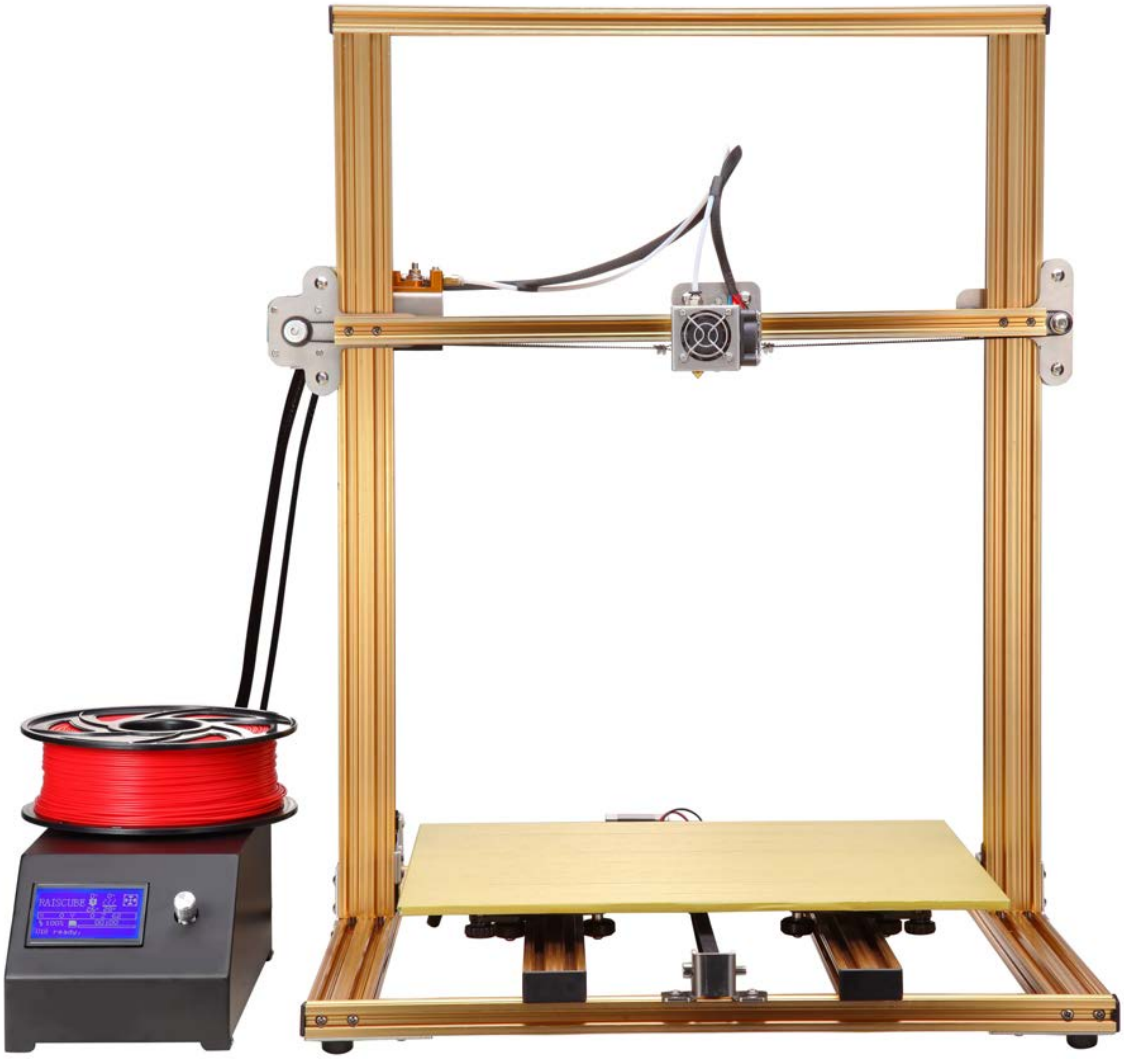
U12/U18 3D Printer Quick Start Guide



Catalogue

Alfawise U12/U18 Specs 1

U12/U18 Packing List 2

Foreword 3

1. Machine parameters 3

2. Flow Diagram for Quick Start Guide 3

3. Important Notice and Attentions 4

4. Remove Packaging 5

4.1 Remove packaging 5

4.2 Check the accessories 5

4.3 Machine placement 5

5. Installation of U12/U18 5

5.1 Step1 Preparation prior to install 5

5.2 Step2 Install printer 6

5.3 Step3 Wire connection 8

Remark 11

6. User Manual 12

6.1 Step1 Leveling the X Axis 14

6.2 Step2 Hotbed Leveling 14

6.3 Step3 Loading Filament 15

6.4 Step4 Preheat Filament 15

6.5 Step5 Feeding Filament 15

6.6 Step6 Printing 17

6.7 Step7 Removing Filament 18

7. Operation: The Four Steps of 3D Printing 18

7.1 Step1 Create or find a 3D file 18

7.2 Step2 Slice the file for 3D printing 19

7.3 Step3 Send the file to the printer. (Off-line Printing or Online Printing) 26

7.4 Step4 Print 28

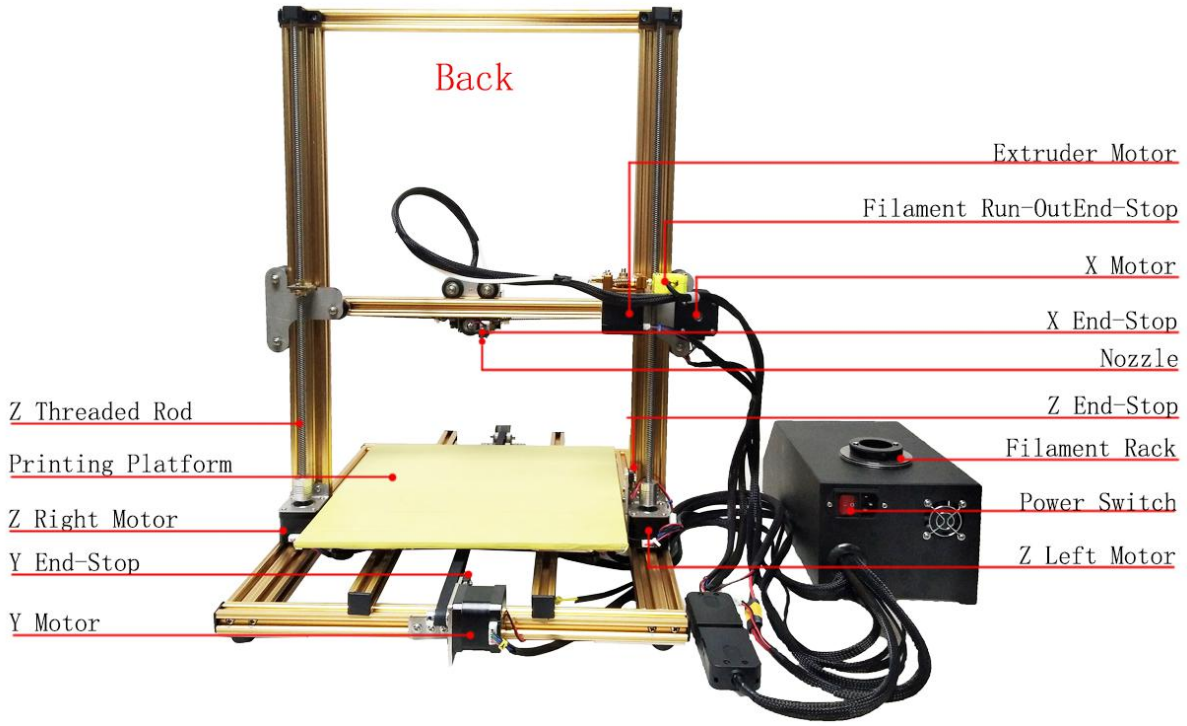
8. Frequently Asked Questions and Solutions 29

8.1 Step-by-Step Troubleshooting 29

8.2 Digital problems: Slicer settings 29

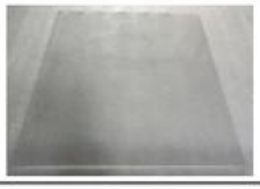
8.3 Mechanical problem: Leveling the build plate 30

8.4 Mechanical problem: Filament is not coming out of the Nozzle 30



RAISCUBE U12/U18 Specs		
Brand	RAUSCUBE	
Model	U12	U18
Extruder Qty	Single	Single
Machine Size	485*560*578(X*Y*Z)	582*690*510(X*Y*Z)
Build Volume	300x300x400mm	400x400x500mm
Package Size	560X640X300mm	705X625X290mm
Machine Weight	11.3kg	13.8kg
Gross Weight	14.2kg	17.3kg
Open Filament System	Optimized for PLA, ABS, Nylon, TPU,etc	
Filament Diameter	1.75mm	
XYZ Accuracy	0.012, 0.012, 0.004mm	
Printing Precision	0.05-0.3mm	
Nozzle	Swappable nozzle (0.4mm by default)	
Feeder type	Geared feeder	
Print Speed	10 ~ 150mm/s	
Power Supply	110/220V, 360W/30A/12V	
Max Temperature of Nozzle	260 °C	
Max Temperature of Hotbed	90°C	60°C
Recommend Temp.	ABS: Nozzle:235 °C Hot Bed:100 °C	
	PLA: Nozzle:215 °C Hot Bed:50 °C	
Operating Software	CURA, Repetier-Host, Simplify 3D	
Software supporting	ROE, Solid-works, UG, 3d Max, Rhino 3D design software, et	
Supported File Types	STL, OBJ, DAE, AMF	
Print File Types	GCODE	
System Compatibility	WIN / XP / MAC / VISTA / LINUX	
Language	English	
File Transfer	SD card (off-line) / USB(on line)	

U12/U18 Pcking List

No.	Items Name	Picture	Qty	NO.	Items Name	Picture	Qty
①	Bottom Part		1	⑧	Shovel *1 Screw Driver *1 Cutting Nipper *1 M5 Hex Wrench*1 M3 Hex Wrench*1 M2.5 Hex Wrench*1 M2 Hex Wrench*1 M1.5 Hex Wrench*1 8mm Hex Wrench *1 14mm Hex Wrench*1		10
②	Frame Part		1	⑨	8G SD Card		1
③	Control Box		1	⑩	Card Reader		1
④	M5*25 Philips Cap Screw*4 M4*8 Button Head Hex Screw*8 T Nut*8 T-Connector*2		22	⑪	USB Cable		1
⑤	405*405 4mm Glass Platform		1	⑫	10m Filament		1
⑥	Masking Tape		1	⑬	Clip		4
⑦	Power Line		1				

3D Printer Quick Start Guide

Foreword

Thanks in advance for choosing our 3D Printer! With the Quick Use Guide (here in after as “Guide”), you could enjoy Test and Printing step by step. Meanwhile, you will also learn the structure principle and its special FDM printing technology deeply, and then you will get great progress on not only the theoretical knowledge but also the manipulative ability.

To make sure players good effect of the printer and ensure the worry-free buying, smooth and enjoyable using, we spare no effect to consider the question and difficulty you will meet with from when you order the printer and make this detail guide to show you the exact settle way. We write this “Guide” elaborately in great detail and we will always accompany with your using on 3D printer. The 3D printer would come to you by assembled 3D printer; you can use it out of box. Please read the “Guide” carefully before you use the printer and then utility our printer in strict accordance with the “Guide” to avoid the any defective or invalid use caused by improper operation. So please keep the Guide for your easy reference.

Special Note:

With our objective of constant update for our printer, we would not make future notice to you if some specifications and models updated. And we apologize for the inconvenience that will lead to you.

1. Machine parameters

Printing technology	FDM	**Nozzle diameter**	0.4mm
Print size	400*400*500mm	**Printing precision**	0.1-0.3mm
Filament	PLA /ABS /HIPS/WOOD	**Printing speed**	40-100mm/s (proposal)
Filament diameter	¢1.75mm	**Working condition**	10-40℃
Number of extruder	1	**Humidity**	20-50％

2. Flow Diagram for Quick Start Guide

Matters and ⟹ Preparatory ⟹ Pre installation Checklist ⟹ Assembly and Calibration ⟹ Printing ⟹ Function Instruction

3.Important Notice and Attentions

For better using experience, please read the following detail instruction carefully before you use the printer:

3.1 Please ensure that you move to the right tap position for input voltage (110V or 220V) of the power supply (as shown in the picture).

Warning: Must switch it to the right voltage in the condition of power off the equipment.

Input Voltage	100~160V	200~260V
Position choice	110V	220V

3.2 Place the machine on a flat table and make sure it will not shake, otherwise it will affect printing.

3.3 Before and after printing model, some supplies will fall out around nozzle due to gravity; you can use tweezers to remove redundant material.

3.4 When printing PLA filament, just use the application's default parameters; when print ABS material, you need to change the print to the corresponding temperature and heat bed temperature.

3.5 When reloading filament, you need to extrude the filament thread a certain distance material and then pull out it. After inserting the new filament thread, push in the new thread a long distance and squeeze out all original materials in extruder. Start printing when it is feed smoothly.

3.6 After several times printing, the wrinkles paper posted on the platform may be broken, you can just remove broken wrinkles paper and stick a new one to get better printing effect.

3.7 When there is filament blocking in the nozzle during printing, you can cut filament thread outside the extruder and use the small screwdriver to clear the feed channel, squeeze out the supplies in the pipes and nozzle.

3.8 After several times printing, the initial height of the platform may be changed and affect printing. If the initial distance between Nozzle and platform is too large, it will cause insufficient adhesion of the model, easy to cock up, or even fall off; the distance is too small and the nozzle cannot discharge properly, easily lead to blocking. At this point you can re-calibrate the height of platform and get the best printing effect. Unless absolutely necessary, do not adjust.

3.9 After printing large models, it will be more convenient to remove the model from the platform if you heat the heated bed to 50 degrees before taking out the model. Do not use violence and make the heating bed platform broken. The temperature of heating bed can be set by entering "Menu" -> "Control" -> "Temperature" -> "Bed" and press the button to start heating.

3.10 Coat some glue or crepe paper tape on the platform before printing big model to increase adhesion between the model and platform, preventing curling edges.

4.Remove Packaging

4.1 Remove packaging

(1) Open the carton, remove EPE protection and take out the machine from carton.
(2) Take out the accessories box.
(3) Remove all fixed tape, fixture and attachments before power up the machine; otherwise it may cause damage to the machine.

