ideaMaker Manual





Using ideaMak	er	2				
Basic infor	Basic information2					
What is	What is ideaMaker?					
Where	e to download ideaMaker?	2				
Install idea	aMaker	3				
Let's Print!	I	6				
How to use	e ideaMaker?	23				
Interfa	эсе	23				
1.	Menu Bar	23				
2.	Tool Bar	26				
3.	Operation property	30				
4.	Model list	30				
5.	Uploading queue	31				
6.	Instant operation bar	31				
7.	Model preview	31				
8.	Perspective transformation	32				
9.	Detecting Information	32				
Advano	ced slicing settings	32				
Ad	Ivance settings	35				
	Layer	35				
	Infill	37				
	Support and Raft	38				
	Cooling	40				
	Ooze	41				
Other						
	GCode					
Mu	Multiple Extruders					
Fila	Filament Setting 49					



Using ideaMaker

Basic information

What is ideaMaker?

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

Where to download ideaMaker?

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



Install ideaMaker

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



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



0 ideaMaker 2.1.2.3160	Setup				
Û	Choose Install Location Choose the folder in which to inst	tall ideaMaker 2.1.2.3160.			
Setup will install ideaMaker 2.1.2.3160 in the following folder.To install a different folder, click Browse and select another folder. Click Next to continue.					
Destination Folder					
D:\Program Files\Raise	23D\jdeaMaker\	Browse			
Space required: 139.9MB Space available: 30.5GB					
Raise3D www.raise3d.com - i	deaMaker 2.1.2.3160	Next > Cancel			

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.

0 ideaMaker 2.1.2.3160 Setup						
(j)	Choose Components Choose which features of ideaMaker 2.1.2.3160 you want to install.					
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 instal	I: IdeaMaker 2.1.2.3160 File Association Microsoft Visual C++ 2008 SP1 Redistributable Printer Driver					
Space required: 139.9MB						
Raise3D www.raise3d.com - id	eaMaker 2.1,2.3160					

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



1 ideaMaker 2.1.2.3160) Setup	
(j)	Installation Complete Setup was completed successfully.	
Completed		
Show <u>d</u> etails		
Raise3D www.raise3d.com -	ideaMaker 2,1,2,3160	ext > Cancel

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

() ideaMaker 2.1.2.3160 Set	tup
	Completing the ideaMaker 2.1.2.3160 Setup
(î)	ideaMaker 2.1.2.3160 has been installed on your computer. Click Finish to close Setup.
ideaMaker	✓ Run ideaMaker 2.1.2.3160
	Show ideaMaker Website
	www.raise3d.com
	< Back Finish Cancel



Let's Print!

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

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



2. Click the button " i " to start slicing the model.



🚺 ideaMa	aker - *														
File I	Edit Slice	View	Model	Repair	Printer	Help									
Add .	D elete	O View	Pan Pan	₩ove	C.	Scele	n Free Cut	Mux Fit	Duplicate	D Reset	Repair) Start		H Preview	Connect
Time: Filam	11 hours, ent: 132.0	18 min	, 22 sec			_	Ŧ	i	H						< Â>
	Model Info			×		$\langle \rangle$				\wedge					
	Extruder: Color:	Primary	Extruder		77										
٢					$/ \bigwedge$										
						V				+ V					
						V				y \			Model: Triang Edges: Non-Ma	rabbit.s ;les: 3824 834 unifold Ec	stl 490 dges: O
	/												Model: Triang Edges: Non-Ma Error	rabbit.s ;les: 382 834 unifold Ed Orientat:	stl 490 dges: O ion Faces: O

3. Select your printer type and filament type.

Printer Type:	RAISE3D N2	2		•			
Filament	ideaPrinte ideaPrinte	er F100 er F100L					
Primary Extrude	ideaFrinte RAISE3D N1	idearinter Fl ideaPrinter F2 RAISE3D N1					
	RAISE3D N2 RAISE3D N2 RAISE3D N2	2 Plus					
	KALSESD NI	L Lite					
		Filament					
		Primary Extruder:	PLA 1.75mm 👻				
			PLA 3mm ABS 3mm	-			
		Sec. 12	ABS 1.75mm				

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



Select Template		? 💌
Printer Type: RAISE3D N2 Plus		-
Filament Primary Extruder: PLA 1.75mm	▼	
High Quality - N2 Plus - PLA		Duplicate
Standard - N2 Plus - PLA		Edit
Speed - N2 Plus - PLA		Delete
Select Template: Standard - N2 Plus -	PLA	
Layer Height: 0.15 mm	Shells: 2	Cancel
Fill Density: 10.0 %	Infill Speed: 70.0 mm/s	Slice

 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.)



