozzle Diameter:	0.40	m	Extruder Offset X:	0.00	m
Build Width:	305.00	mm	Extruder Offset Y:	25.00	mm
Build Depth:	305.00	mm	Primary Extruder:	PLA 1.75mm	•
Build Height:	305.00	mm			
Step-E per MM:	0.00	mm			
X-axis Compensation:	100.00	*			
Y-axis Compensation:	100.00	x			
🕅 Heat Bed					
			_	OK C	Concol
				10	Cancel

Figure 4.87: Set parameters for the new printer.





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