4.2 Check the accessories

(1) Power cable: connect the machine and power.
(2) USB data cable: connect 3D printer and the computer.
(3) Allen Key suit: for printer fix and maintenance.
(4) SD card, card reader: Using for offline print. Document and software are In the SD card,please copy all the files to computer for future reference (in case the data lost). When SD card insert into the machine's SD card slot, make sure that the yellow copper pin side is upward.
(5) High temperature wrinkles stickers: stick on the printing platform and make the model better attached to the printing bed.
(6) Shovel: strip down models from the print platform.
(7) Slotted screwdriver: for printer fix and maintenance.

4.3 Machine placement

(1) Connect the power cord to the interface behind the machine.
(2) Put machine on the flat desktop to assure the machine will not shake. If the desktop is not smooth enough, you can pad some paper at the bottom of the machine to ensure the machine does not shake, otherwise it will affect printing.

Pay attention to the safety:
Please do not reach into the machine's inner space.

5. Installation of U12/U18

This manual will offer your assemble instruction including wire connection, operation instruction, and support resources.

Unlike Acrylic kit required to be built piece by piece for over 10 hours to complete, you will find it done to be ready for first print in minutes!

5.1 Step1 Preparation prior to install

① Get all parts including tools and screws out of box, and check all if any damage exists, especially the wires and their connectors.

② Switch PSU to the correct position according to the voltage in your country. E.g: 110V for US, 220V for

UK. (Fig.01)

③ Che k heating bed if it is wobbly. Adjust the tension of heating bed by Allen key and wrench (obtained in tool pack) to fix this if so.

Fig.01

5.2 Step2 Install printer

Items needed: Bottom Part, Frame Part, 4 pcs of M5*30 Philips cap screw, 8 pcs of T Nut, 4mm Glass Platform, 8 pcs of Button Head Hex Screw and 2 pcs of T-Connector.

Insert screws from bottom of “Bottom Part” to reach holes of “Frame Part”.

Please refer to Fig.02

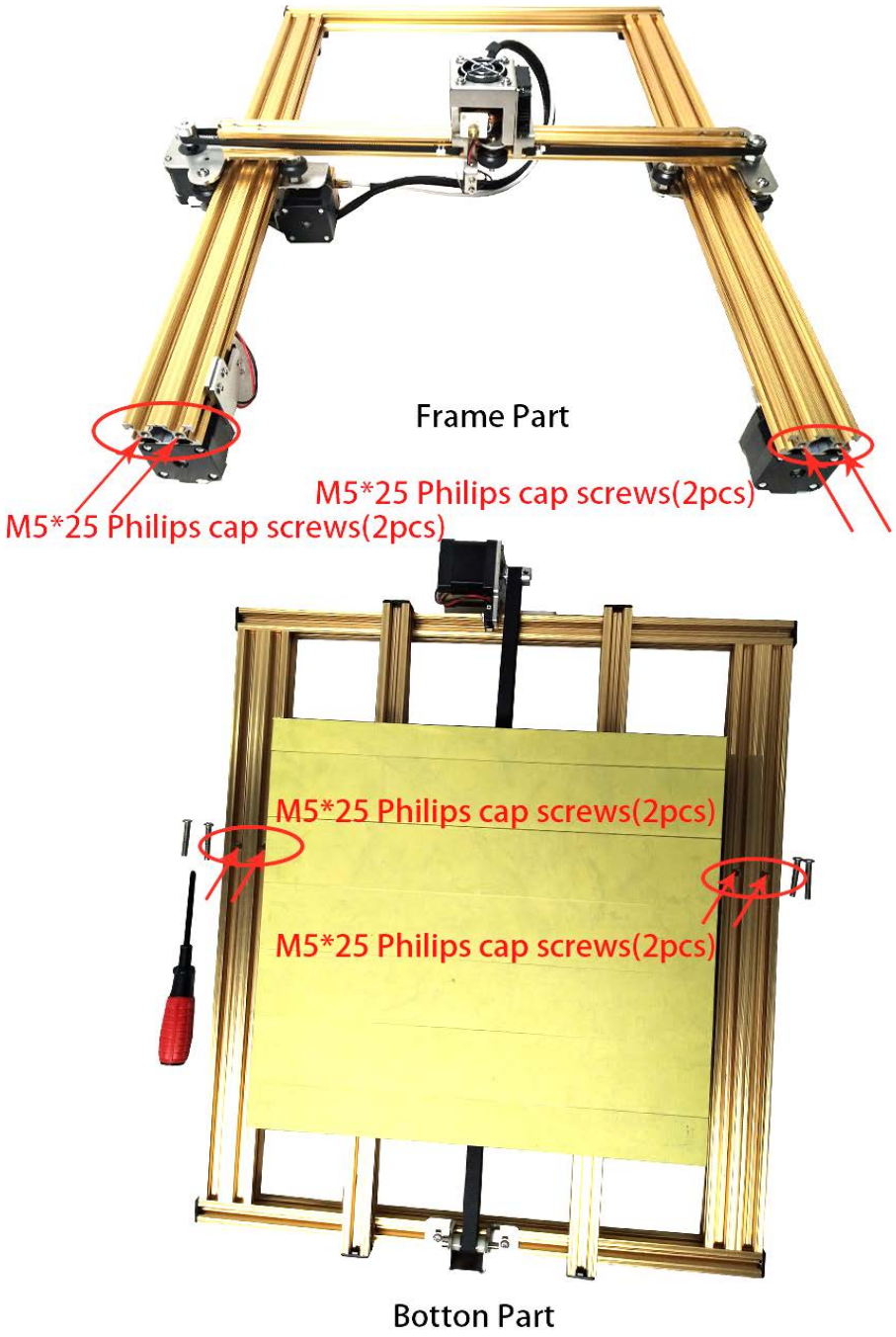


Fig.02

Insert screws from bottom of “Bottom Part” to reach holes of “Frame Part”.

Please refer to Fig.03.

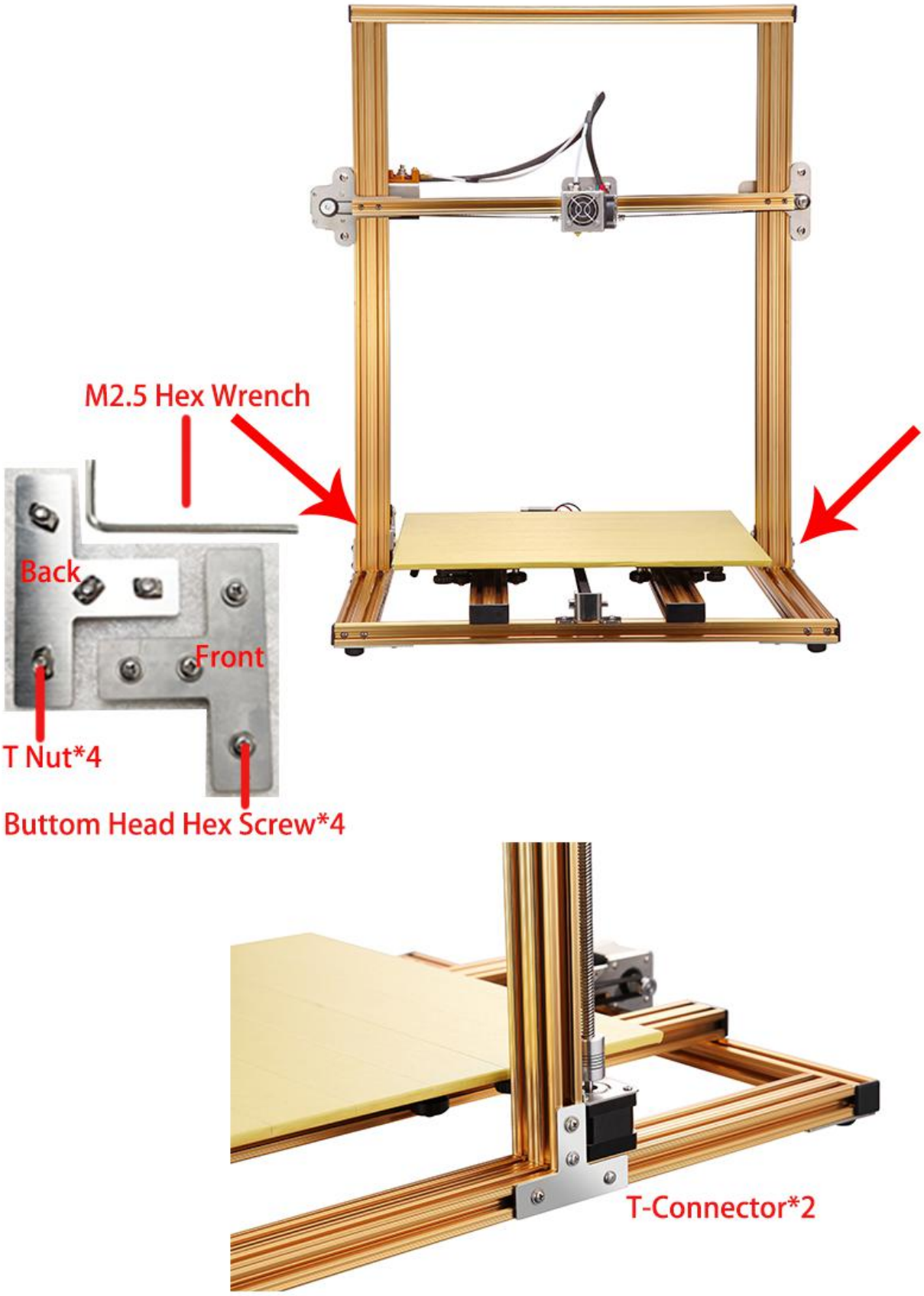


Fig.03

Then, put the 4mm glass platform on the top of the heatbed.

And Fig.04 is what is to be end of this step.

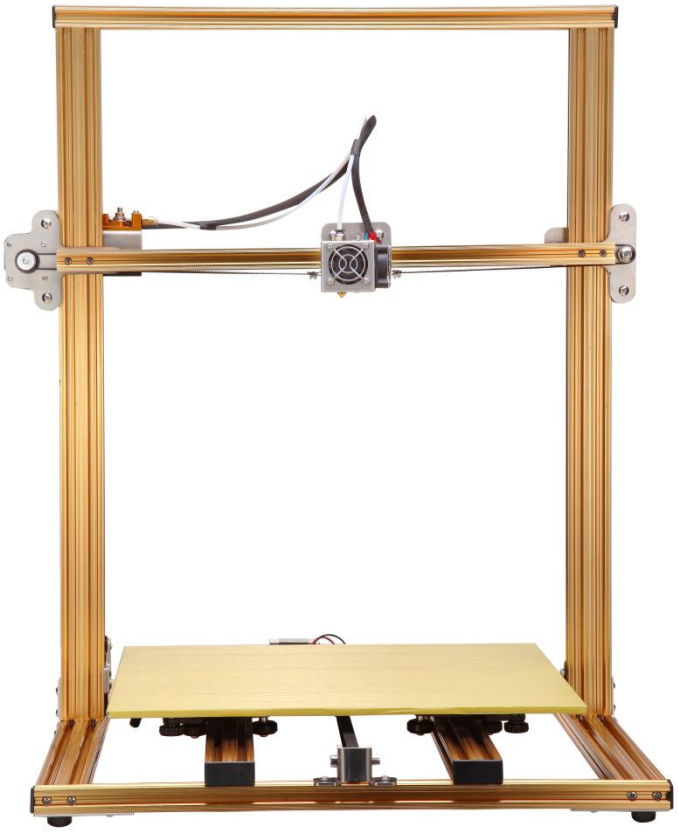


Fig.04 (Build Frame)

5.3 Step3 Wire connection

Items needed: Control box and Build Frame.

Make “Control Box” circuits and “Build Frame” (Fig.04) circuits connected well. Please note that there are labels of each wire, please refer to Fig.05, and connector at end of each wire.

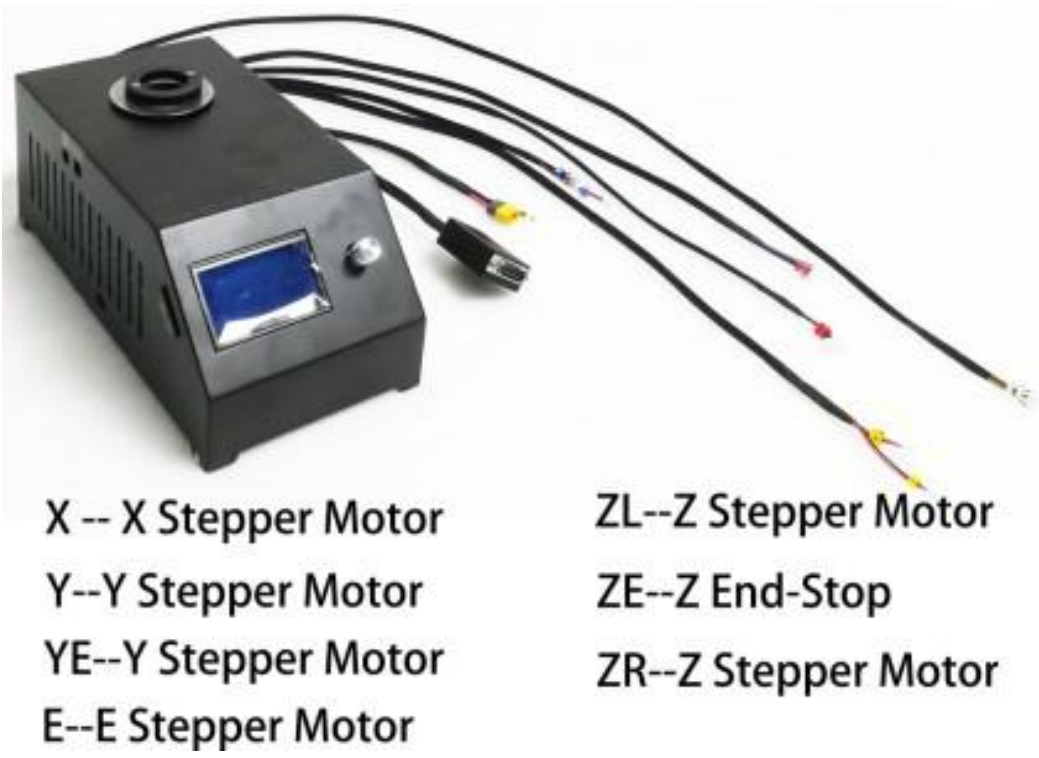


Fig.05

Plug a connector to the other matching connector, please refer to Fig.06.