🚺 Edit Template			? 💌
General			
Template Name:	Standard - N2 Plus - PLA		
Fill Density:			10 %
Shells:			2
Raft: Raft	•	Support: None	•
Advance	Restore Defaults		
Don't Save and	Close	Save As Save a	and Close

6. Click "*Slice"* button.



Select Template		? 🗙
Printer Type: RAISE3D N2 Plus		•
Filament Primary Extruder: PLA 1.75mm	•	
High Quality - N2 Plus - PLA		Duplicate
Standard - N2 Plus - PLA		Edit
Speed - N2 Plus - PLA		Delete
Select Template: Standard - N2 Plus - Pl	A	
Layer Height: 0.15 mm	Shells: 2	Cancel
Fill Density: 10.0 %	Infill Speed: 70.0 mm/s	Slice

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

Estimated Print Result	? 🔀			
Prepared Print File:	rabbit.gcode			
Estimated Print Time:	19 hours, 11 min, 27 sec			
Estimated Amount:	219.8 g / 70.29 m			
Estimated Price:	\$ 6.59			
Note: Result above is only for reference.				
Preview	Upload Export			



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



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

The yellow lines refer to the solid print part. The blue lines refer to the moving path of nozzle. The red marks refer to the retraction points.





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



9. Close the preview dialogue box after confirm.



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

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

Estimated Print Result	? 💌			
Prepared Print File:	rabbit.gcode			
Estimated Print Time:	19 hours, 11 min, 27 sec			
Estimated Amount:	219.8 g / 70.29 m			
Estimated Price:	\$ 6.59			
Note: Result above is only for reference.				
Preview	Upload Export			

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



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





Option2: Upload the sliced file by WIFI

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





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





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





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

🕕 Estimated Print Result	? 🔀
Prepared Print File:	rabbit.gcode
Estimated Print Time:	19 hours, 11 min, 27 sec
Estimated Amount:	219.8 g / 70.29 m
Estimated Price:	\$ 6.59
Note: Result above is o	only for reference.
Preview	Upload Export



🕽 Select Prin	nter			? 💌				
Print File	2							
rabbit.gcode								
Printer:	Raise3D N1	- Raise3D11@192.168.1.120	•	Refresh				
Туре:	Raise3D N1	- Raise3D11@192.168.1.120						
Name:	Raise3D N2Pl	us - plustest@192.168.1.194						
Address:	Raise3D N2	- Raise3Du@192.168.1.179						
	Raise3D N2	- Rn2@192.168.1.109						
	Raise3D N1	- R1@192.168.1.127		Cancel				
	Raise3D N2	- N2No2@192.168.1.166						

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



- 6. When finish uploading, you can start a print.
- 7. You can choose the print from the touchscreen on the printer. The uploaded files are in *Local storage*.





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





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

Printer Connect		8 ×					
	Remote Connection 9						
Raise3D N2Plus	plustest 192.168.1.195	(\Rightarrow)					
Raise3D N2	Raise3D 192.168.1.179	()					
	1 / 1						

9. The connecting page.





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



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





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



How to use ideaMaker?

Interface

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



1. Menu Bar

Menu bar includes all the operation commands and advanced setting.

1> File



File	(F) Edit(E) Slice(P) View(V)		
	New(<u>N</u>) Open(<u>O</u>) Close(<u>W</u>)	Ctrl+N Ctrl+O Ctrl+W		Create a new empty workspace Open a .idea file (which is used for old version of F series printer) Close current file
	Save(<u>S</u>) Save As	Ctrl+S		Save file Save project file as .idea
	Import Model Export Model	•		Import a .stl file 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		
File(<u>F</u>)	Edit(E) Slice(P) Vie	w(<u>V)</u> Mode	
1	Undo(<u>Z</u>)	Ctrl+Z 🔶	Revoke the last edit
■ F Add	Redo(<u>Y</u>)	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
L	Duplicate(<u>D</u>)	Ctrl+D 🔶	Create a duplicate object of selected model
	Select All(<u>A</u>)	Ctrl+A 🔶	Select all the models
•	Preferences	•••	Set language

3> Slice



4> view





Note:

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

5>	Model
5/	Piouei

/(V)	Mo	del(M) Repair(R) Pri	inter(C) Help	
(۲	View(V) Pan(P)	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(M) Rotate(R) Scale(S) Free Cut	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
		Mirror Center Lay Flat Auto Fit to Build Volun	nn •	Change model into a mirror shape Move model to the center of printing area Lay the model flat in printing platform Scale model into its maximum size which is able to be printed
		Reset All Reset	,] -	 Reset settings of the model
		Check on Platform Auto Ungroup Cross Section		Check that what is on the print platform Automatically ungroup the print Enable it to see details inside the model

Note:

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



Note:

You can also find auto repair icon in tool bar.



