iNSTONE

User Manual
for Mini 3D Printer

Content

Technical Parameters

Name	Mini 3D Printer
Model	EASIER
Consumable	PLA
Printer size	195*210*275mm
Build volume	110*110*125mm
Printing accuracy	0.1mm(Max)
Printing principle	FDM
Printing temperature	210°C
X, Y, Z axis position accuracy	0.1mm
N.W.	2.4kg
G.W.	4kg

Packing List

- MINI 3D Printer *1
- Power Adapter *1
- Power Cable *1
- Printing plate *1
- TF card *1
- Card Reader *1
- Glue Stick *1
- PLA Filament (280g) *1

Note：Once you received the parcel, please check all the parts; if anything missed, contact us (sales01@instone3d.com) ASAP

Installation

I. Remove the protecting clip on the plain shaft

II. Install the Feed Pipe

1. Press down the feeding coupling aside the radiator, then insert the white feed pipe
2. Insert the other end of the feed pipe into the socket of the remote extruder

III. Place the printing plate

IV. Feed

1. Unpack the PLA filament from the package, and find out the end of it
2. Bend the head material straightly, and trim the end of the material with scissor
3. Press the spring of the extruder by hands, then insert the filament from the hole on the bottom of the extruder to the other end of the Feed Pipe (until you can't insert anymore). Then hang the filament tray on the hook on side of the printer (Note: Arrange the filament neatly and DO NOT in a mess).

V. Inserting the TF Card

VI. Plug in and Power on

Instructions

I.Printing

1. The LCD screen will display the home page after starting up
2. Press the control knob to enter into the main menu; rotate the knob to select the option "Print from SD card", and press the knob to enter into the submenu
3. Rotate the knob, and select the option "print file"; press the knob to enter into the submenu
4. Rotate the knob, select the file to be printed, then press the knob to begin printing

II. Taking down the finished printing models

Take down the printing plate after printing finished, then take down the finished printing model by a slight bending. After that, clean the surface of the printing plate (washable).

III.Pause printing

1. During the printing process, press the control knob, and the control menu pops up on the display screen;
2. Rotate the knob, select the option "pause print", and press the knob to pause printing.

IV.Continue printing

☆ **This function is only applicable to continue printing after pausing in condition of continuous power on**

1. Press the control knob, and the operation menu pops up on the display screen;
2. Rotate the knob, select the option "continue print", and press the knob to continue printing.

V. Stop print

☆**It is not available to continue printing after stopping printing, so please BE CAUTIOUS when using this function**
1. During the printing process, press the control knob, and the control menu pops up on the display screen;
2. Rotate the knob, select the option "stop print", and press the knob to stop printing.
Note: It's IMPOSSIBLE to continue printing after "stop print".

VI. Retract/reload the filament

This printer has been equipped with the function of automatic retract of filament, which can be utilized in condition that the consumable is to be used up or it is necessary to change to different colors

1. At the home page, press the control knob, then the main menu pops up on the display screen

2. Rotate the knob, select the option "Auto-stripper", and press the knob to enter into the operation

3. "Processing..." will show on the interface, and please wait patiently

4. When the automatic retract of material ends up, the display screen will return to the home page, and the printing head's temperature will fall, then the operation is finished

Note

I. Clogs & Clear the nozzle

In normal conditions, there's no need to worry about this issue as high quality printing filaments are selected for this printer. If you select other consumables, the nozzle may be blocked due to impurities in the material or other reasons, in which condition the following steps may help you.

1. Press the control knob at the home page, press the control knob, and the main menu pops up on the display screen;

2. Rotate the knob, select the option "heating print head", and press the knob to enter into the operation;

3. "Processing" appears on the interface; at this time, the printer is heating the nozzle, and please wait patiently;

4. When the heating is finished, and the display screen returns back to the home page, it shows that the temperature of the nozzle is 230℃;