Fig.06

After done this, we get what in Fig.07.

Fig.07

Also please refer to Fig.08 when connecting

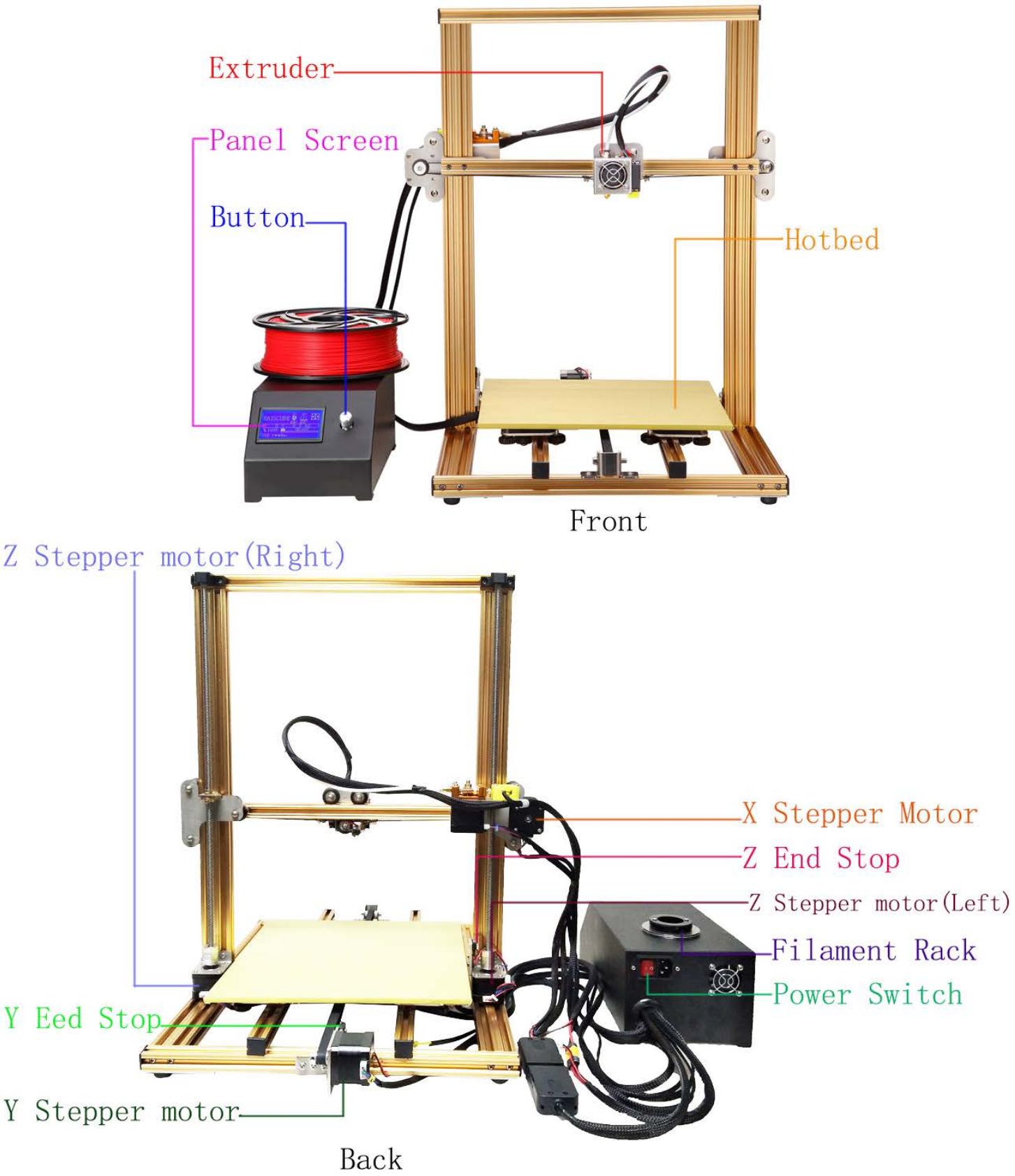


Fig.08 (Overall Details)

Remark:

Tension of X and Y axis belt.

A proper tension of belt is required ahead of printing. Tension is well adjusted in factory, so generally speaking, it shall be ok. But may loose, check tension check is required. If too loose, need to fix it.

Load filament system.

Place filament on to spool holder on top of control box. And feed filament into extruder. Plug power line to socket, and press switch to turn on printer. Will see it on screen. And build is done. (Fig.09)

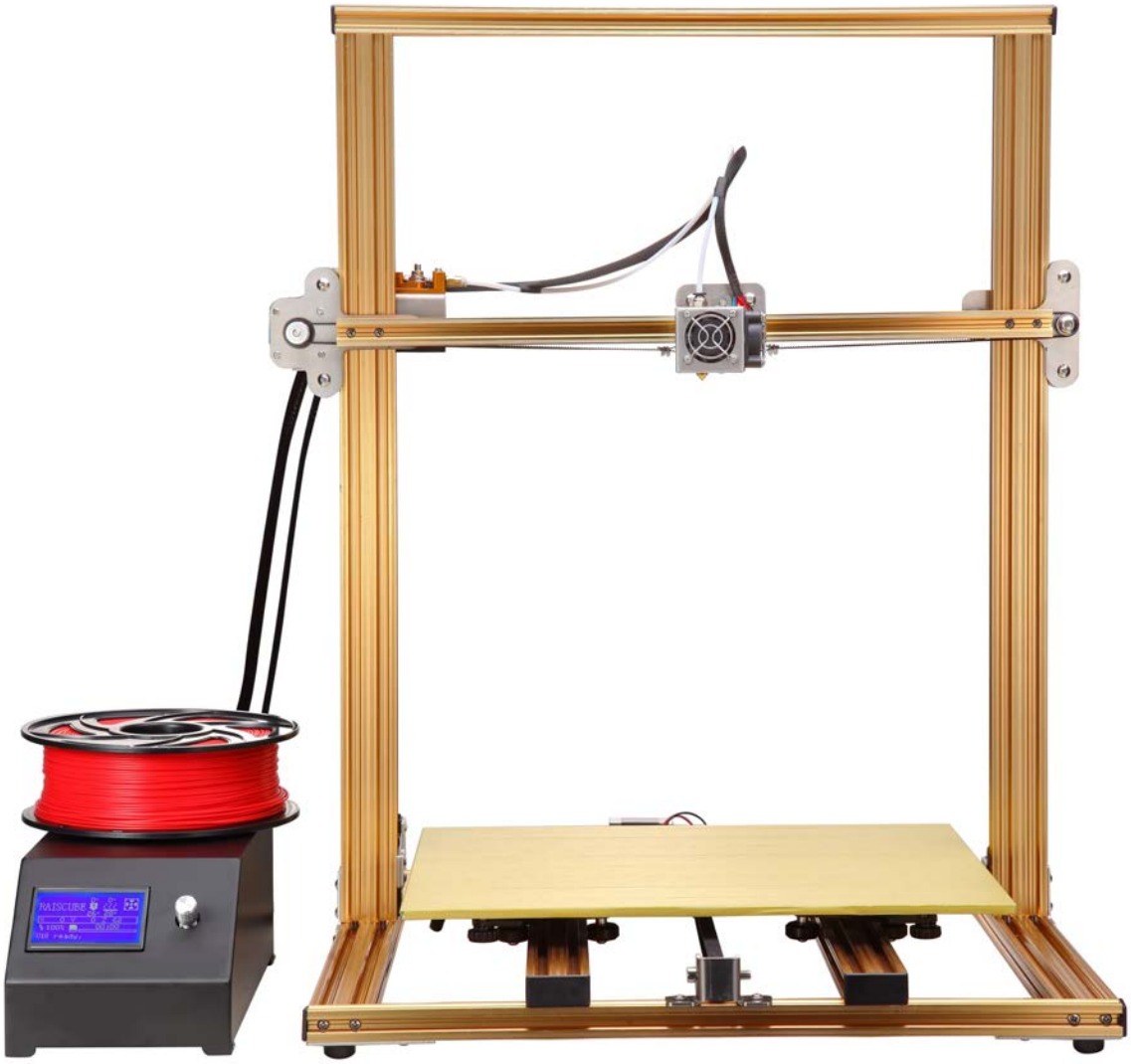


Fig.09

6. User Manual

Screen Button
Press Down: Confirm/ Yes
Turn Left: Return/ Plus
Turn Right: Next/ Minus

Panel Screen

Info Screen

Prepare

- Main
- Auto home
- Preheat PLA
- Preheat ABS
- move Z up
- Move axis
- Disable steppers
- Level bed
- Z home

Control

- Main
- Temperature
- Motion
- Store memory
- Load memory
- Restore failsafe

Print from SD

Unload SD card

Screen Information.

① Select “**Control** ” → “ **Temperature** ” → “ **Bed** ”.

Turn the bed temperature to 55°C. Then navigate back to the Info Screen. This will heat the bed to actual printing conditions, making the leveling more accurate.

② Select “**Prepare** ” → “ **Preheat PLA**”.

This will heat the nozzle and melt any hardened plastic that might be on the nozzle and affect the leveling process.

③ Select “**Prepare** ” → “ **Auto home**”.

This will move the nozzle to the home position at the front left corner of the build plate.

④ Select “**Prepare** ” → “ **Disable steppers**”.

This will allow you to move the nozzle assembly (X axis) and build plate (Y axis) by hand.

Pro Tip: The Z axis stepper is also disabled, and you don’t want it to move at all.
So you should handle the printer gently during the leveling process. Keep it flat and move the nozzle assembly carefully.

⑤ Wait for the print bed temperature to reach 55°C and the nozzle temperature to reach 200°C. The Info screen displays this information below the nozzle and bed icons.

⑥ Slide a piece of paper between the nozzle and the build plate.

⑦ Adjust each of the four thumbscrews under the bed until the piece of paper sides, with just a tiny bit of drag, in all locations on the build plate

⑧ Select “**Prepare** ” → “ **Auto home**”, then **Prepare** ” → “ **Disable steppers**”, and recheck in several places. This is to make sure the Z-axis lead screw didn’t rotate during the leveling process.

⑨ During printing, there will be three selections coming out: “**Pause print** ”, ”**Stop print**”, ”**Resume print**”.

Select “**Pause print**”, this will be stop printing momentarily when need to feed/change another filament in the halfway of print process. After finishing feeding/changing, select ”**Resume print**”, this will continue to print unfinished process.

Select “**Stop print**”, this will abandon the current print task. This will start another print process.
Pro Tip: Can be turn the temperature of hotbed into 0°C for hotbed protection when making a long-time print model in the halfway of process.

⑩ **Resume Print Function：**

Attention: If you ordered the 3D printer with function of Resume Print, accidentally power off during printing,you can use this function to recover the printing model and continue to print.During printing, the 3D printer can automatically record the break point,when the new boot is working, there will be option of “**Continue Print** !” on the LCD screen, then you can continue to print start from the break point last time the power off.

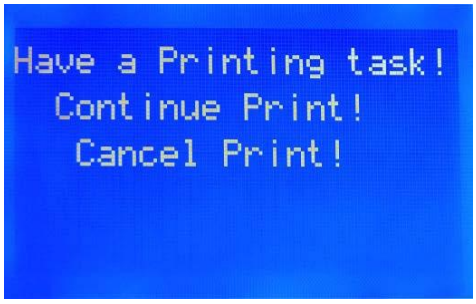


Fig.10

6.1 Step1 Leveling the X Axis

Get a regular rectangle object, to keep a same distance between top surface of Z-Axis Motor Fixed Plate and bottom face of Z-Axis Link Block (Left&Right). If not, turn a side of “Flexible Coupling” to keep two sides same to make it level.
Please refer to Fig.11.

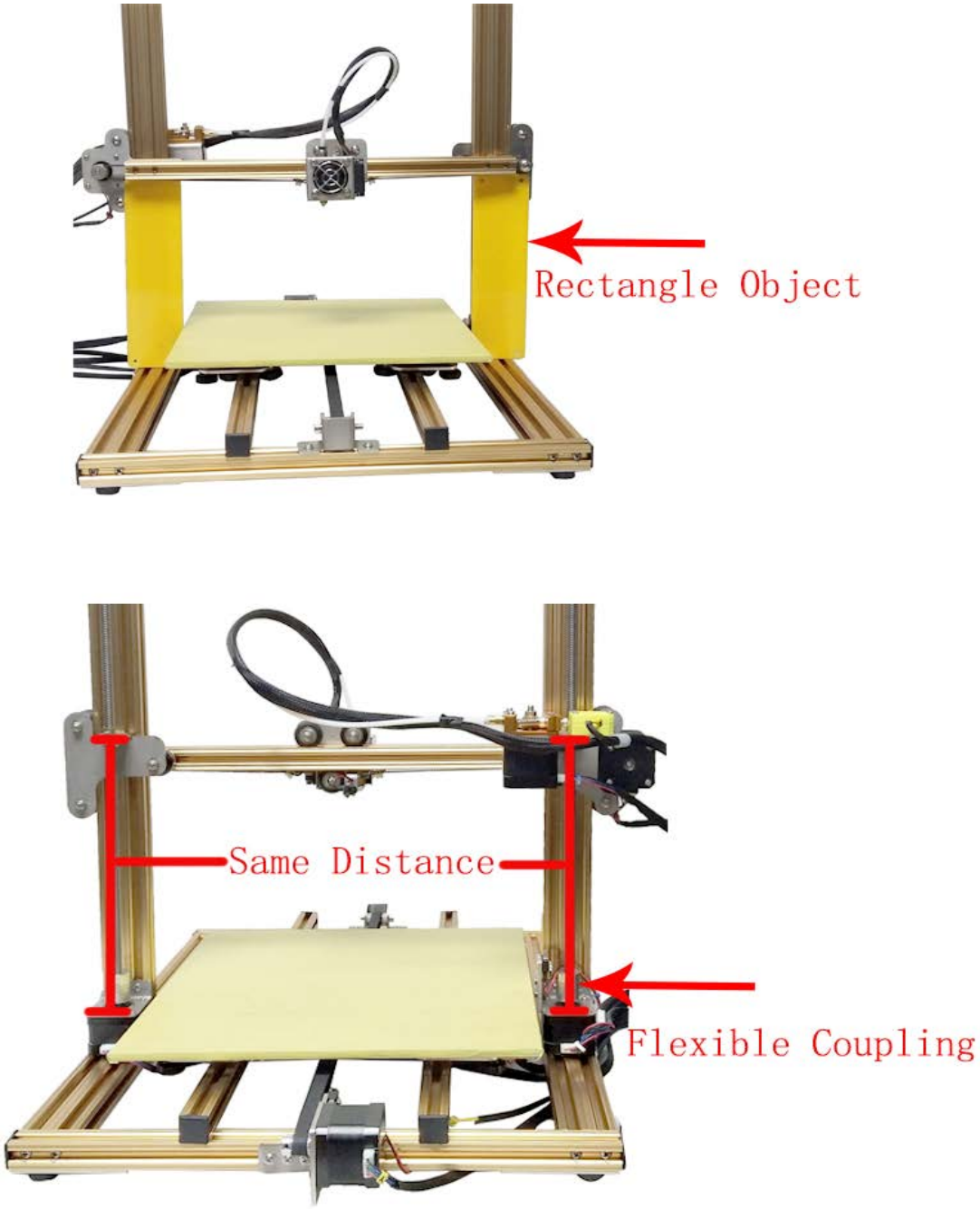


Fig.11

6.2 Step2 Hotbed Leveling.

① Panel Control, press control button, “**Prepare**” → “**Auto Home**”, to move nozzle to origin point. Check if nozzle top almost touches hotbed surface. If not, find the long screw behind Left Z-Axis Link Block, and adjust it to be.

