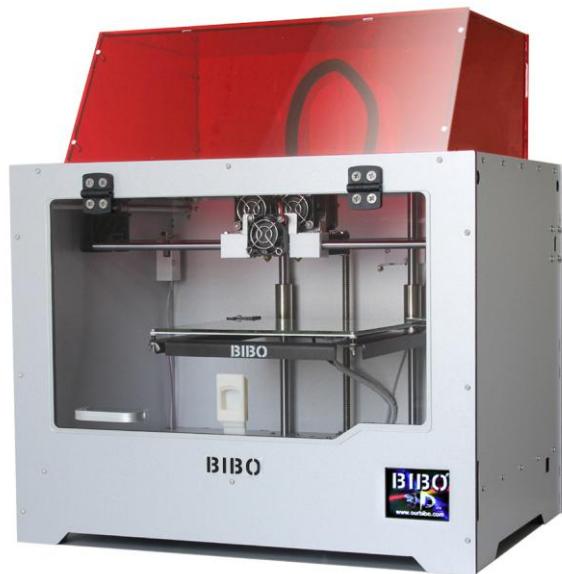

BIBO2 TOUCH 3D PRINTER

---Operation Manual



SHAOXING BIBO AUTOMATIC EQUIPMENT CO., LTD.

V2.0c

Please operate the machine strictly according to this operation manual to have a promised warranty from BIBO! As the warranty is based on correct operation according to the operation manual.

REVISION RIGHTS RESERVED

Content

1. Caution	2
2. What's in the box?.....	3
3. Initial Hardware Installation	4
4. First Print	9
5. Some Notices for Printing	21
6. Software Installation	23
7. Slicing for A Gcode File	26
8. Acrylic Cover Installation.....	42
9. Function Introduction	45
10. Driver Installation (not necessary)	47
11. WIFI (optional function).....	49
12. Maintenance/Optimal Operational Tips	57

1. Caution

★★★ If unloading the filament, please load it first, then unload it.

1. **Do not leave the extruder heated for long time**, just stay with the printer for heating. The filament might decompose after leaving it at high temperatures, leaving a layer of contaminants behind in the hot-end nozzle, and **the extruder will be clogged**.
2. **Printing PLA not more than 200 degree, usually 180-190 degree is the best.** Setting a printing temperature above 230-245degree for PLA is starting to get into bad territory as the plastic will start to change properties if left in the nozzle for too long and **can cause clogs**.
3. **During heating or close heating or stop the printing on touch screen, the other operation on touch screen will be not responded** as the former action (heating or close heating) should be finished first, and temperature reading will not be correct. **You can power off the printer and power on again, everything will be ok too.** Touch screen sometimes may have **crash fault like computer or phone.** Restart the printer and it will be ok.

NOTE: Before power on the printer, please check printer input voltage showed on nameplate whether fit for your local power supply. Nameplate is stick on the backside or two sides of printer. If not fit for your local power supply or no power to the printer, change the switch position on power source under the bottom plate of the printer to avoid damage.



The BIBO 3D printer is very sensitive to static electricity, so make sure you contact a grounded object before operating the machine.

Before repairing or making any alterations to the BIBO 3D printer it is essential that the machine is turned off and the power cord is unplugged.

The BIBO 3D printer operates at very high temperature; allow the nozzle, the extruded plastic and heating.

Do not wear gloves when operating or repairing, as entanglement may occur and cause injury.

Do not leave the machine unattended when in operation.

2. What's in the box?

The printer may be re-taped during inspection at the customs.

There is troubleshooting files in the SD card which can help you.

The frame of BIBO 3D printer is CNC machined, and we cleaned the printer before delivery. But during violent transportation vibration, **there maybe some machining powder** of the aluminum composite panel come out from the frame junctions or some dust from the cartons. Sorry for this trouble. Please blow the dust away.

Along with your BIBO 3D printer, this package contains the following which are in the small carton above the build platform, there are(the words and pictures are lined in the same order):

1 roll PLA filament, 1 or 2 filament holders, some allen wrenches (packed with black clamps), some bolts and nuts, 1 glue stick, 1 power cord, 1 USB A to B cable, 1 or 2 filament guide tubes, 1 filament tubes couplers support, 2 extruder wire clamps, 1 spare thermistor, acrylic cover bolts and nuts, 1 cross screwdriver, 1 SD card(not inside this small box, but above it)



Please remove all the files we sent to you in SD card to your computer, or SD card printing maybe not run. When you want to print something, please copy the gcode files to SD card.

Red acrylic cover panels may have brown paper protected.

If you need to transport the printer to other place, please keep all the packing carton and material, and take the photos step by step when you do unboxing. In this way, you can pack the printer the same as us, and the transportation of the printer will be safe.

3. Initial Hardware Installation

BIBO 3D printer introduction video

Please kindly check this link from one of our customers:

<https://youtu.be/XdRIPFhfYbI>

His printer is the old model, and the printer you received will have the difference. His printer has the y axis homing problem with noise because of violent transportation. We solved his problem at once after he sending us an email, no need to return. Your printer usually will not have this problem as it was an accident and we also improved the packing.