7> Printer

Repair(<u>R</u>)	Printer(C) Help(H)		
,t→	Current Printer	••••	Select the type of your printer
↓ Move	Raise3D N-series	••••	Set a remote connection with your printer(s)
\square	ideaPrinter F-series	•••	Set a connection with your printer(s) via USB wire
	Printer Settings Filament Settings	\leftrightarrow	Set your printer's parameters 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.



2. Tool Bar

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









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

it (if you have our dual-extruder printer).

File(F)	Edit(E) Slice	(P) View(V) Mode	l(M) Rep	air(R) Pri	nter(C)	Help(H)				
Add.	M	O	())	↔	C		Frag Cut	May Eft	Duplicato	D	∦ + Papair
Adu	Delete	view	Fall	Move	Rotate	Scale	Hee Cut			Reset	Kepair
2	Model Inf	ō		×							
<i>.</i>	Extruder:	Left Extr	uder								
≔	Color:										
*											
Ľ											

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



: When you enable this button, you can move ideaMaker's model window by

left-clicking your mouse.



: When you enable this button, you can move model to another place by

left-clicking your mouse. You can also set the exact X/Y coordinate value to move the model in the operation property zone.





: When you enable this button, you can rotate model to another angle by

left-clicking your mouse. You can also set the exact angle value to rotate model in the operation property zone.





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

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



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

model apart.





te: 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.



: 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 WIFI.



3. Operation property



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

4. Model list



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



5. Uploading queue



This section shows the uploading list to your printer.

6. Instant operation bar



: Start slicing directly



: Save as another printing file

7. Model preview

The section is designed for previewing the current model.



8. Perspective transformation

Set ideaMaker to the default view of the object.

9. Detecting Information

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

Advanced slicing settings

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

 Select Template 		? 🔀
Printer Type: RAISE3D N2 Plus		•
Filament Primary Extruder: PLA 1.75mm	▼	
High Quality - N2 Plus - PL	A	Duplicate
Standard - N2 Plus - PLA		
Speed - N2 Plus - PLA		Delete
Select Template: Standard - N2 Plu	s - PLA	
Layer Height: 0.15 mm	Shells: 2	Cancel
Fill Density: 10.0 %	Infill Speed: 70.0 mm/s	Slice



🕕 Edit Template	? 💌
General	
Template Name: Standard - N2 Plus - PLA	
Fill Density:	10 %
Shells:	2
Raft: Raft	•
Advance Restore Defaults	
Don't Save and Close Save As Sav	ve and Close

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

Shells refer to the thickness of model's wall.

Raft refers to the type of bottom layer.

The *Raft* layer will print couple thick layers as model's ground. The *Brim* layer will only print a single layer of shell.

brin layer will only print a single layer of shell.

Raft:	Raft 🔹
	None
	Raft
	Brim

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

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

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

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



With 2 shells, 10% fill density and Brim





With 5 shells, 20% fill density and Raft





Advance settings

1 Edit Template	? <mark>- × -</mark>
General	
Template Name: Standard - N2 Plus - PLA	
Fill Density:	- 10 %
Shells:	- 2
Raft: Raft	-
Advance Restore Defaults	
Don't Save and Close Save As Save	and Close

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

Layer

General:

Layer Height refers to the height of every single layer.

Speed:

Default Printing Speed refers to the speed of printing non-specified area.
 Inner Shell Speed refers to the speed of printing the model's inner shell.
 Outer Shell Speed refers to the speed of printing the model's outer shell.
 X/Y Axis Movement Speed refers to the speed that the nozzle moving at to another place without printing in X and Y directions.

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





Skirt:

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

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

First Layer Settings:

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

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

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

Reset:

Reset Defaults refers to going back to the factory defaults.



general	and Raft Cooling Ooze	Other GCode		
Layer Height:		o	.15 mm	
Speed				
Default Printing Speed:	50.0 🚔 mm/s			
Inner Shell Speed:	40.0 🚔 mm/s	Outer Shell Speed:	25.0 🚔 mm/s	
X/Y Axis Movement Speed:	150.0 🚔 mm/s	Z Axis Movement Speed:	30.0 🚔 mm/s	
Skirt				
Loop Lines:	1	Offset Distance:	3.0 🔺 mm	
First Layer Settings				
First Layer Height: 📁			0.30 mm	
First Layer Speed:	30.0 🚔 mm/s			
First Layer Flowrate:	100.0 💌 %			