② Moving the Extruder towards the one of corners of hotbed. Panel Control, “**Prepare**” → “**Z Home**”, then

Extruder moves towards hotbed. It should be only allow a piece of A4 paper to pass through. If not, make clockwise (or counterclockwise) rotation of the Wing Nuts beneath hotbed surface. Please refer to Fig.12.

③ Manually move hotbed or slide extruder to other vertexes of hotbed to make four corners of hotbed level evenly.

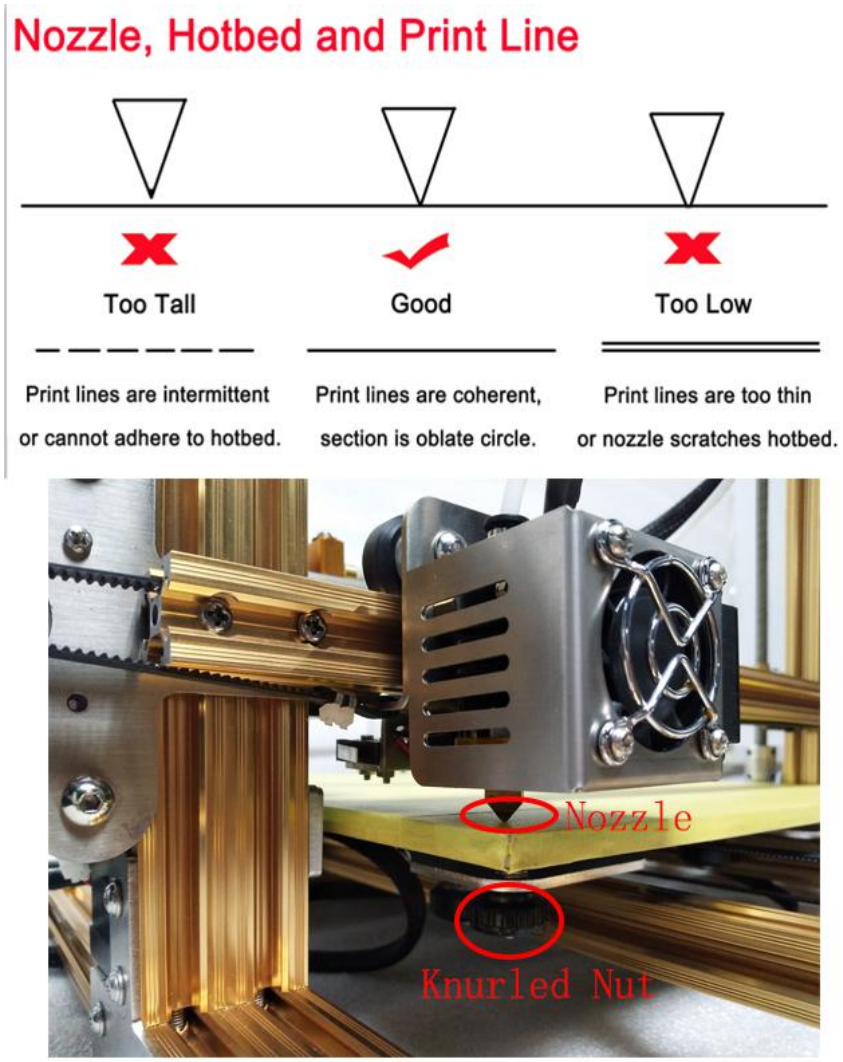


Fig.12

6.3 Step3 Loading Filament.

① Load a spool of filament onto the filament rack.

② Cut off the used or bent end of the filament at an angle and it will be easier to feed into the Extruder.

6.4 Step4 Preheat Filament.

Panel Control, “**Prepare**” → “**Preheat PLA**”.

6.5 Step5 Feeding Filament.

① When the temperature of extruder reaches about 180℃, feed the end of the filament into the extruder with one hand as you squeeze the squeeze head release lever with your other hand, to put the filament into the inlet port.

② Panel Control, “**Prepare**” → “**Move Axis**”→ “**Move Extruder**”, turning the button to right, to make the filament pass through the Teflon hose, and out of the outlet port.

Please refer to Fig.13.

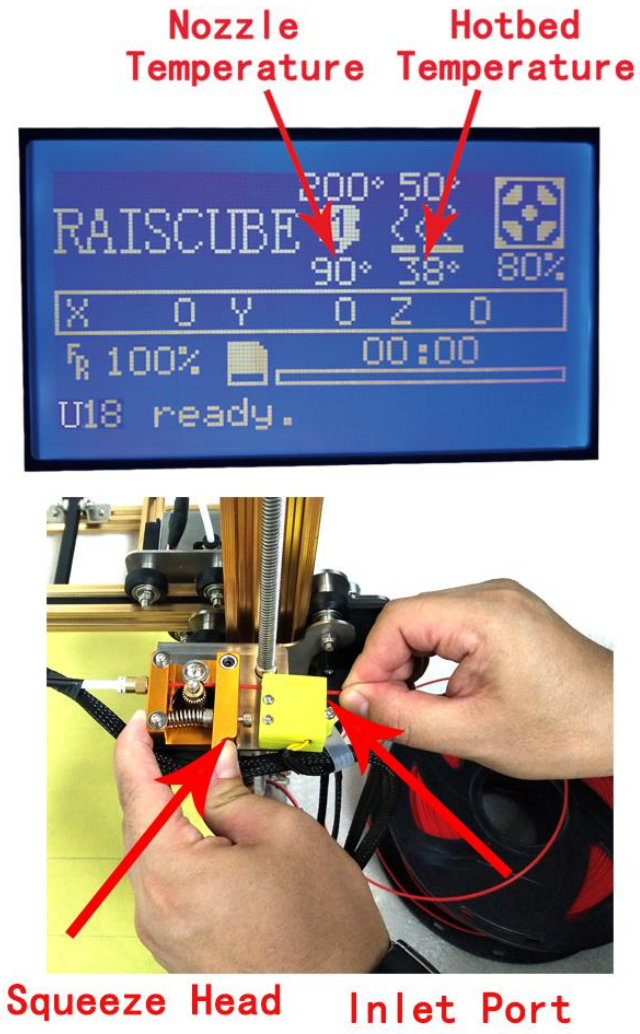


Fig.13

Remark:

① When the printer is fully heated, check to make sure the filament is fully loaded. This also removes all of the previous material left in the printer from other prints. To do this, manually push filament through the heated nozzle until the new color begins coming out of the nozzle.

② Never leave the printer preheated with material inside of it for long periods of time. This will cause the filament to bake inside of the nozzle assembly and can cause it to clog. And please do not touch the nozzle with your hands after heating.

After done this, we get what in Fig.14.

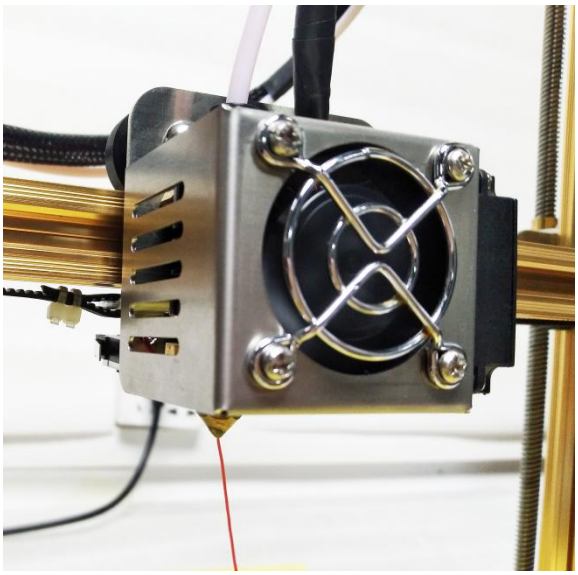


Fig.14

6.6 Step6 Printing.

In this step, clip the edge of hotbed with clips for firmness. Stick some masking tapes on the surface of hotbed if you like. Also be okay to prints directly on the surface of glass.

Panel Control, “**Print from SD**”, wait for the extruder moving and start to print.

Remark:

You may need to make fine adjustments to the bed level when you start printing.The first layer of the print will show whether the distance between the nozzle and build plate is correct. Refer to the following graphic.

① If the bed is too low, the extruded plastic may not stick to the build plate.

② Parallel lines of filament on the first layer will look rough, with high tracks and low valleys that are not stuck to the build plate.

③ If the bed is too high, the extruded plastic will squeeze out the sides of the nozzle. The nozzle will plow through the plastic and leave a first layer that is too thin.

④ You can try to carefully adjust the thumbscrews during the first layer of the build while the plate is moving until the distance between the build plate and the nozzle is producing smooth extruded lines. Be careful of the moving parts to not get pinched.

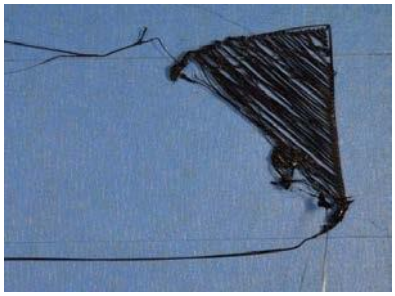
⑤ After you have fine-tuned the bed level during the first layer, you may want to stop the build, clear the build plate, and restart the build.

Attention:

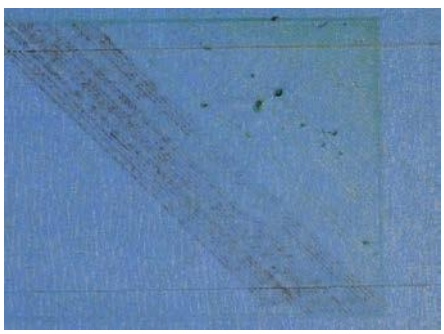
Please pay attention to the first layer filament size:

If the filament string is obviously smaller than 0.4mm (the standard nozzle size is 0.4mm),or it can’t stick to the platform, or too big distance between two strings, these all means nozzle is too far from the heated bed, the printing item will be easy to warped, so we need to adjust the screws as following photos, to make the heated bed platform rise up to make sure the nozzle and heated bed in perfect distance.

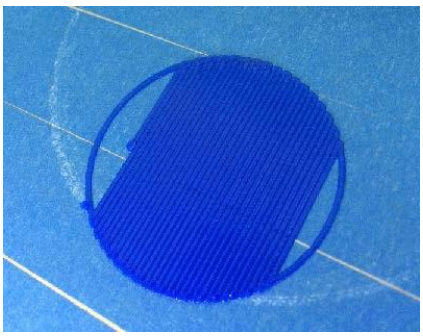
On the contrary, if the filament comes out from the nozzle is too thick, or two lines are lapped, it means the nozzle and heated bed is too close, after long time printing, the filament will not smoothing or clog from the nozzle, then we need to adjust the screws, to down the heated bed, until the normal size filament comes out from the nozzle.



Nozzle and heated bed too far in distance



Nozzle and heated bed too close in distance



Nozzle and heated bed perfect in distance

6.7 Step7 Removing Filament.

To remove the filament cleanly, you should do a “soft pull.” This involves heating the nozzle to 100°C, then pulling the filament out of the printer.

① Panel Control, “**Prepare**” → “**Preheat PLA**”, and wait until the nozzle temperature reaches 100°C. (You can also select “**Control**” → “ **Temperature** ” → “ **Nozzle**”, spin the temperature to 100°C, and then navigate back to the Info Screen.)

② Wait for the nozzle to reach the set temperature.

③ When the nozzle temperature is 100°C, squeeze the extruder release lever with one hand and pull the filament out of the extruder with your other hand in one fluid motion.

④ Cut the end of the filament to remove the partly melted plastic.

Remark:

Feed the end of the filament into the hole on the side of the spool. This will insure it doesn’t get tangled and cause a build to fail the next time you use it.

7 Operation: The Four Steps of 3D Printing

3D printing involves four steps:

Step 1 Create or find a 3D file
Step 2 Slice the file for 3D printing
Step 3 Send the file to the printer
Step 4 Print!

Note:

(1).We suggest new user to use Cura for Slicing.

(2).The data you will slice depend on the item you need to print.

(3).Before you start printing, please make sure the parts of printer work well and the distance between nozzle and aluminum plate is about 0.1mm (A4 Paper thickness).
Ps: Most data was default, you need to change them depends on the object you want to print.

7.1 Step1 Create or find a 3D file

① You can create a 3D model—a digital geometry—using any CAD(computer-aided design) software

program. We recommend free programs, such as TinkerCAD, OnShape, and Blender. Another is SketchUp, which has free licenses available for schools. These programs take some time to learn, and can be challenging. But they are also rewarding, especially when you see your own creation being printed on your 3D printer!

Every CAD program allows users to save or export digital files for 3Dprinting. You want to save your file as an STL file ("filename.stl"). The STL file is the standard file format for 3D printing. In some programs, it's as easy as the Save command. In others, you need to select the File > Export option, and choose the STL format. OBJ files can also be sliced and 3D printed.

② You can find 3D models in many online libraries of digital content. Thingiverse.com, GrabCAD.com, and Instructables.com are examples, and many other sites exist.