5. Insert hard the filament into the nozzle to clear, until you can feed smoothly.

6. Power off for cooling down after clearing

II. Adjust the height of the nozzle

Too high or too low position of the nozzle will influence the printing (too high, the printing material cannot stick on the bottom plate; too low, the nozzle will pin the bottom plate, and the filament cannot be extruded). Adjust the positioning knob properly as per the following figure, to adjust the distance between the nozzle and the bottom plate:
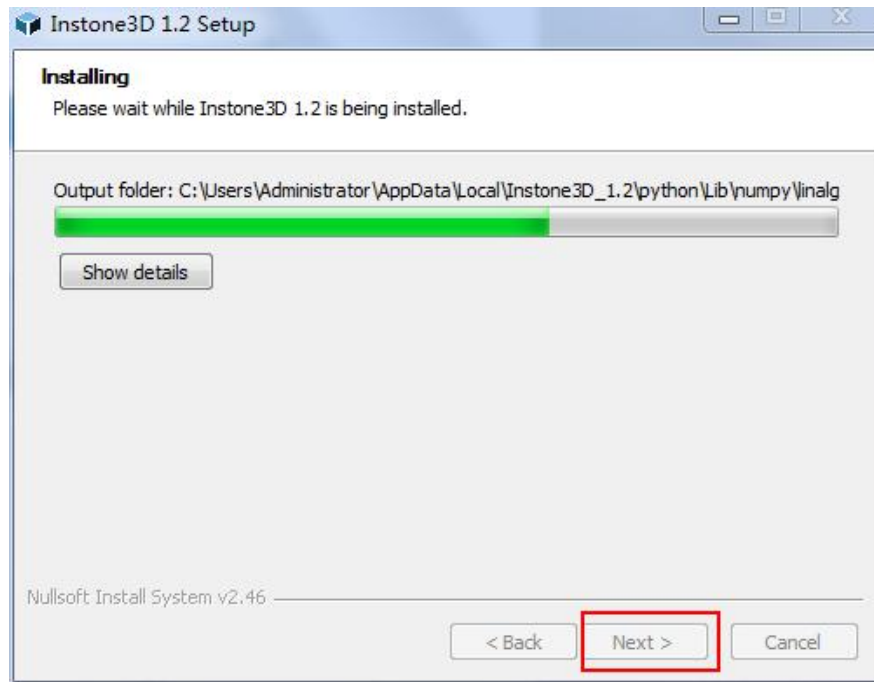
I. Installation and Operating Guide of Slicing
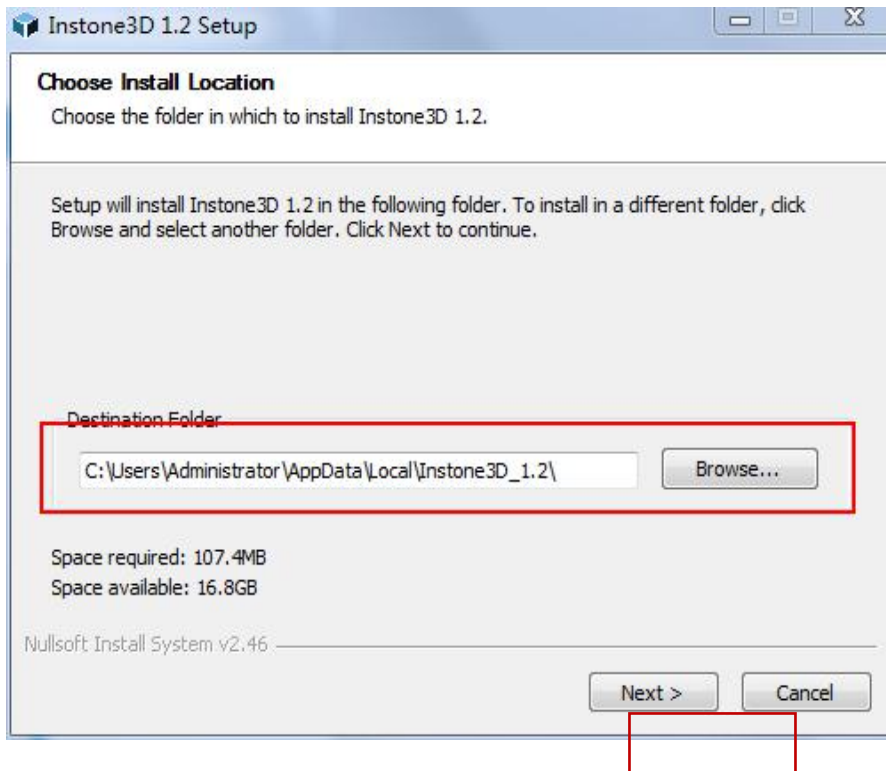
1. Double click the file Instone3D_1.2.exe in TF card to enter software installation interface. User can select installation catalog in the red frame, then single click [Next]
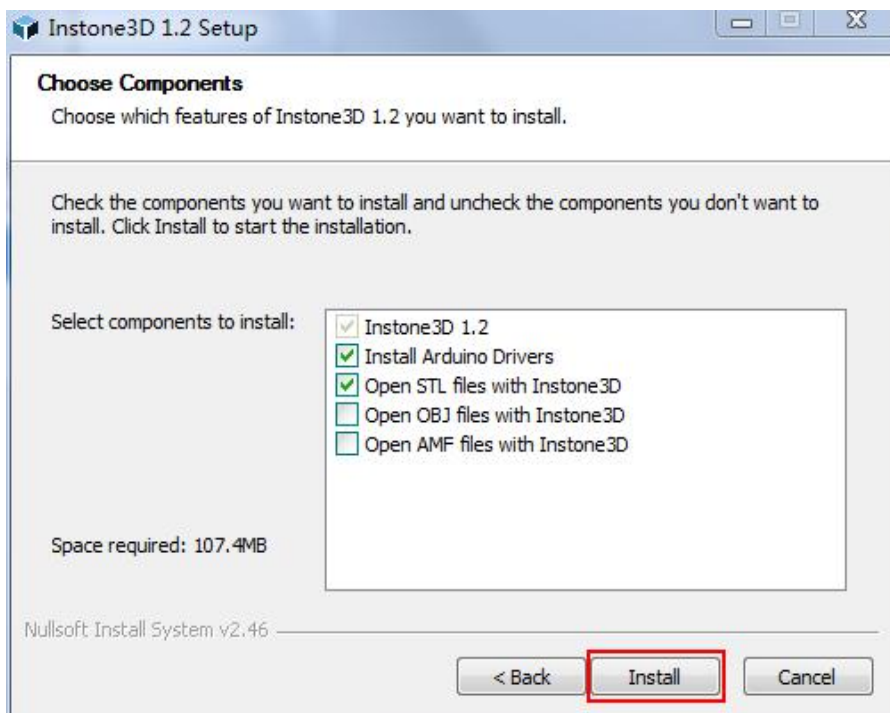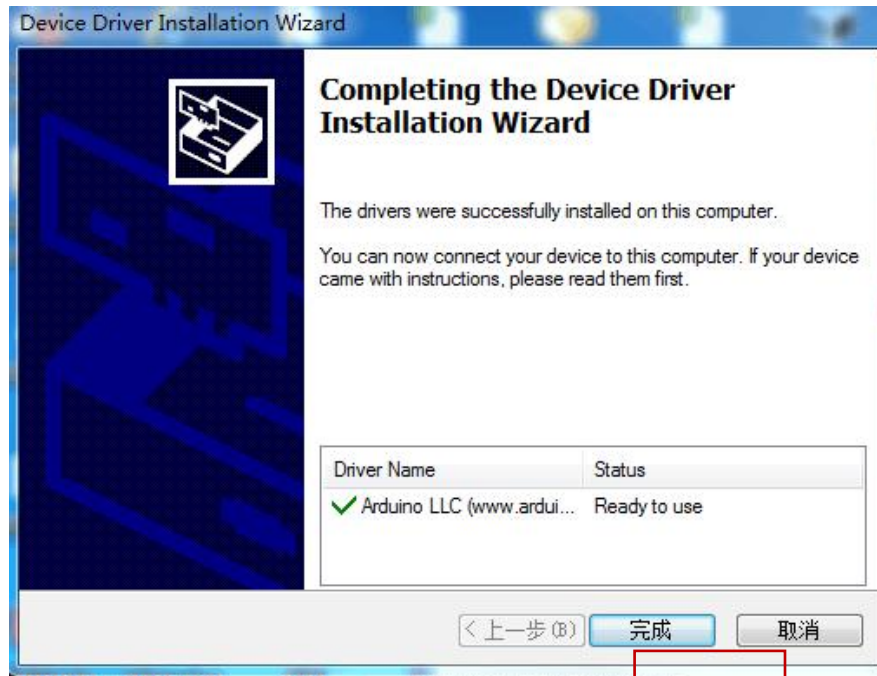

