

i-fast Slicer operation process

NOTICE: QidiPrint slicer does not support i-fast printing via USB cable, support ethernet connection printing.



For Windows users

Slicer install

1 Insert the USB flash drive into the computer.



(2) Create a new folder on the computer desktop, copy the USB flash drive software to the new folder.



③ Double click to run the installer, install the software as below pictures.



Please click on the desktop shortcut to open the slicer.

Choose the printer type "i-fast" .





(A) Introduction of Mouse Function

Left Button: Click to select and the model for operations.

Wheel: Roll to zoom in and out

Click and drag to pan the camera.

Right Button: Click and drag to change camera viewpoint.



(B) Icon function introduction

Menubar: Includes Control Panel,



on this button to slice

(C) First print

- ① Select your print type.
- ② Select your material type.
- ③ Select your printing precision(layer height).
- ④ Select an infill density.
- (5) Click the checkbox to enable print support.
- 6 Click prepare to slice your model.
- ⑦ Save the file or use Wifi to send the file.

File	Edit	View	Settings	Control Panel	Extensions	Preferences	Help											
	,											Select		type.	i-fast			••
												elect you		type.@>	Extr	uder	💷 Extru	der
									Select	vour r	rintina	precisio	n (layer he	eight) (3)				_
															Material	PLA		~
															Config	Fine		~
														ensity.@>	Layer Heigh	it	7	
									Click			to enab	le print sup	oport.⑤>	Infill			
															Add Suppo	15%		
															Extruder	·		<
															8 Layer			<
															🖾 Infill			<
															Addition	IS		<
							Follow t	ho stor	os to m	ako v	our fir	rct prin			Speeds			<
								ine ster	psiton	ake y		ist prin			I Temper			<
															Cooling			<
															Support			<
															Advance	:d		<
(((*																		
									Sta	t								
									41	21								
			zÆ												(Expert m	ode	
			THE															
													lice your n			Prepar	10	
		0.0 x 0	.0 x 0.0 m	m Print Time	: 00h 00min	Material Cos	t: 0.00m / 0g		Sa	ve the	file or u	use Wifi	to send th	e file.		Crieban		

(D) Addition Feature Instruction

1. Adding the support manually

The operation of adding the support manually. First, please click the model, and add support tools manually in the toolbar on the right side of the main interface **•** .

Here you can set the basic parameter for support (including the support size, support type, support density), then click on the model to add support part where you want to add on the model.Then click the done.



2. Support Blocker

Cick the model then click the support blocker **b**, click on the part of the model that does not need the support, and the support will not be automatically added to this part.



3. The function of the fan

	🚱 Cooling			
Chamber loop	Enable Print Cooling 🗸			
Enables the chamber looping fans while printing.	Chamber loop 🗸			
	Support	<		
	Advanced	<		
Build Volume Temperature	emperature Build Plattial Layer 60 °C			
The temperature of the	Build Voperature 0	°C		
environment to print in. If this is 0,	🛞 Cooling			
the build volume temperature will not be adjusted.	Enable Print Cooling 🗸			
not be aujusteu.	Chamber loop			



Enable Print Cooling	😵 Cooling	
Enables the print cooling fans while printing. The fans improve print	Enable Print Cooling 🗸	
	Chamber loop 🗸	
quality on layers with short layer	Support	<
times and bridging.	Advanced	<
Affects	Expert mode	
Fan Speed		



4. Expert model



5. Z-axis shifting

The extruder is too close to the platform, resulting in printing failure. Set Z offset 0.2 to increase the distance between the extruder and the platform by 0.2mm.



ild platform has relative

model

lue on

The extruder is too far to the platform, resulting in printing failure. Set Z offset -0.2 to reduce the distance between the extruder and the platform by 0.2mm.



ick on

i plate.

6. Control Panel

You can remotely control the printer and view the current status of the printer.



Save the gcode file

Method 1: When the USB flash drive is inserted into the computer, the gcode file can be save to the USB flash drive or the computer.



Method 2 : When the USB flash drive isn't inserted into the computer, the gcode file can be saved to the computer.

♥ Save to File				×		g Fine Height 0.2mm
← → ~ ↑ 🖬 > Thi	s PC > Local Disk (D:) >	✓ Č Searc	th Local Disk (D:)	م.	Infill	15%
Organize 🔻 New fol	der		lise .	- 0	Add s	
	Name Adobe AppData Creative Cloud Files Documents	Date modified 7/18/2020 2:43 PM 7/16/2020 2:45 PM 4/8/2020 11:12 AM 5/26/2020 3:38 PM	Type File folder File folder File folder File folder	Size	Retra Retra Enabl	e Retraction ction Distance ction Speed e Coasting Ction Speed Ction Speed
Local Disk (D:) Local Disk (E:) Local Disk (F:) Network DESKTOP-S4525CH	★ Favorites ■ maker新UI ■ Pictures ■ 初岐	4/19/2020 11:11 AM 5/12/2020 8:55 AM 7/15/2020 10:43 AM 5/12/2020 8:47 AM	File folder File folder File folder File folder		Line V Wall Top L	Height 0.2 Nidth 0.4 Line Count 2 ayers 4
HWL-2020 WIN-8M9P23OBHEI File name: Ozoba	< ot_Tie_Fighter			~	Print R Int	m Layers 3 Thin Walls 5 All Density 15
Save as type: Ozob	ot_Tie_Fighter (*.gcode)			~		Pattern Zig Zag
A Hide Folders			Save Ca	ance	Z Sea Print Build Build Skirt Skirt	Iditions m Alignment Shortes In Advance Plasion Type Skirt Pla Extruder Extrude Line Count 2 Distance 4 e Prime Tower
2	+ THE					