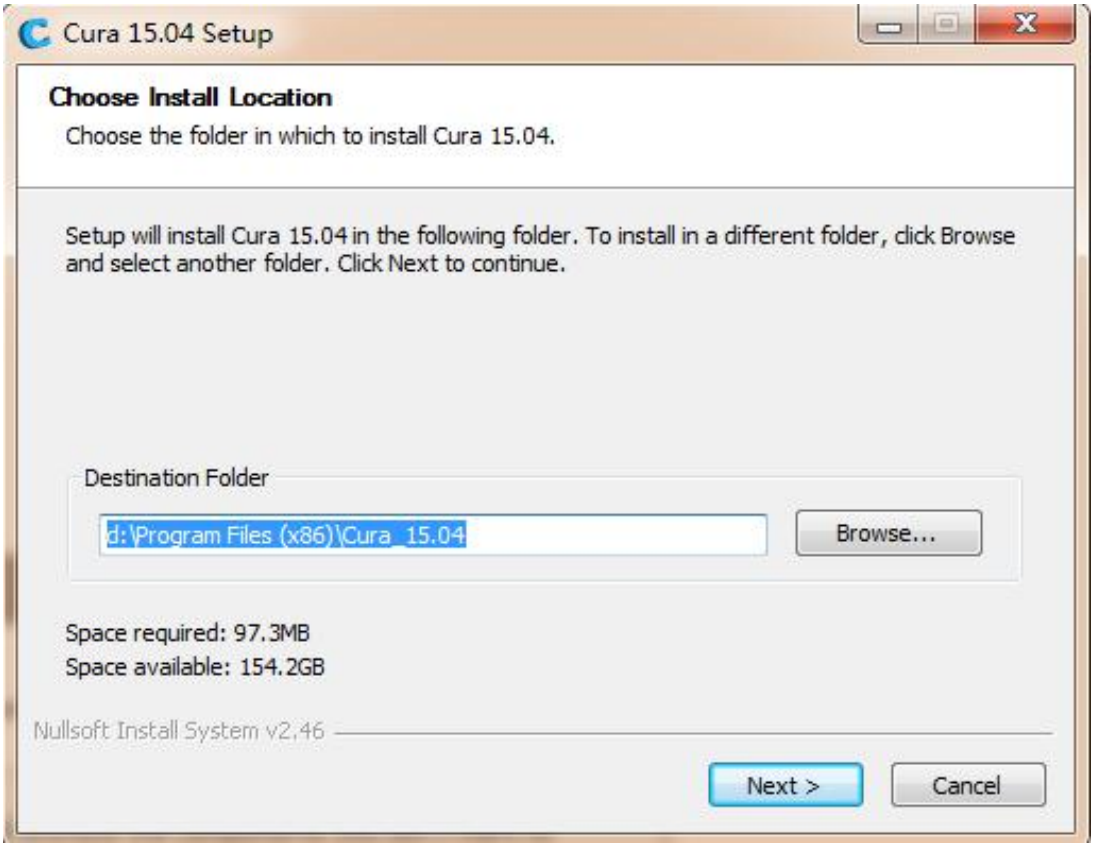
7.2 Step2 Slice the file for 3D printing

Slicing is the process of converting your 3D model into the layered print language your printer can understand, called G-Code. We recommend using free, open-source slicer programs. Our favorite slicers are Cura and Repetier Host. Both are free and easy to use. We've included Cura and its setup .ini file on your printer's SD card because it's easier for beginners. We recommend that you install Cura on a PC or Mac and use it to do your slicing.

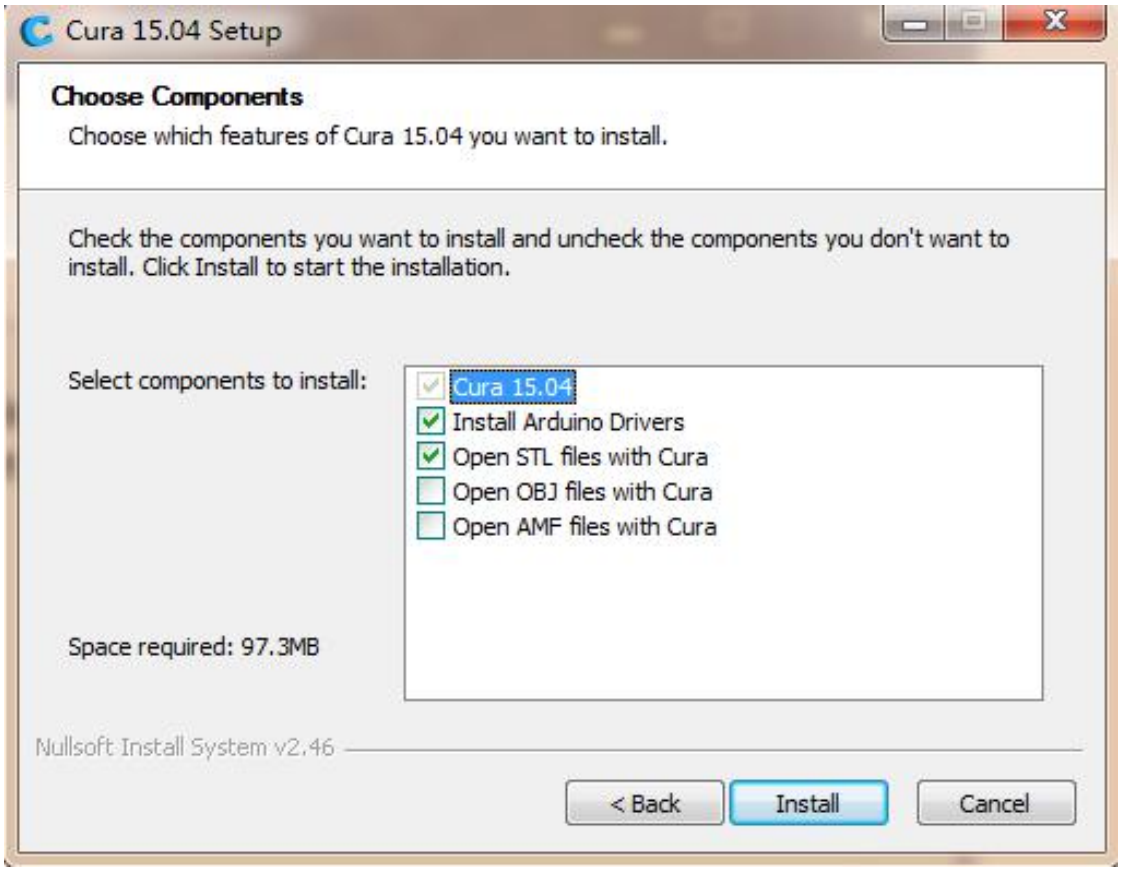
There are a lot of slicers that will create G-Code automatically for your printer.
All you have to do is input the correct settings for your printer (using the files we included), import your 3D model, and click slice! It's that easy.

① Install the Cura program on a PC or Mac. (**How to use slicing software:** Please open the "SD-Card:\Software\Cura 15.04" and click the software "Cura15.04" to slice the printing file as per the use manual shown on "How to use Cura15.04".)

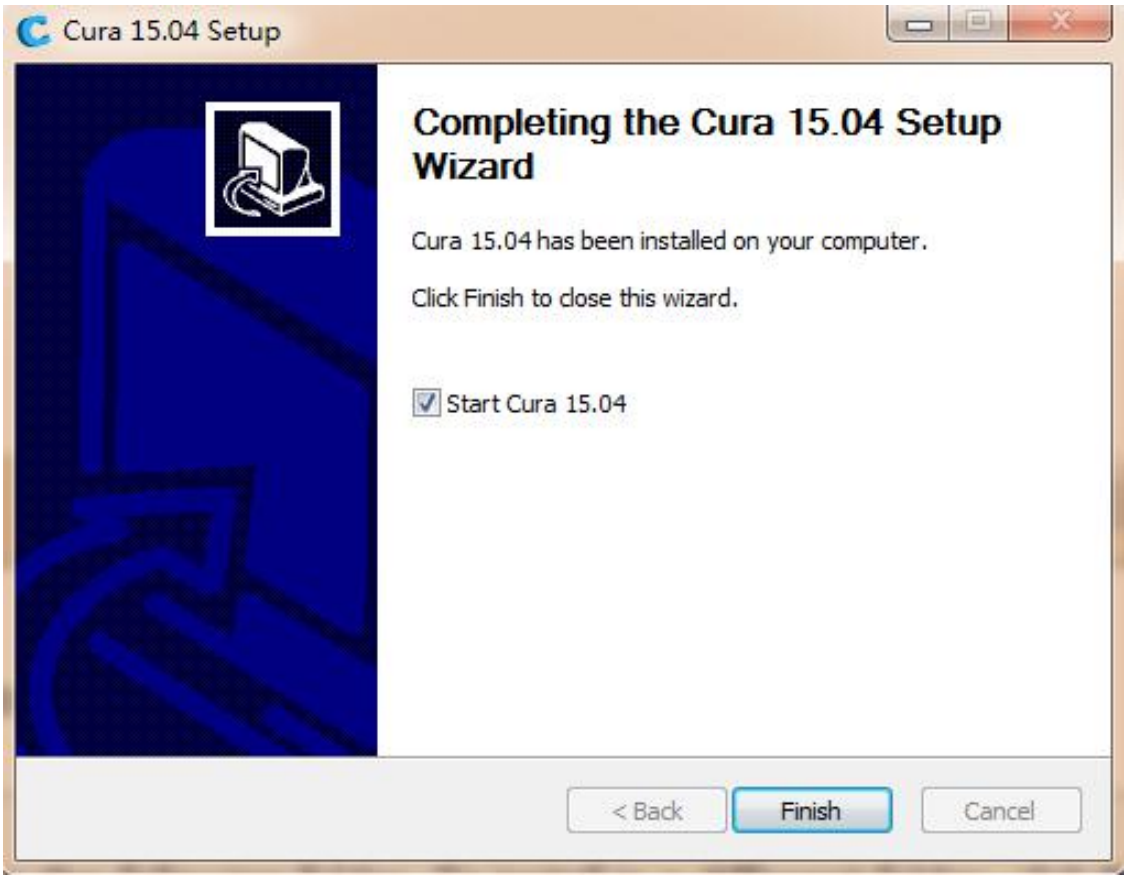
Step1:



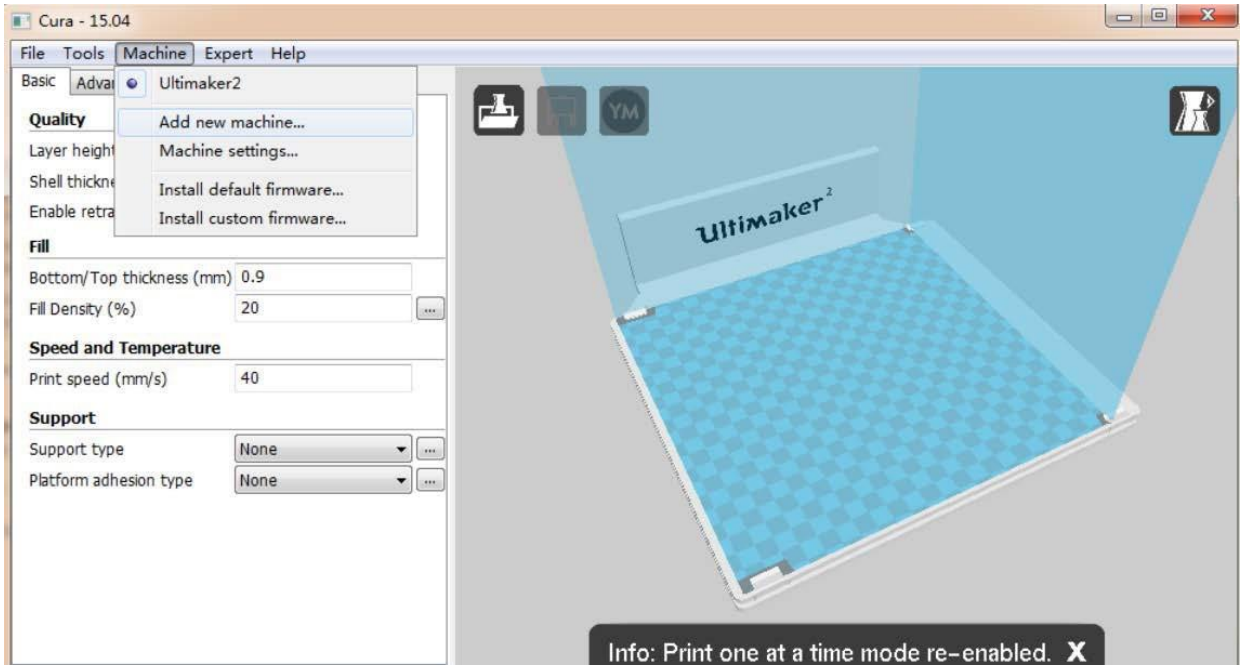
Step2:


Cura 15.04 Setup
Choose Components
Choose which features of Cura 15.04 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 install:
Cura 15.04
Install Arduino Drivers
Open STL files with Cura
Open OBJ files with Cura
Open AMF files with Cura
Space required: 97.3MB
Nullsoft Install System v2.46
< Back
Install
Cancel