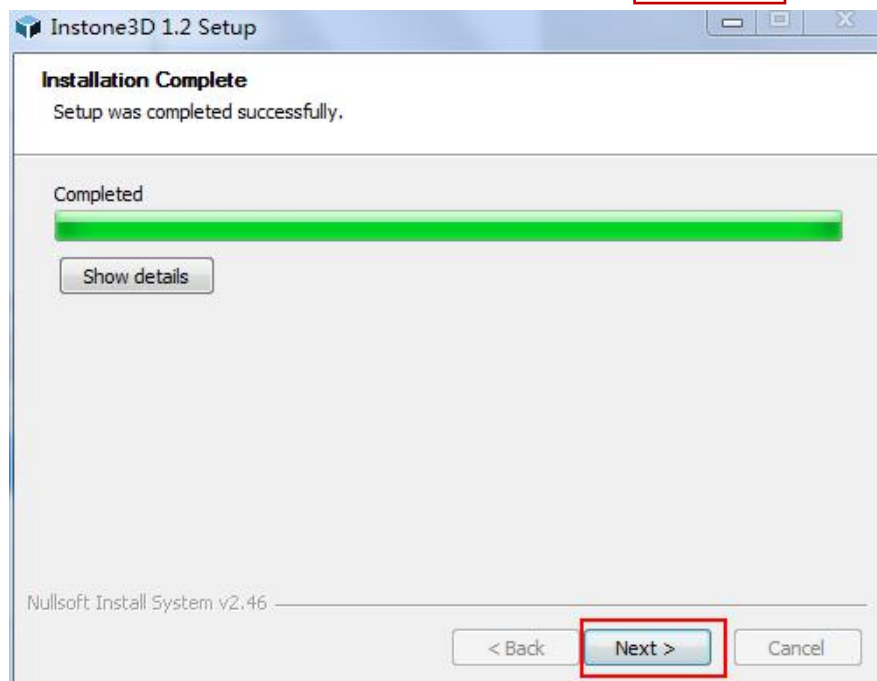2. Installation option, Click [Install] for installation.

3. Installation is processing.

4. Single click [Next] to install drive

5.After installing drive, single click [Finish]

6. Single click [Next] to complete installation.

II. Use of Slicing Software:

1. Understand and Know Well About Software Interface:

Import model	View model	Transparent
Save Gcode file	Normal	X-Ray
	Overhang	Layers

Rotate	Scale	Mirror
Lay flat	To max	Mirror X
Reset	Reset	Mirror Y
		Mirror Z
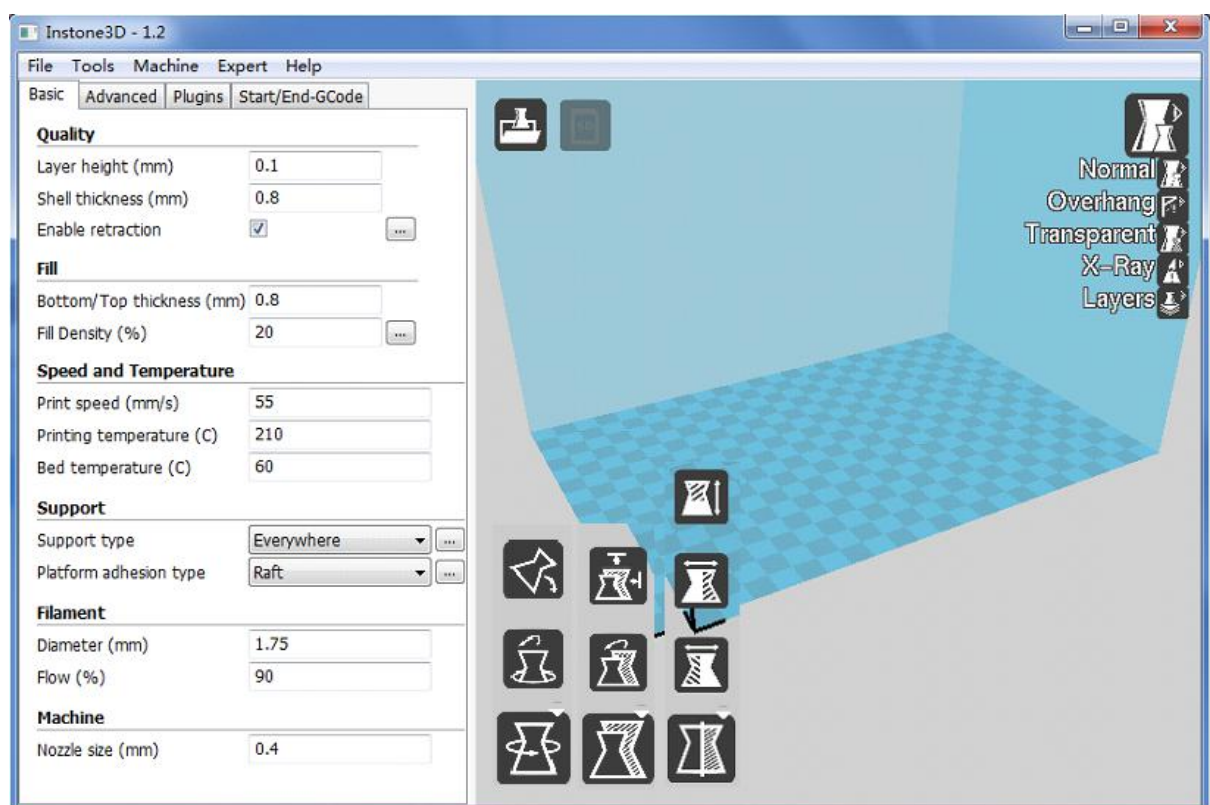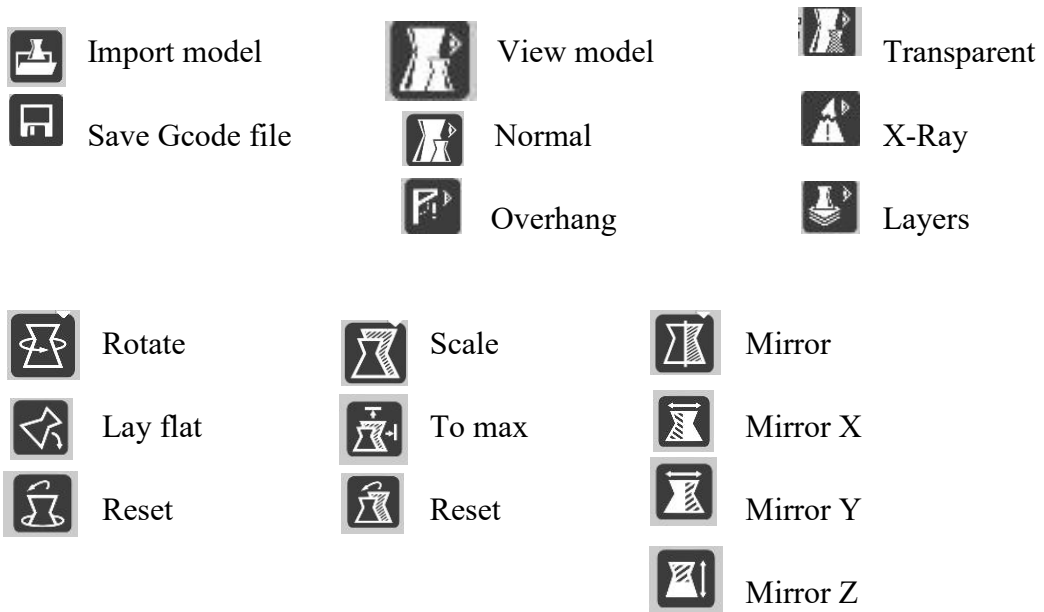
2. Machine Settings:

Select corresponding machine for setting, click the "Machine" and select "2Um1+Normal Print", as shown in the following picture;
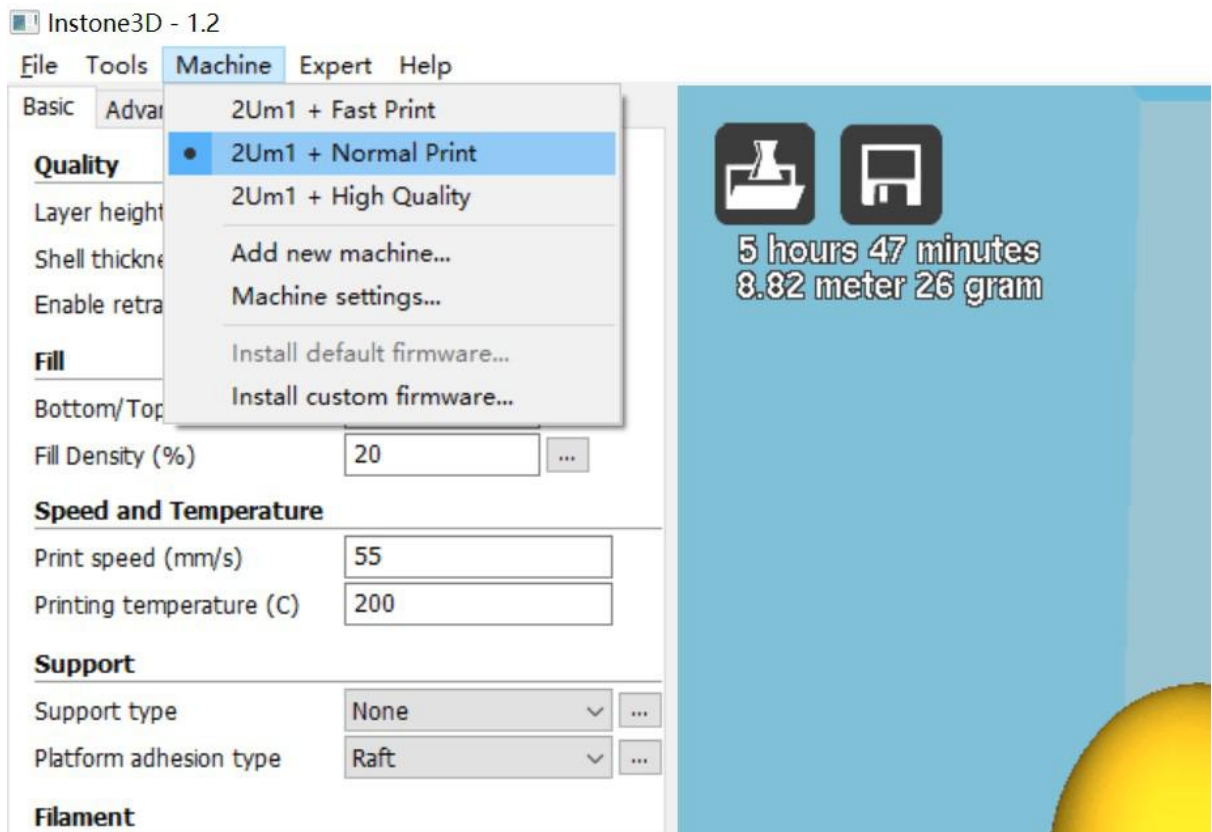
3. Load Model File:

Open the model file, it supports file in the format "stl,obj,dae,amf,bmp,jpg,jpeg, png,g,gcode". There are three ways of opening the model file: (1) click "file", select "load model file", open the catalog where the model file is stored and sel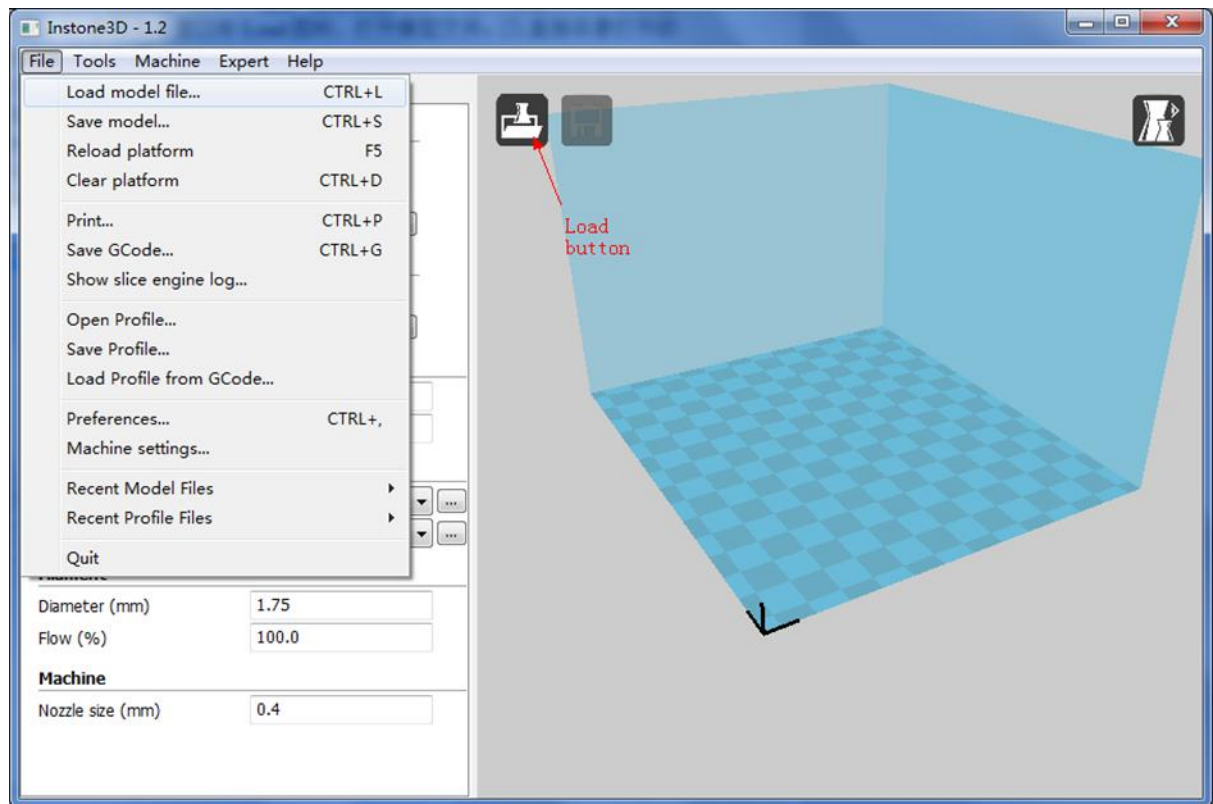ect the file; (2) click Load icon in the window directly to open the model file; (3) Drag the model file to be opened in the blue region as shown in the picture directly.