Infill

Infill:

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

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

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

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

Top and Down Solid Part:

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

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

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

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

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

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



Layer Infill Suppor	t and Raft Cooling Ooze	Other GCode		
Infill		Top and Down Solid Part		
Infill Speed:	90.0 🖍 mm/s	Bottom Solid Fill Layers:	5 🌲	
Infill Overlap:	15 🔹 %	Top Solid Fill Layers:	5 🌲	
Infill Flowrate:	100.0 🔹 %	Bottom Solid Fill Speed:	60. 0 🍝 r	nm/s
Infill Pattern Type:	Grid 💌	Top Solid Fill Speed:	60. 0 🚔 r	nm/s
		Bottom Solid Fill Flowrate:	100.0	8
		Top Solid Fill Flowrate:	100.0 🚔 s	x

Support and Raft

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

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

Support:

Support Speed refers to the speed of printing the model's support structure. *Infill Ratio* refers to the density of infill structure of the support material.

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

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



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

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

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

Sparse Connection refers to enabling the loose connection with the print and the support. **Support Flowrate** refers to the amount of material extruded which will be multiplied by

this value while printing support.

Raft:

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

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

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

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

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

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

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

ayer Infill Suppo.	rt and Raft	Cooling	Ooze	Other	GCode				
Support Extruder: Lef	t Extruder	•		Raft Ext	ruder:	Left Ex	truder	-	
Support				-Raft -					
Support Speed:	50.0	🚔 mm/s		Raft O	ffset:		5.00	mm	
Infill Ratio:	30	*		Raft G	ap from	Model:	0.20	mm	
Max Overhang Angle:	60.0	🌲 Deg		First	Layers:		2	* *	
Horizontal Offset:	0.70	🚔 mm		Surfac	e Layers	:	2	×	
Vertical Offset:	0.15	mm		First	Layer Sp	eed:	8.0	🚔 mm/s	
Support Infill Type:	Line	•		Interf	ace Laye	r Speed:	30.0	🚔 mm/s	
🔲 Sparse Connection				Surfac	e Layer	Speed:	60.0	🚔 mm/s	
Support Flowrate:	100.0	*							



Cooling

Cooling:

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

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

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

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

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

Temperature:

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

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

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

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

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



Advance Settings	·			? <mark>×</mark>
Layer Infill Support and Raft	Cooling Ooze	Other GCode		
Cooling		- Temperature		
Minimal Layer Print Time: 1.0	sec	Bed Temperature:	40 🔹 °C	
Default Fan Speed: 100	* *	Left Extruder:	215 🔹 ° C	
Maximum Fan Speed: 100	* %	Right Extruder:	215 🔹 ° C	
Turn Fan On At Layer: 2	× v			
Minimal Printing Speed: 10.	D 🚔 mm/s			
Restore Defaults			ОК	Cancel

Ooze

Retract:

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

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

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

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

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

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

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



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

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

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

Multiple Extruders Ooze Control:

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

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

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

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

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

1 Advance Settings				?	×
Laver Infill Support and Raft Coolin	ng Ooze	Other	GCode		
Retract					
Retract Speed: 20.0	mm/s		Restart Speed:	20.0 👘 mm/s	
Retract Material Amount: \$1.0	mm		Z hop:	0.000 🍦 mm	
Minimal Travel of Retraction: 0.5	mm		📝 Force Retract	On Layer Change	
Minimal Amount of Retraction: 0.02	🚔 mm		🔲 Avoid Traveli	ng Through Holes	
Extra Restart Amount: 0.00	mm				
Multiple Extruders Ooze Control					
🔽 Enable WipeWall					
Retract Speed of Extruder-switch:	20.0	🚔 mm/s			
Retract Amount of Extruder-switch:	3, 50	i mm			
Restart Speed of Extruder-switch:	20.0	🚔 mm/s			
Extra Restart Amount of Extruder-switch:	0.00	mm			
Restore Defaults				OK Cancel	



Other

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

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

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

Advance Settings	? <mark>- × -</mark>
Layer Infill Support and Raft Cooling Ooze Other GCode	
Spiral Vase Mode Merge Open Segments of Model Parts	
Check Thin Wall (Single Extrusion Width) Thin Wall Width: 0.500 — mm	
Restore Defaults OK	Cancel

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

GCode

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



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

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

Advance Settings	? 💌
Layer Infill Support and Raft Cooling Ooze Other GCode	
Spiral Vase Mode	
✓ Check Thin Wall (Single Extrusion Width) Thin Wall Width: 0.500 ★ mm	
Restore Defaults OK	Cancel