Extruder 2 printing

1. Load the single color model , then click the model and select the Extruder 2.



2. Click "Setting", select "Extruder 2" and set the printing parameters.

Extruder Extruder	
Material PLA 🗸	
Config Fine ~	
Layer Height	
Infill	,
Add Support	
Additions ~	
Z Seam Alignment Shortest V	
Print In Advance	
	When pr
Build Plasion Type Skirt ~	
Build PlExtruder 🤊 Extruder 2	extruder
Skirt Line Count 2	((Euclidean all
Skirt Distance 4 mm	"Extrude
Enable Prime Tower	
Enable Ooze Shield	
🛠 Speeds <	
I Temperature <	
Cooling <	
🕤 Support 🗸 🗸	
Add Support 🤊 🗸	Choose t
Support Extruder 🤊 Extruder 2	au na na mt
Support Placement Everywhere 🗡	support

When print with the left extruder, please select "Extruder 2"

Choose the "Extruder 2" for support extruder as well.

3. Click "Prepare" to generate the gcode file , save the file to Removable Driver (U-disk). Insert the U-disk into the 3d printer and start to print.

	Support Placement Supportng Angle	Everywhere ~
	Support Pattern	Lines ~
	Expert	mode
Solid X-Ray Layer	Assembling 🖉	
🏺 🖉 🗗 🗗 🚺 🛛 📚	Prep	are

34

Double color printing

1.Load the double color model.



2. Combine model

Click the model, and select the correct extruder for each model in the toolbar (red color is for Extruder 1, yellow color is for Extruder 2).



3. Click the "Select All Models" in the "edit", then click "Merge Models" and "Reset All Model Positions".



4. Set the parameter of the 1 and 2 Extruder.

Extruder 1

Extruder Extruder
Material PLA 🗸
Config Fine ~
Layer Height
Infill 15%
Add Support
🕒 Extruder <
🗑 Layer <
🖸 Infill <
🖉 Additions 🛛 🗸 🗸
Z Seam Alignment Shortest 🗸
Z Seam Alignment Shortest Print In Advance
Print In Advance
Print In Advance Build Plasion Type Skirt
Print In Advance Build Plasion Type Skirt Build Pla Extruder Extruder 1
Print In Advance Image: Constraint of the second secon
Print In Advance Build Plasion Type Skirt Build Pla Extruder Extruder 1 Skirt Line Count 2 Skirt Distance 4 mm
Print In Advance Build Plasion Type Skirt Build Pla Extruder Extruder 1 Skirt Line Count 2 Skirt Distance 4 mm Enablee Tower V
Print In Advance Build Plasion Type Skirt Build Pla Extruder Extruder 1 Skirt Line Count 2 Skirt Distance 4 mm Enablee Tower Prime ToPosition 50 mm
Print In Advance Build Plasion Type Skirt Build Pla Extruder Extruder 1 Skirt Line Count 2 Skirt Distance 4 mm Enablee Tower Prime ToPosition 50 mm Prime ToPosition 25 mm

Extruder 2

Extruder	Extruder
Extruder	
Material PLA	~
Config Fine	•
Layer Height 🖳	0.2mm
Infill 🛄	<u>, , , , , , , , , , , , , , , , , , , </u>
Add Support	
🕒 Extruder	<
🗑 Layer	<
🔯 Infill	<
Additions	
Z Seam Alignment	Shortest 🗸
Z Seam Alignment Print In Advance	Shortest 🗸
-	 Image: A start of the start of
Print In Advance	v Skirt v
Print In Advance Build Plasion Typ	v Skirt v
Print In Advance Build Plasion Typ Build Pla Extrude	✓ e Skirt ✓ Extruder 1■✓
Print In Advance Build Plasion Typ Build Pla Extrude Skirt Line Count	✓ e Skirt ✓ Extruder 1■✓ 2 4 mm
Print In Advance Build Plasion Typ Build Pla Extrude Skirt Line Count Skirt Distance	✓ er Skirt ✓ 2 4 mm
Print In Advance Build Plasion Typ Build Pla Extrude Skirt Line Count Skirt Distance Enablee Tower	ve Skirt v r Extruder 1 V 2 4 mm 50 mm
Print In Advance Build Plasion Typ Build Pla Extrude Skirt Line Count Skirt Distance Enablee Tower Prime ToPosition	ve Skirt v r Extruder 1 V 2 4 mm 50 mm
Print In Advance Build Plasion Typ Build Pla Extrude Skirt Line Count Skirt Distance Enablee Tower Prime ToPosition Prime ToPosition	ve Skirt v r Extruder 1 ● v 2 4 mm 50 mm 25 TICK mm

Once you set the parameter of the left one, the right one will synchronized changed, except the setting of infill, speed, temperature.

5. Save the gcode file.

Method 1 : When the USB Pen Drive is inserted into the computer, the gcode file can be saved to the USB Pen Drive or the computer.



Method 2 : When the USB Pen Drive isn't inserted into the computer, the gcode file can only be saved to the computer.



Virtual Graph:



The effect diagram when print out:



① Enable Prime Tower

Adding printer profile for simplify 3D

1. Open Simplify3D, select "import FFF profile" in the File



2. Import the i-fast printer profile for Simplify 3D from the USB flash drive that we provided.



3. The message pops up when import successful.



4. Open the FFF setting, choose "Qidi Technology i-fast" that we just added in the select profile section.

									>
rocess Name:	Name: Process1								
	Default Default		-	-	Update Pr Quality		Save as New	Remo	ve
Auto-Configure				-			Configure Ext	ruders	
PLA	-	• • •	Medium	•	•	Both	Extruders		•
General Settin	gz.								
Infill Percent	age:			10%	Inclus	le Raft	🗹 Genera	te Supp	ort