4. Process and Edit Model Files:

User can rotate, zoom, mirror the model according to the demand.
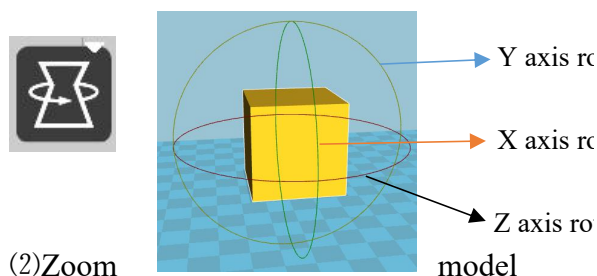
⑴Rotate model

Operation instruction:
Single click left key of mouse for selecting corresponding rotation line, drag the mouse to rotate the model with the unit of 15°(press Shift+ left key of mouse to rotate model with the unit of 1°)

⑵Zoom model

Single click left key of mouse to s

drag them to zoom out the corresponding axial size. Meanwhile, the model center hints the value. User can input specific figure in the dialog box. Please operate the model as per the following picture.
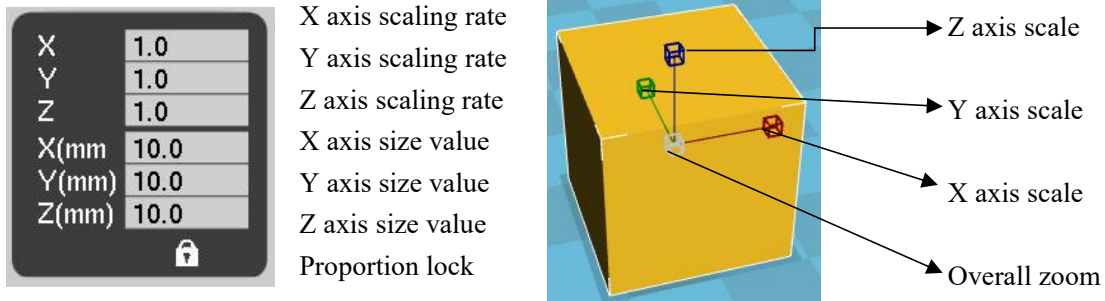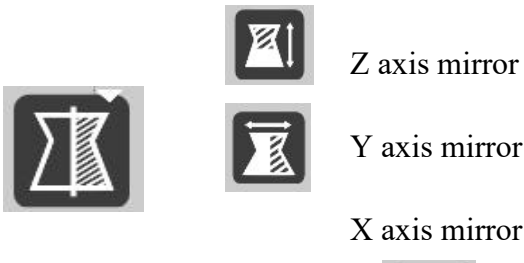
X	1.0
Y	1.0
Z	1.0
X(mm	10.0
Y(mm)	10.0
Z(mm)	10.0
	🔒

X axis scaling rate
Y axis scaling rate
Z axis scaling rate
X axis size value
Y axis size value
Z axis size value
Proportion lock

Z axis scale
Y axis scale
X axis scale
Overall zoom

(3)Mirror model

Z axis mirror

Y axis mirror
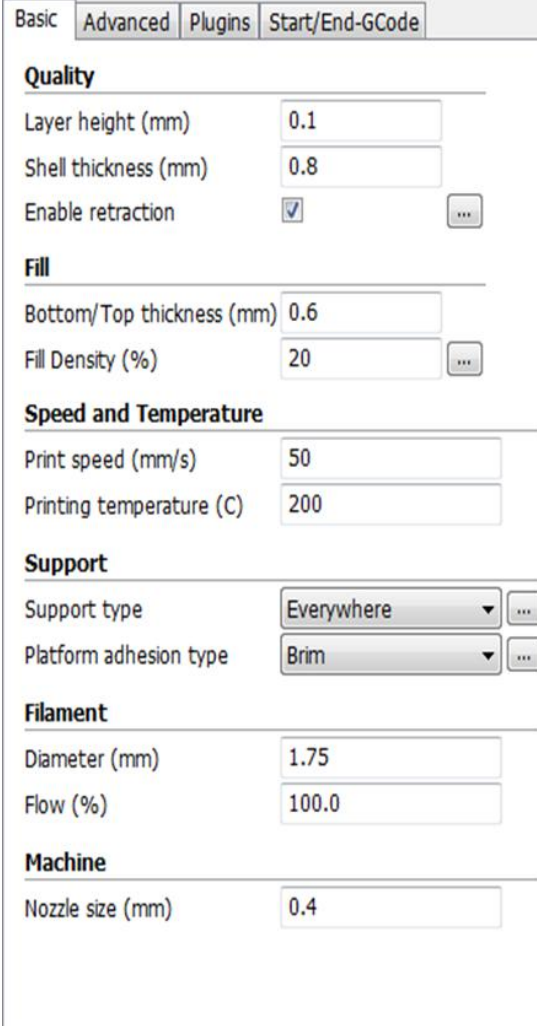
X axis mirror

Single click the icon, three sub-options pop up, which are Z axis mirror image, Y axis mirror image and X axis mirror image from top to bottom. Single click corresponding option to realize transformation.