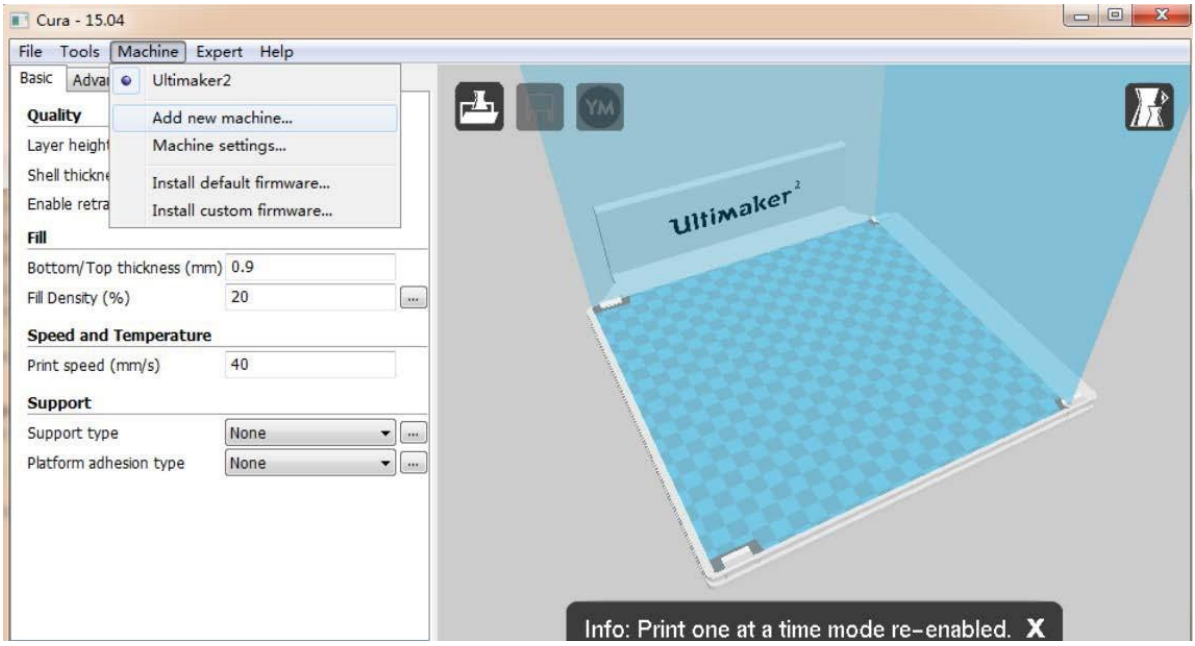
Step3:


Cura 15.04 Setup
Completing the Cura 15.04 Setup Wizard
Cura 15.04 has been installed on your computer.
Click Finish to close this wizard.
Start Cura 15.04
< Back
Finish
Cancel

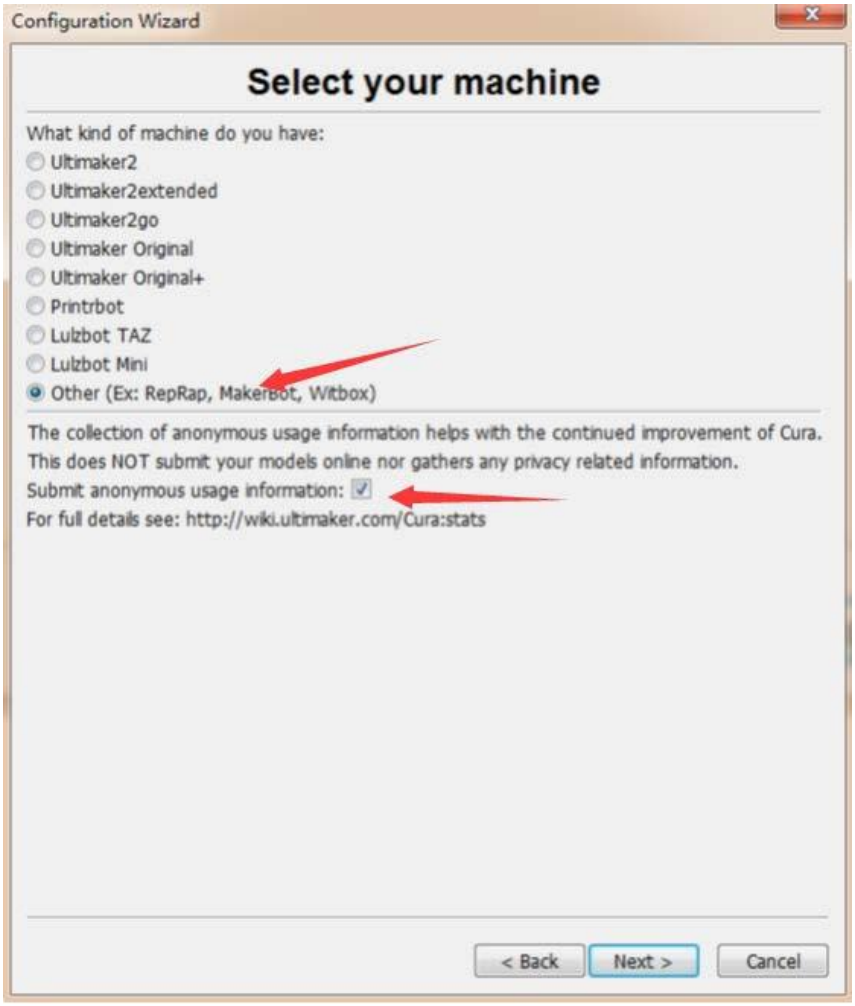
Step4:

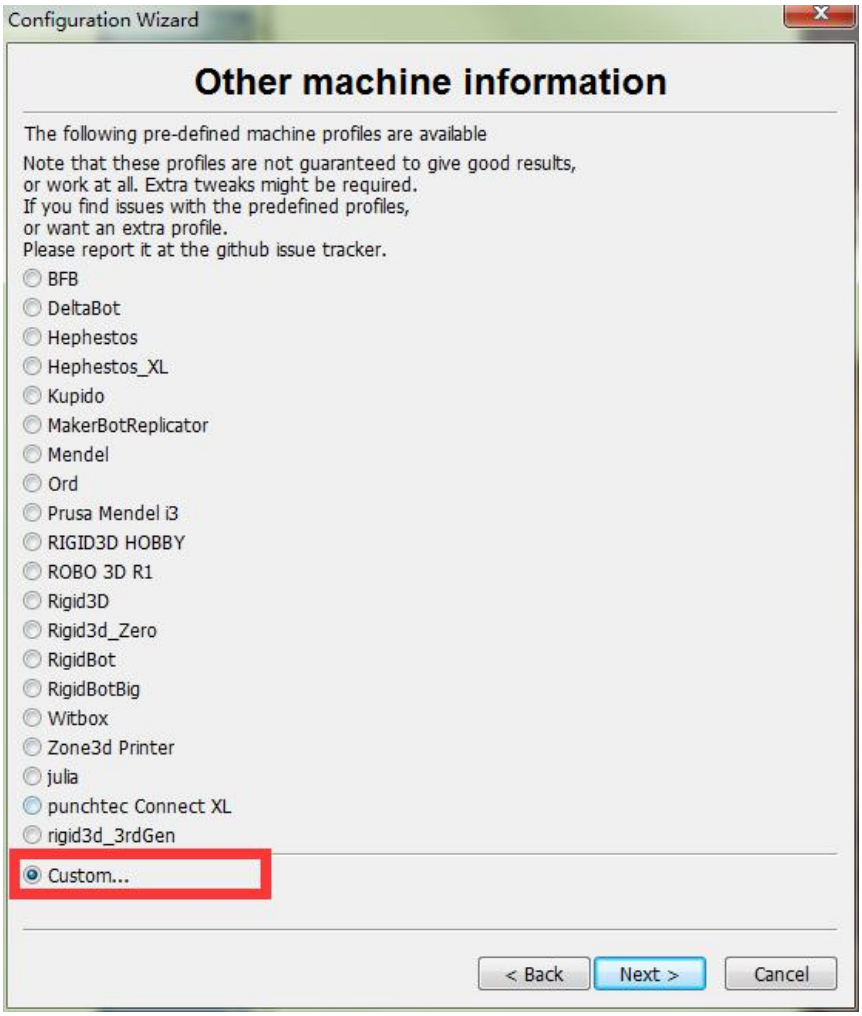


② Open the Cura application. You'll be given several options. In the Configuration Wizard, choose the RepRap Machine and Mendel in other machine settings (RepRap (Marlin/Sprinter) G-Code Flavor, if it asks).

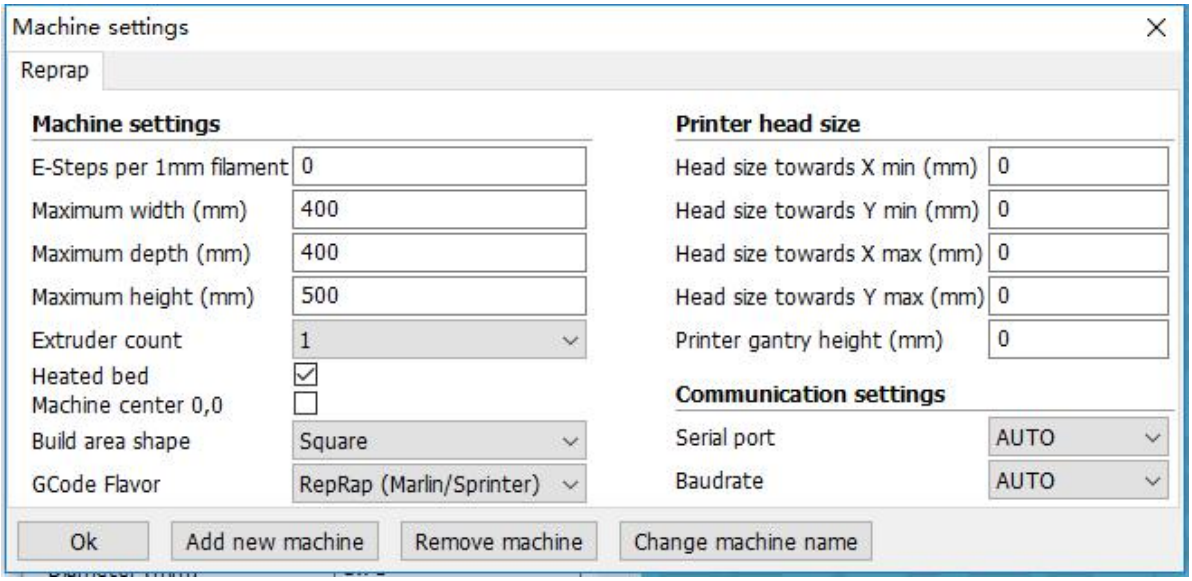



Configuration Wizard
Add new machine wizard
This wizard will help you in setting up Cura for your machine.
< Back
Next >
Cancel

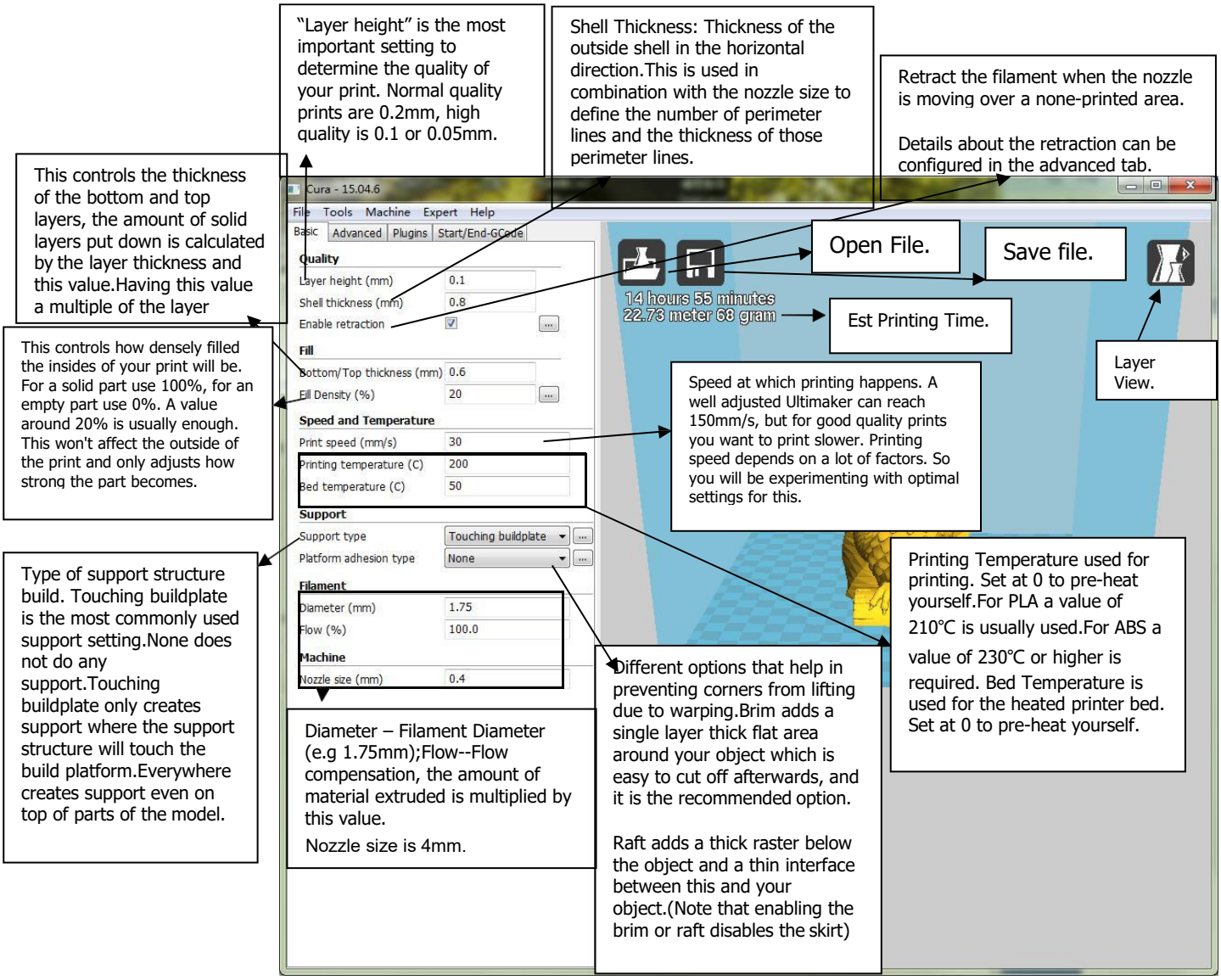

Configuration Wizard
Select your machine
What kind of machine do you have:
Ultimaker2
Ultimaker2extended
Ultimaker2go
Ultimaker Original
Ultimaker Original+
Printrbot
Lulzbot TAZ
Lulzbot Mini
Other (Ex: RepRap, Makerbot, Witbox)
The collection of anonymous usage information helps with the continued improvement of Cura.
This does NOT submit your models online nor gathers any privacy related information.
Submit anonymous usage information:
For full details see: http://wiki.ultimaker.com/Cura:stats
< Back
Next >
Cancel



③ Once you add the new machine, click Machine > Machine settings and change the machine settings to: Max width 400, Max depth 400, Max height 500, extruder count 1, and check Heated bed. Click“OK.”



④ Load the model file in Cura. Rotate the file to the best orientation for printing. You may want to scale the file at this point as well.