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





Printer Settings			? 💌
Printer			
Type: RAISE3D N2			 Add 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:	100.00	* %	
📝 Heat Bed			
Filament			
Primary Extruders: PLA	1.75mm		 Multiple Extruders
Communication			
Baud Rate: 230400	<u>.</u>		
			OK Cancel

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



1 Multiple Extruders	? 💌
Extruders	Filament
Extruder Count:	Left Extruder: PLA 1.75mm 🔻
Extruder Offset X: 25.00	Right Extruder: 🛛 🔽 PLA 1.75mm 🔻
Extruder Offset Y: 0.00	
	OK Cancel

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

File(F)	Edit(E) Slice(P) View(V)	Model(M)	Repair(R) P	Printer(C)	Help(H)				
Add	S Delete	O View	Pan Mov	e Rotate	أ∭ Scale	Free Cut) Max Fit	Duplicate	D Reset	K epair
							2	E		
4	Model Info)	×							
-	Extruder:	Left Extrud	ler 🔻							
Ξ	Color:					Г			λ	
Ť						\ ,	Rall			
/							S No	1	10	
2						9	1010		1	
7								1		
+										
*										





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

Advance Settings			?
Layer Infill Suppor	rt and Raft Cooling (Doze Other GCode	
Support Extruder:	t Extruder -	Raft Extruder: Left Ext	ruder 🔻
Support Speed:	50.0 🚔 mm/s	Raft Offset:	5.00 🚔 mm
Infill Ratio:	30 🔹 %	Raft Gap from Model:	0.20
Max Overhang Angle:	60.0 📮 Deg	First Layers:	2
Horizontal Offset:	0.70 🚔 mm	Surface Layers:	2
Vertical Offset:	0.15 🚔 mm	First Layer Speed:	8.0 🚔 mm/s
Support Infill Type:	Line 🔻	Interface Layer Speed:	30.0 🚔 mm/s
🔲 Sparse Connection		Surface Layer Speed:	60.0 🚔 mm/s
Support Flowrate:	100.0 🚔 %		
Restore Defaults			OK Cancel

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



Advance Settings	? 💌
Layer Infill Support and Raft Cooling Ooze	Other GCode
Cooling	Temperature
Minimal Layer Print Time: 1.0 🚔 sec	Bed Temperature: 40 🛉 ° C
Default Fan Speed: 100 🚔 %	Left Extruder: 215 🔹 °C
Maximum Fan Speed: 100 🚔 %	Right Extruder: 215 🔹 °C
Turn Fan On At Layer: 2	
Minimal Printing Speed: 10.0 🚔 mm/s	
Restore Defaults	OK Cancel

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

These settings as follows:

Multiple Extruders Ooze Control:

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

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

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

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

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

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



Advance Settings Layer Infill Support and Re	ft Coolin	ng Ooze	Other	GCode	<u> </u>
Retract					
🔽 Enable Retraction					
Retract Speed:	20.0	🚔 mm/s		Restart Speed:	20.0 🚔 mm/s
Retract Material Amount:	\$.0	mm		Z hop:	0.000 🚔 mm
Minimal Travel of Retraction:	0.5	mm		🔽 Force Retract	On Layer Change
Minimal Amount of Retraction:	0.02	mm		🔲 Avoid Travelin	ng Through Holes
Extra Restart Amount:	0.00	mm			
-Multiple Extruders Ooze Control					
🔽 Enable WipeWall					
Retract Speed of Extruder-swit	ch:	20.0	🚔 mm/s		
Retract Amount of Extruder-swi	tch:	3.50	mm		
Restart Speed of Extruder-swit	ch:	20.0	🚔 mm/s		
Extra Restart Amount of Extrud	er-switch:	0.00	mm		
Restore Defaults					OK Cancel

Filament Setting

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

1. Edit the existing template.

Printer—Filament Settings





Filament Settings					? ×
Туре:	ABS 1.75mm		•	Add	Filament
Diameter:	1.750	▲ mm			
Density:	1000.00	🔹 kg/m3			
Price:	30.00	\$/kg			
Flowrate:	100.00	∲ %			
			ОК		Cancel

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

2. Add a new template.

1 Add Filament		? <mark>×</mark>	
Type:	new filament		
Diameter:	2.930	▲ ▼ mm	
Density:	1300.00	▲ kg/m3	
Price:	30.00	* \$/kg	
Compensation:	100.00	<u>◆</u> %	
		OK Cancel	

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

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



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