(4)Function introduction of right key menu

Move the mouse on the model, single click right key, then the menu pops up, as shown in the following picture:

Center on platform
Delete object
Multiply object
Split object into parts
Delete all objects
Reload all objects
Reset all objects positions
Reset all objects transformations

Put the model in the middle of print plate.
Delete model
Copy multiple models (quantities can be set manually)
Split the model into separate component
Delete all the models on the platform
Reload the model to software (initialize)
Reset the position of all the models
Reset the changes of all the models

5. Basic Parameter Setting of Slicing:

| Basic | Advanced | Plugins | Start/End-GCode |

Quality

Layer height (mm)	0.1
Shell thickness (mm)	0.8
Enable retraction	☑

Fill

| Bottom/Top thickness (mm) | 0.6 |
| Fill Density (%) | 20 |

Speed and Temperature

| Print speed (mm/s) | 50 |
| Printing temperature (C) | 200 |

Support

| Support type | Everywhere |
| Platform adhesion type | Brim |

Filament

| Diameter (mm) | 1.75 |
| Flow (%) | 100.0 |

Machine

| Nozzle size (mm) | 0.4 |

⑴**Quality**

Layer height: it is usually set as 0.2, Setting range: 0.1-0.3. The lower the value is, the printing is smoother and the corresponding printing time is longer.

Shell thickness: it is usually set as integral multiple of nozzle aperture. The common value is 0.8.

Enable retraction: when retraction is enabled, the nozzle rolls back filament automatically and improves the filament drawing.

⑵**Fill**

Bottom/top thickness: the thickness of top layer and bottom layer for the model is usually 1.2mm.

Fill density: the fill density of the model is 10%, if the intensity is not required to be high. The higher the value is, the higher the intensity is, and the longer the corresponding printing time is.

⑶**Speed and temperature**

Print speed: it is the default overall speed and usually set as 60. Too high speed results in bad effect of model formation.

Printing temperature: nozzle temperature. Usually PLA consumable is set as 200. It depends on the physical property of printing consumables.

Thermal bed temperature: refers to the temperature of printing platform. The machine doesn't support platform heating, so the parameter is invalid.

⑷**Support**

Support type: three options: ①None; ②Touching build-plate; ③Everywhere; usually select②, establish support according to the demand.

Platform adhesion type: three options: ①None; ②Brim; ③Raft. Usually select②, Option ② and ③ makes the base layer of model adhere to printing platform better, Brim can solve edge warping problem effectively. however, when the bottom surface is big, it increase the difficulty of taking out the model. When Raft can make model on the print plate easier to take out, but the bottom surface of model is not smooth, user can select correspondingly.

⑸**Filament**

Diameter: the diameter of filament for nozzle is 1.75mm.

Flow: percentage of consumable usage amount is usually set as 100%.
⑹**Machine**
Aperture of nozzle: 0.4mm.

6. Senior Parameter Setting of Slicing:

(It is applicable to professional printing effect contrast. The common printing just adopts the default value without entering this setting)
Click next tab control "Advanced" to enter advanced printing setting.

⑴**Retraction**
Speed: corresponds to retraction function of basic parameter for detailed setting.
Distance: the length of filament for retract.
⑵**Quality**
Initial layer thickness: thickness of the bottom layer. If the bottom layer is thick, it is good for increasing adhesive force.
Initial layer line width: line width when printing the initial layer
Cut off object bottom: subsidence model. The subsidence part will not be printed.
Dual extrusion overlap: the overlapping length of consumables at the joint when double-color nozzle prints.
⑶**Speed**
Travel speed: the movement speed of nozzle in non-printing.
Bottom layer speed: the speed of printing the first layer. When it is low, the adhesive force is increased.
Infill speed: the speed of printing the internal infill part.
Top/bottom speed: speed of printing top/bottom layer.
Outer shell speed: speed of printing the outer shell of the model.
Inner shell speed: speed of printing the inner shell of the model.
⑷**Cool**
Minimal layer time: minimum time consumed for printing each layer
Enable cooling fan: turn on fan in the process of printing
Special notes:

(1)Dual extrusion overlap (the machine doesn't support this function, so the parameter is invalid): it is used for double-color printing to make the fusion of two colors more natural.

(2)Minimal layer time: ensure PLA to have full cooling time and make the printing effect perfecter.

(3)Enable cooling fan: please check this option when printing with PLA filament.

7. Additional Parameter Setting of Slicing:

(It is applicable to professional printing effect contrast. The common printing just adopts the default value without entering this setting)

Select menu bar "Expert" -> "Open expert setting" to enter the expert printing setting.

Retraction	
Minimum travel (mm)	1.5
Enable combing	All
Minimal extrusion before retracting (mm)	0.02
Z hop when retracting (mm)	0.0

Skirt	
Line count	1
Start distance (mm)	3.0
Minimal length (mm)	150.0

Cool	
Fan full on at height (mm)	0.5
Fan speed min (%)	100
Fan speed max (%)	100
Minimum speed (mm/s)	10
Cool head lift	☐

Infill	
Solid infill top	☑
Solid infill bottom	☑
Infill overlap (%)	15
Infill prints after perimeters	☑

Support	
Structure type	Lines
Overhang angle for support (deg)	60
Fill amount (%)	15
Distance X/Y (mm)	0.7
Distance Z (mm)	0.15

Black Magic	
Spiralize the outer contour	☐
Only follow mesh surface	☐

Brim	
Brim line amount	5

Raft	
Extra margin (mm)	5.0
Line spacing (mm)	3.0
Base thickness (mm)	0.3
Base line width (mm)	1.0
Interface thickness (mm)	0.27
Interface line width (mm)	0.4
Airgap	0.0
First Layer Airgap	0.22
Surface layers	2
Surface layer thickness (mm)	0.27
Surface layer line width (mm)	0.4

Fix horrible	
Combine everything (Type-A)	☑
Combine everything (Type-B)	☐
Keep open faces	☐
Extensive stitching	☐

Ok

(1)Retraction

Minimum travel: the movement distance that triggers retraction.

Enable combing: if user turns off the function, when the nozzle moves, it keeps retract.

Minimum extrusion before retracting: there should be a certain quantity of extrusion before retract.

Z hop when retracting: when starting retract, Z axis raises slightly so as to improve the filament drawing.

(2)Skirt

Line count: print a round of filament surrounding model at the bottom layer.

Start distance: the distance between the skirt and the first layer of model.

Minimal length: the minimal length of skirt.

(3)Cool

Fan full on at height: height triggered when the fan runs at full speed.

Fan speed min: the lower limit rate of rotation of fan.

Fan speed max: the upper limit rate of rotation of fan.

Minimum speed: minimal filament feeding speed, avoid too slow printing speed causes consumable overflow.

Cool head lift: when the minimal extrude speed is cooled too slowly, withdraw the nozzle to cool for a while.