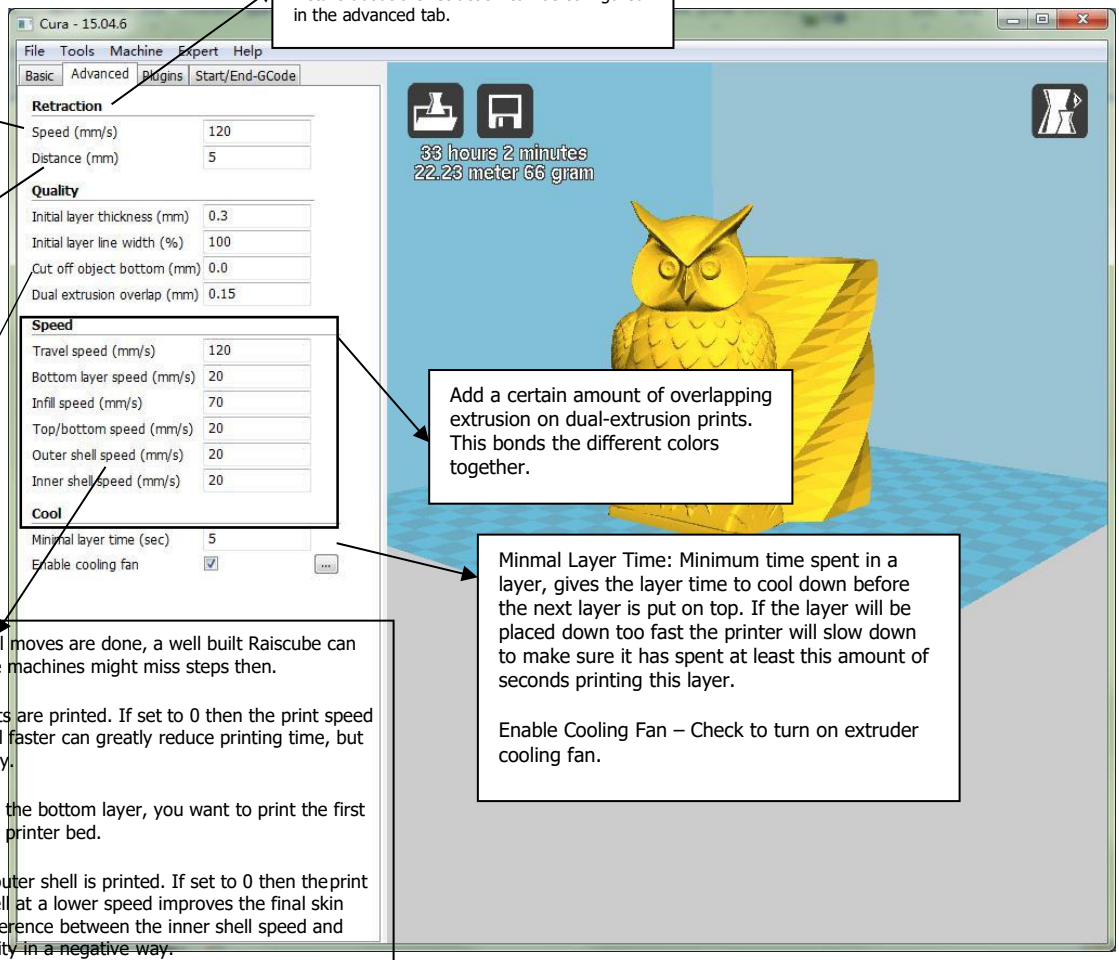

"Layer height" is the most important setting to determine the quality of your print. Normal quality prints are 0.2mm, high quality is 0.1 or 0.05mm.
Shell Thickness: Thickness of the outside shell in the horizontal direction.This is used in combination with the nozzle size to define the number of perimeter lines and the thickness of those perimeter lines.
Retract the filament when the nozzle is moving over a none-printed area.
Details about the retraction can be configured in the advanced tab.
This controls the thickness of the bottom and top layers, the amount of solid layers put down is calculated by the layer thickness and this value.Having this value a multiple of the layer
This controls how densely filled the insides of your print will be. For a solid part use 100%, for an empty part use 0%. A value around 20% is usually enough. This won't affect the outside of the print and only adjusts how strong the part becomes.
Type of support structure build. Touching buildplate is the most commonly used support setting.None does not do any support.Touching buildplate only creates support where the support structure will touch the build platform.Everywhere creates support even on top of parts of the model.
Diameter – Filament Diameter (e.g 1.75mm);Flow--Flow compensation, the amount of material extruded is multiplied by this value.
Nozzle size is 4mm.
Different options that help in preventing corners from lifting due to warping.Brim adds a single layer thick flat area around your object which is easy to cut off afterwards, and it is the recommended option.
Raft adds a thick raster below the object and a thin interface between this and your object.(Note that enabling the brim or raft disables the skirt)
Printing Temperature used for printing. Set at 0 to pre-heat yourself.For PLA a value of 210°C is usually used.For ABS a value of 230°C or higher is required. Bed Temperature is used for the heated printer bed. Set at 0 to pre-heat yourself.
Speed at which printing happens. A well adjusted Ultimaker can reach 150mm/s, but for good quality prints you want to print slower. Printing speed depends on a lot of factors. So you will be experimenting with optimal settings for this.
Open File.
Save file.
Est Printing Time.
Layer View.
Cura - 15.04.6
File Tools Machine Expert Help
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) 30
Printing temperature (C) 200
Bed temperature (C) 50
Support
Support type Touching buildplate
Platform adhesion type None
Filament
Diameter (mm) 1.75
Flow (%) 100.0
Machine
Nozzle size (mm) 0.4
14 hours 55 minutes
22.73 meter 68 gram

Retraction: Retract the filament when the nozzle is moving over a none-printed area. Details about the retraction can be configured in the advanced tab.

Speed: Speed at which the filament is retracted, a higher retraction speed works better. But a very high retraction speed can lead to filament grinding. 120 is a common value for it.

Distance: Amount of retraction, set at 0 for no retraction at all. A value of 4.5mm seems to generate good results. Normally set to be 3~5mm.

Cut off object bottom: Sinks the object into the platform, this can be used for objects that do not have a flat bottom and thus create a too small first layer. Also can be used to printing



Add a certain amount of overlapping extrusion on dual-extrusion prints. This bonds the different colors together.

Minmal Layer Time: Minimum time spent in a layer, gives the layer time to cool down before the next layer is put on top. If the layer will be placed down too fast the printer will slow down to make sure it has spent at least this amount of seconds printing this layer.

Enable Cooling Fan – Check to turn on extruder cooling fan.

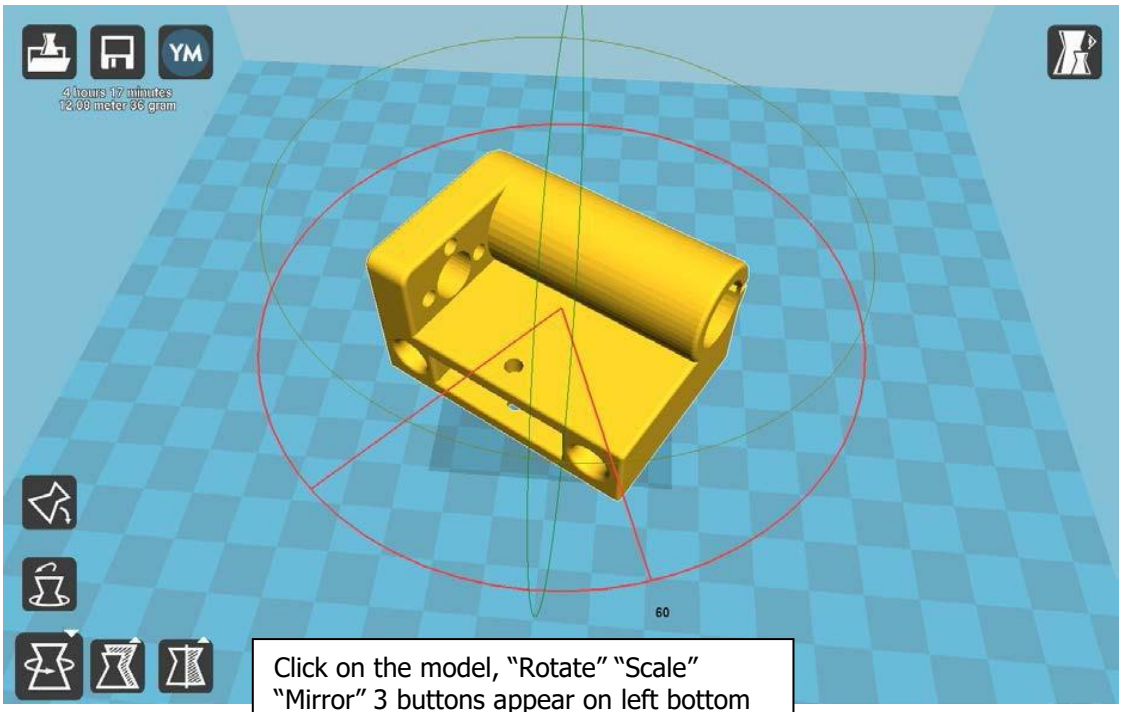
TravelSpeed -- Speed at which travel moves are done, a well built Raiscube can reach speeds of 250mm/s. But some machines might miss steps then.

InfillSpeed: Speed at which infill parts are printed. If set to 0 then the print speed is used for the infill. Printing the infill faster can greatly reduce printing time, but this can negatively affect print quality.

Bottom Layer Speed: Print speed for the bottom layer, you want to print the first layer slower so it sticks better to the printer bed.

Outer Shell Speed: Speed at which outer shell is printed. If set to 0 then the print speed is used. Printing the outer shell at a lower speed improves the final skin quality. However, having a large difference between the inner shell speed and the outer shell speed will effect quality in a negative way.

Inner Shell Speed: Speed at which inner shells are printed. If set to 0 then the print speed is used. Printing the inner shell faster then the outer shell will reduce printing time. It is good to set this somewhere in between the outer shell speed and the infill/printing speed.



Click on the model, "Rotate" "Scale" "Mirror" 3 buttons appear on left bottom coner.

⑤ When you are satisfied with the Cura settings and the position, orientation, and scale of the model, select File > Save GCode. Save the file to the micro SD card that came with your printer.

7.3 Step3 Send the file to the printer. (Off-line Printing or Online Printing)

Off-line Printing

① After saving the G-Code to the it, remove the micro SD card from the computer.

② Insert the micro SD card, upside-down, into the slot on the right side of the control box.

③ By using this method, no computer needs to be plugged into your printer and it will run autonomously until the print is finished.

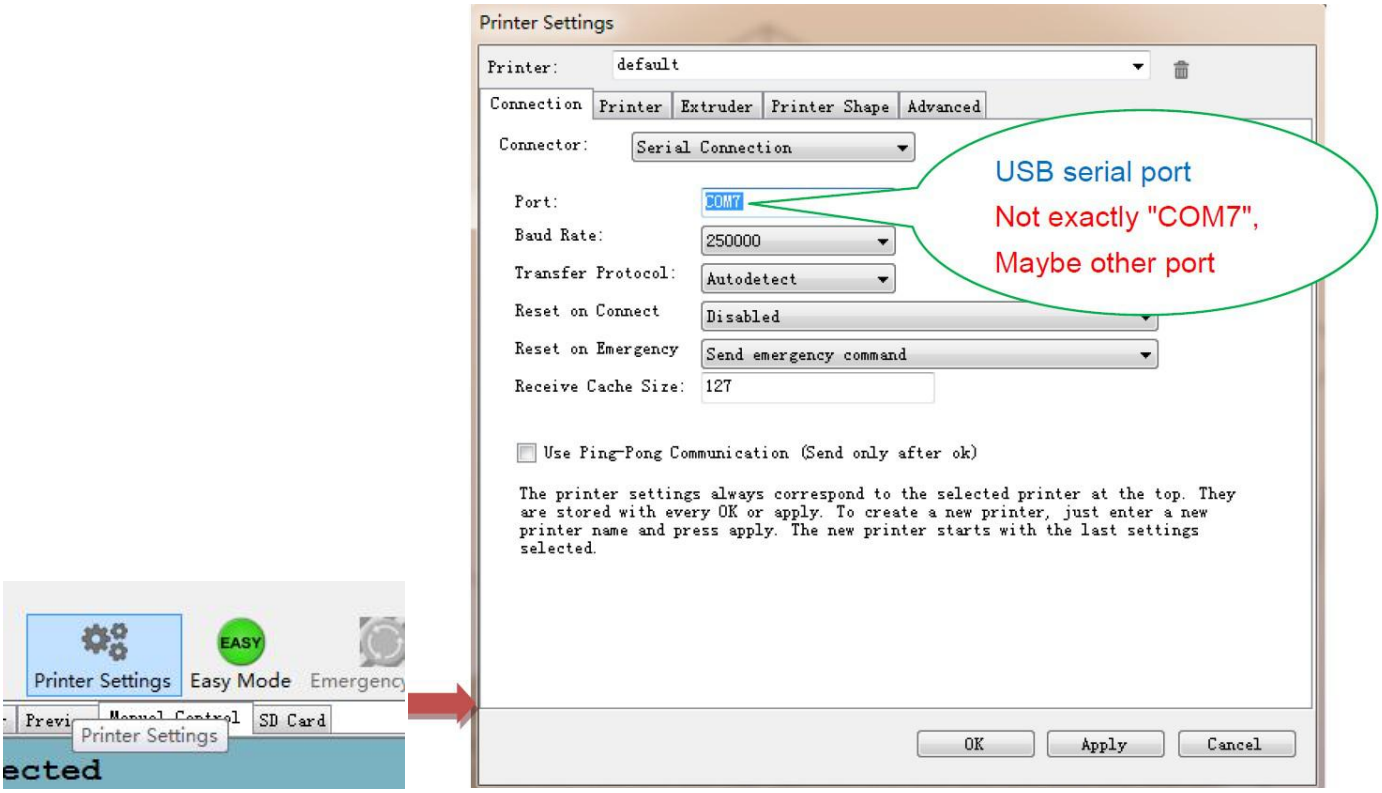
Online Printing

Online Printing: Connect the printer to your computer and then you can get start printing by computer control.