(4)Infill

Solid infill top: check whether the printing model is capped.

Solid infill bottom: check whether the bottom of printing model is covered.

Infill overlap: if the overlapping value of internal packing and outer wall is bigger, the better they are connected, but more consumables are needed in this case.

⑸**Support**

Structure type: the type of support structure.

Overhang angle for support: minimum angle triggering the printing support.

Fill amount: fill rate of support.

Distance X/Y: the distance between support and model subject on X/Y axis.

Distance Z: distance between support and model subject on Z axis.

⑹**Black magic**

Spiralize the outer contour: after it is enabled, it can print the surface in a spiral shape and eliminate the interface of each layer.

Only follow mesh surface: after it is enabled, it only prints the surface layer, without printing top, bottom and internal support.

⑺**Brim**

Brim line amount: the bigger the number of brim is, the adhesion effect between it and platform is better, but it reduces the printing size.

⑻**Raft**

Extra margin: additional round of edge on the periphery of raft.

Line spacing: the distance of each line when the raft prints.

Base thickness: thickness of raft printing.

Base line width: base layer lines width of raft

Interface thickness: thickness of the interface layer, which connected the raft and the model.

Interface line width: line width of interface layer.

Airgap: the gap between the raft and model.

First layer airgap: gap between the last layer of the raft and the first printing layer.

Surface layers: amount of surface layers put on top of the raft.

Surface layer thickness: thickness of each surface layer.

Surface layer line width: width of the lines for each surface layer

⑼**Fix horrible**

Combine everything(Type-A): namely integrate all the separate parts of the model into one, Type-A is dependent on the model normal and tries to keep some internal holes intact.

Combine everything(Type-B): namely integrate all the separate parts of the model into one, Type-B ignores all internal holes and only keeps the outside shape per layer.

Keep open faces: software will repair some small damaged surfaces automatically.

Extensive stitching: repair the damaged surface at a higher level.

8. Browse slicing file:

Click the icon on the top right corner and select Layers. The layer number bar occurs on the right side of interface. Move up and down to view the printing path. Click Normal to restore normal model view.
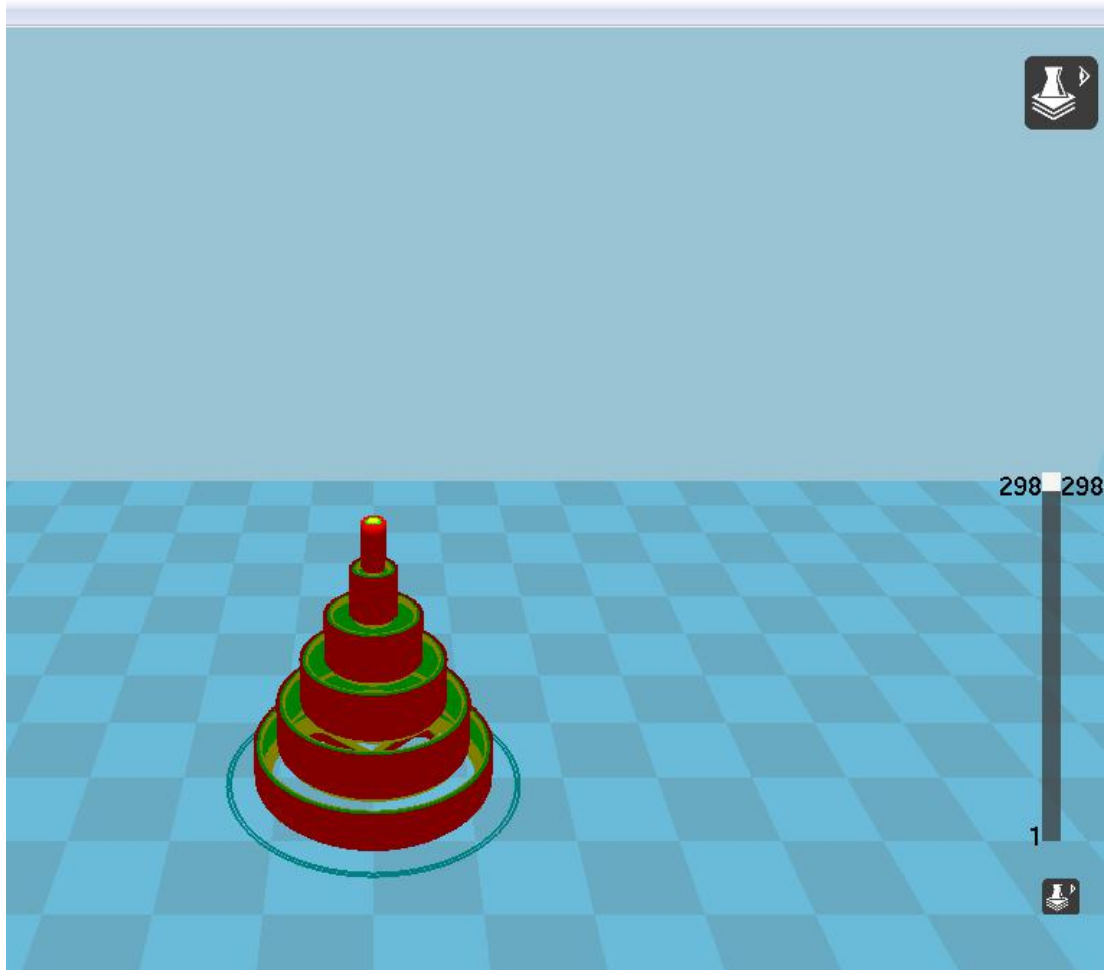
9. Export of slicing file:

After completing all the settings, please click the button indicated in the following pictures or select "file" —"Save GCode..." to save GCode printing file after slicing. (if SD card has been inserted to computer, the file will be saved to SD card automatically)
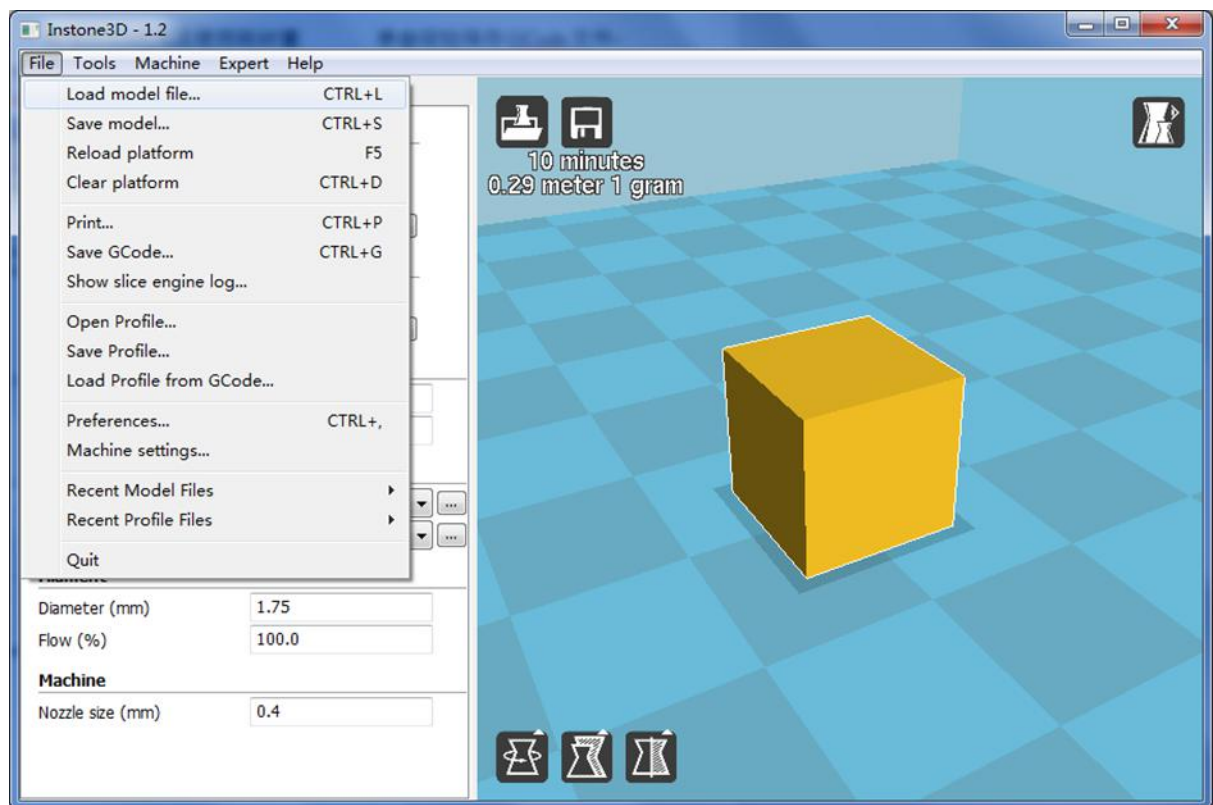

GCode file in the cache Gcode

Printing time and used quantity

of filament

Click to save

Online Printing and Updating Firmware

I. Online Printing:

Note: online printing has offline risk: if USB data is disconnected or the computer is shut down occasionally, the equipment will stop printing and fail to continue the last printing.

1. Install the driver: When installing "Instone3D 1.2" software, it has finished the installation of driver as default. If the equipment fails to be identified online, please contact us to ask for driver installation package.

*Mac: "Drivers" - "Mac" - "CH341SER_MAC"

2. Please use the standard USB data line to connect computer and printer

3. Open "Instone3D 1.2" software, click menu bar "file"—"machine setting", check "communication settings" at right bottom corner of dialog box---"serial port"---"COM 3" (please select according to the port distribution of computer) "Baud rate" —"AUTO", click "OK" button.

4. Icon as shown in the right picture pops up. When "USB" appears above the icon, it means successful connection.

5. Load model on "Instone3D 1.2" software according to the Operation guide in the previous chapter. Then set slicing parameter. When the software slicing is finished, click "USB" icon, the printer starts heating and printing automatically.
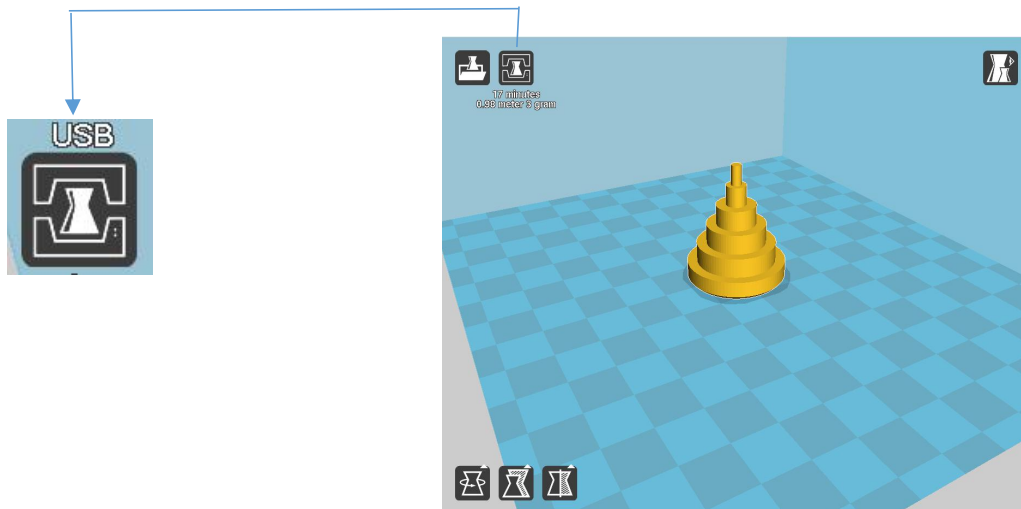
II. Update Firmware:

(only supported by specific machines)

The machine has been installed the latest firmware prior to leaving factory. If it is not for maintenance or commissioning, please don't update the firmware programs.

1. As is mentioned above, connect printer to computer successfully, click "machine" in the software menu bar—"install custom firmware", select file INSTONE*.*.*.hex.

2. When the dialog box progress bar is finished and displays "Done!Installed firmware:INSTONE*.*.*.hex", it means successfully updated.

3. Turn off the power of printer and restart it. Then, the firmware program finishes the updating.

Tips

1. Avoid warping of the filament:

In order to make the filament attached to the bottom plate in a preferable way, apply a thin layer of glue on the bottom plate before printing.

2. The TF card attached with the printer has contains some printing models which you can choose for printing; please log in www.instone3d.com to choose more models.

3. You can visit www.instone3d.com to download the slicing software.

4. With the quick updates on product versions, this guide may not reflect all the characteristics of the product, please log in www.instone3d.com for more updates and supports.

Service support

YIWU INSTONE 3D TECHNOLOGY CO.,LTD
Website: www.instone3d.com
Email: support@instone3d.com
Tel: +86-579-89986523
Skype: Instone 3D
Twitter: Instone 3D
Facebook: Instone 3D
Addr: Bld. 2, Area B, No.E21 Xinke Road, Yiwu, Zhejiang, China