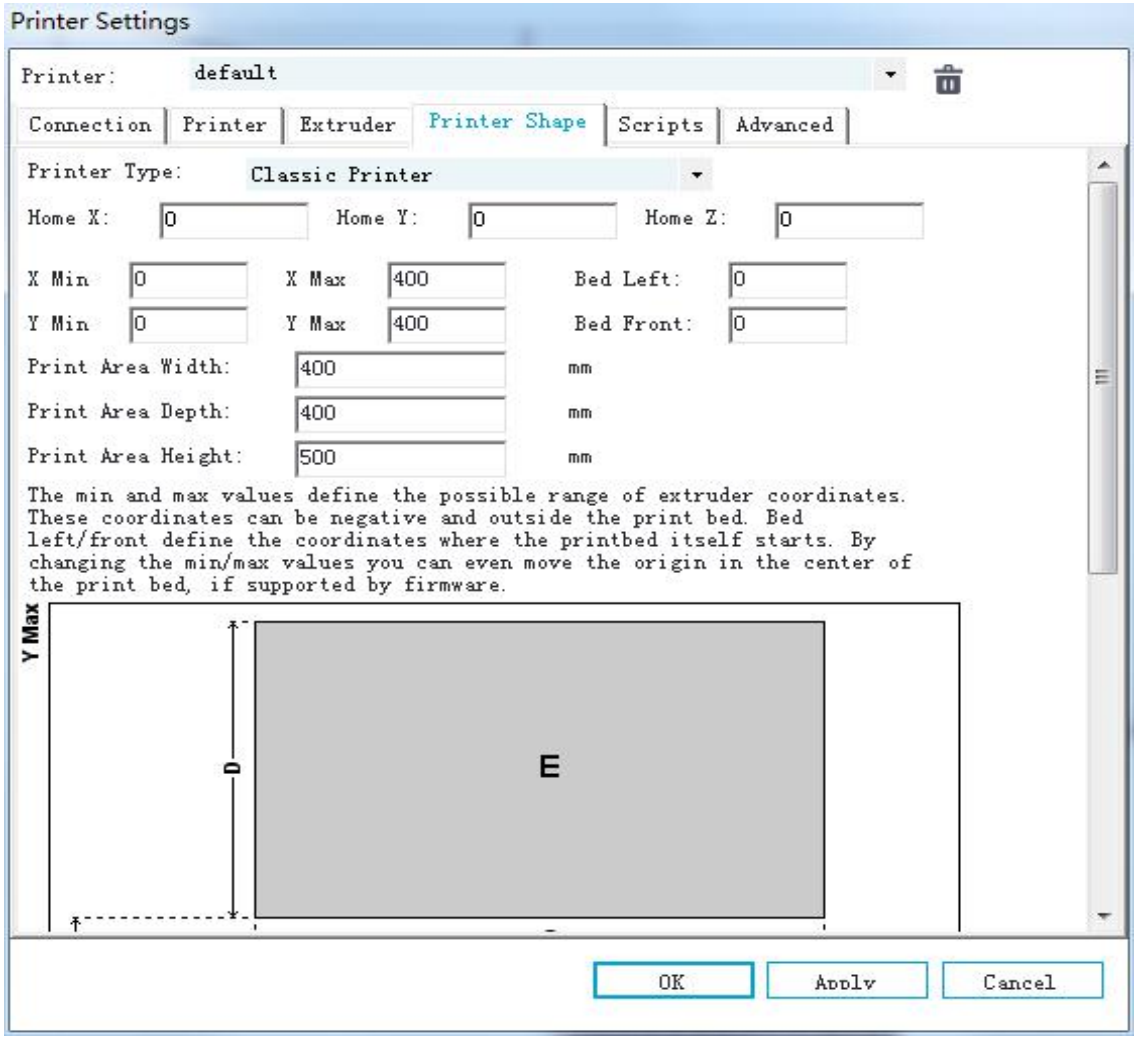
Connecting to your computer with USB cable: Please open the "SD-Card: \ Software \Repetier Host_1_0_6" and click the software"Repetier Host_1_0_6", then you can start printing with slicing the printing file by Repetier according to "How to use Repetier".

Note: How to set up 3D printer and computer connection

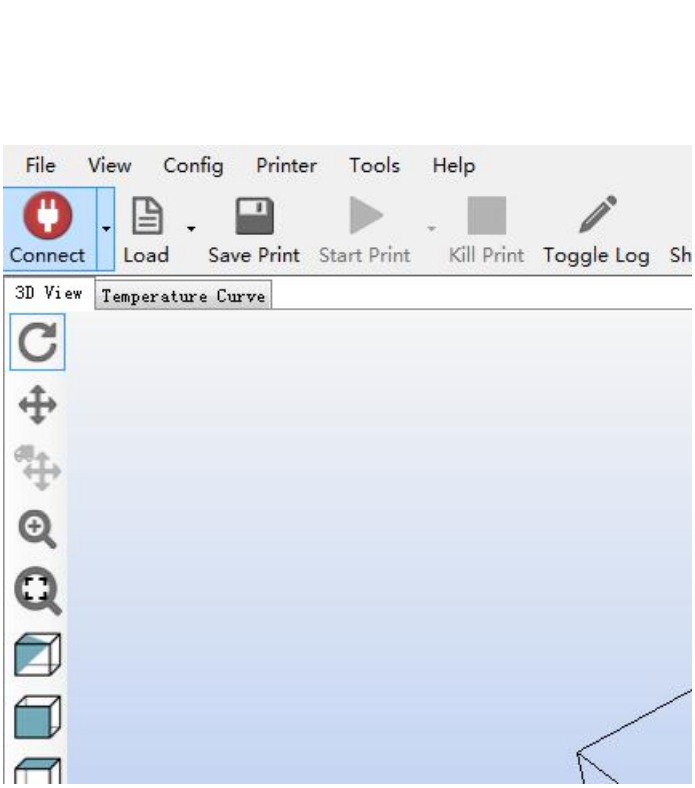
Step1: Choose the correct baud rate and COM port

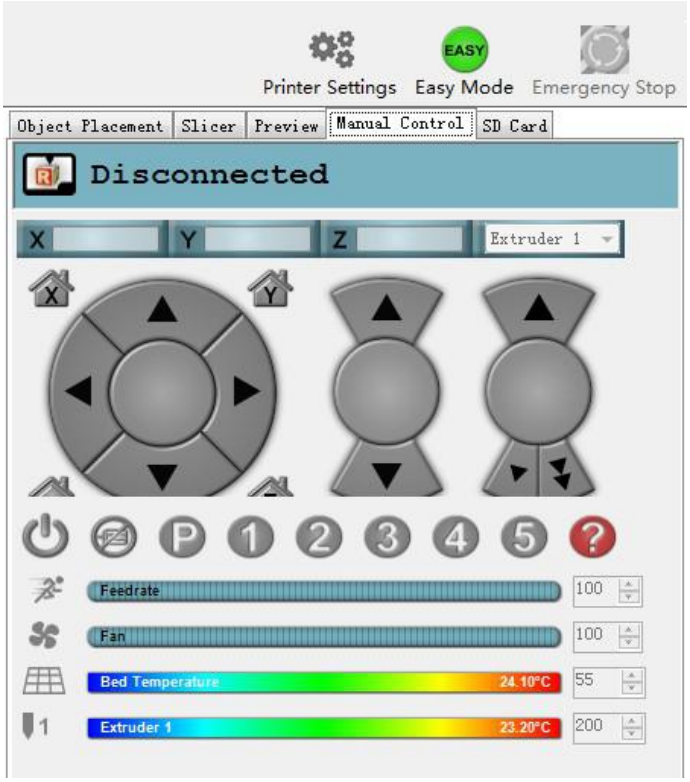


Step2: Set the print range

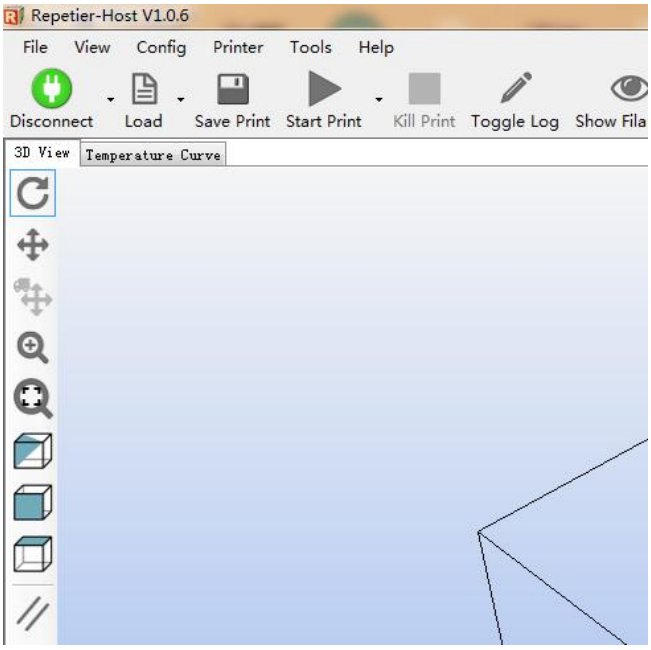


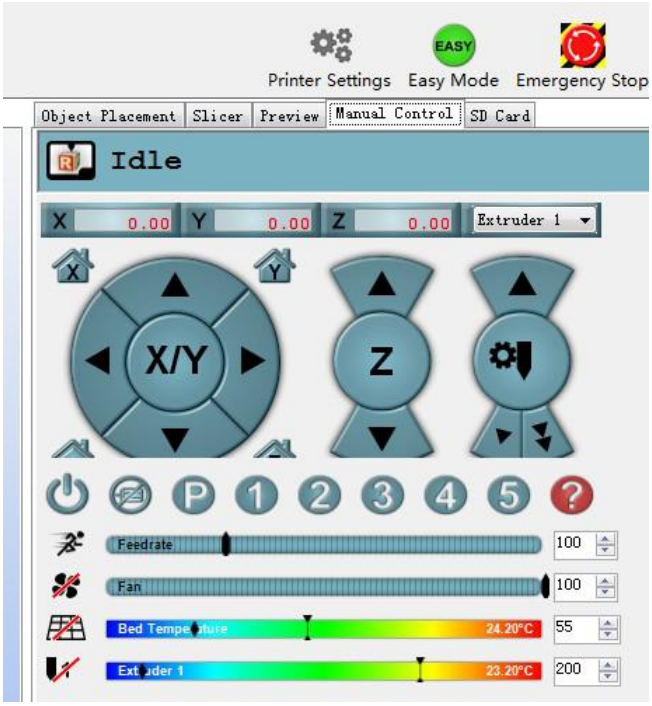
Step3: Start to connect





Step4: Connecting successfully





7.4 Step4 Print

① On the U12/U18 control screen, select Init./Change SD card, the last selection on the bottom of the main screen.

② Select Print from SD.

③ Select your print (*.gcode) file.

④ Watch the 3D printer create your model!

8 Frequently Asked Questions and Solutions

Please get the “Common problem” from the SD card for your reference (SD-Card: \Common problem)

This section covers the most common printing problems and how to fix them. If this section doesn’t solve your printing problems, then contact our Customer service. We’re here to help you every step of the way to get you 3D printing!

8.1 Step-by-Step Troubleshooting

Step 1: Diagnose the problem.

This may seem obvious, but many problems can be solved if you take a step back and see exactly what the printer is doing incorrectly.

Step 2: Determine if the problem is mechanical or digital.

① Mechanical problems are in the actual operation of the printer, such as the motors that drive the X, Y, and Z axes, the motor that pushes the filament, the nozzle heater, and the level of the build plate. The most common mechanical problems are caused by a build plate that is not adjusted correctly, a clogged nozzle, or an unplugged connector on a motor or limit switch.

② Digital problems are in the slice file that you prepare in Cura or other slicing program. Important slice file settings include layer height, print (nozzle) temperature, and print speed.

Step 3: Fix the problem.

Once you’ve discovered what the actual problem is, you can fix it with the following procedures. (If none of these fix the problem, contact us.)

8.2 Digital problems: Slicer settings

① Check your slice file in Cura. Make sure the print is centered in the build area, making good contact with the build surface, and isn’t too big for the build envelope.

② Check the slice settings. Make sure that the layer height is between 0.1mm (high quality prints) and 0.3mm (low quality prints).

③ Check the speed and temperature. For PLA, they should be set to 30–50mm/s and 230°C.

④ Make sure the filament diameter is 1.75mm and the flow rate is 100%.

⑤ Check the Fill Density (infill) and make sure it is at least 5%. You may need to adjust this value to your liking for your model.

⑥ Your part may need supports. If it’s rounded you may need to select “Everywhere” for the support type. If it’s intricate, it may need to select “Raft” for the Platform adhesion type. A raft is a hatch pattern on the build plate that the model will print on. If the part warps when you begin to build it, you may need to select “Brim” Adhesion support type and re slice the file. (A brim will help the part stick so it doesn’t curl up at the edges.)

8.3 Mechanical problem: Leveling the build plate

Many of the failures in 3D printing are caused by a build plate that is not level or is not the correct distance from the nozzle. See the section “Hotbed Leveling” in this manual.

8.4 Mechanical problem: Filament is not coming out of the Nozzle

When filament is not coming out of the nozzle, the nozzle may be clogged. You have several options for clearing a clogged nozzle.

- You can pull the clog backward out of the nozzle using the “soft pull” method.
- You can heat the nozzle and push the clog through the nozzle.

NOTE: Never leave your printer nozzle at build temperature (210°C) while the printer is idle. The melted plastic remaining in the nozzle will “bake” onto the nozzle and become a hard carbon blockage.

① Try to pull the clog out of the nozzle using the “soft pull” method:

● Turn the machine on, select Prepare PLA, and wait until the temperature

● When the nozzle has reached 210°C, squeeze the extruder release lever with one hand and pull the filament backward out of the extruder with your other hand.

NOTE: Every time you change filament on your 3D printer, remove the filament with the nozzle at around 200°C. This method usually removes all the old color of filament from the hot end, so when you begin printing again, the new color will begin printing immediately.

② Try to push the clog through the nozzle.

● Select “**Prepare** ” → “ **Preheat PLA”**. Wait for the nozzle to reach the set temperature of 210°C for PLA.

●When the nozzle has reached 210°C, squeeze the extruder release lever with one hand and push the filament into the extruder with your other hand.

● Watch to see if any plastic is coming out of the nozzle.

●Turn the machine off and wait 10 minutes. The nozzle needs to cool down completely.

●Do a “soft pull” (step ①) to remove remaining filament in the nozzle.