We also have finished **BIBO parts assembly video and acrylic cover assembly video**, and you can copy it from the SD card we sent to you or download from this link:

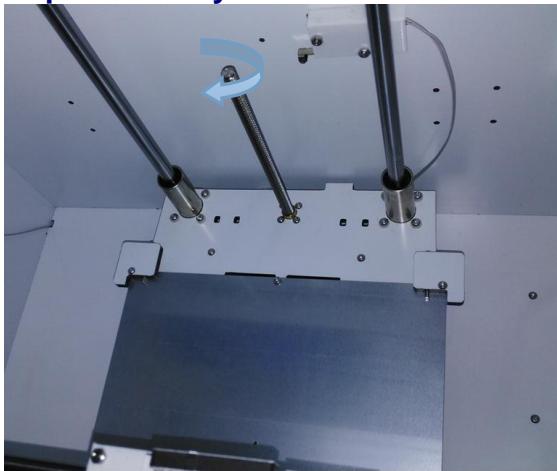
https://drive.google.com/file/d/0B148jWiv_vJeOW5wWmxSd3p3TDg/view?usp=sharing

The file is ZIP format which was compressed, and you can use winrar or other software to open these videos. **There may be some difference, but the operation principle is similar.**

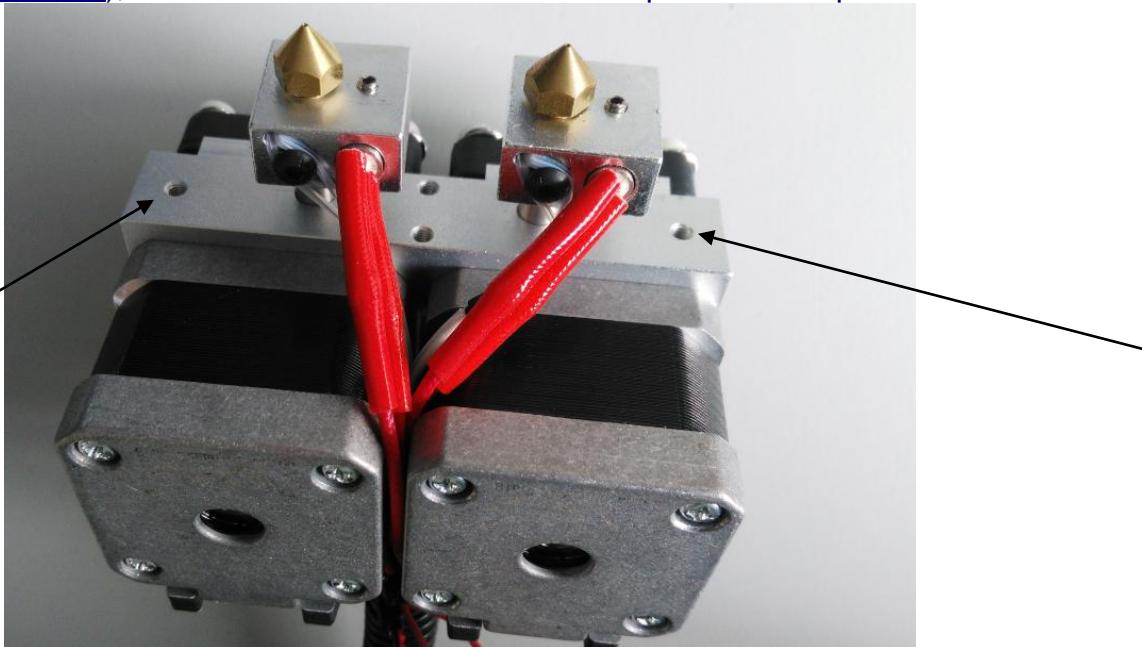
The printer is usually transported by courier. If the printer is all assembled, its volume will be much bigger, and the courier freight will be much more as freight depends on its volume, not actual weight. So we will let big volume part attached such as acrylic cover (fragile for shipping), filament run-out detection part not be assembled, to reduce the expensive transportation freight and let the transportation be more safe. These parts are easy to assemble.

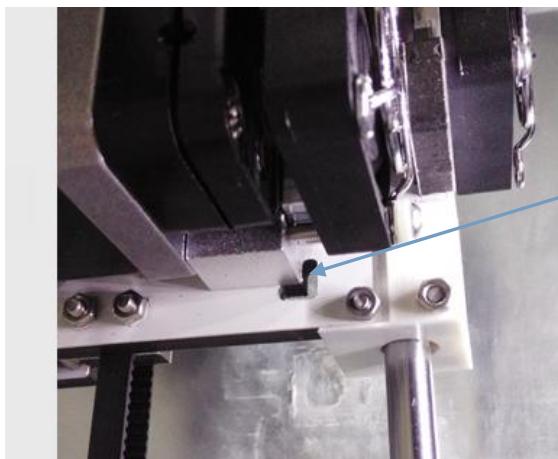
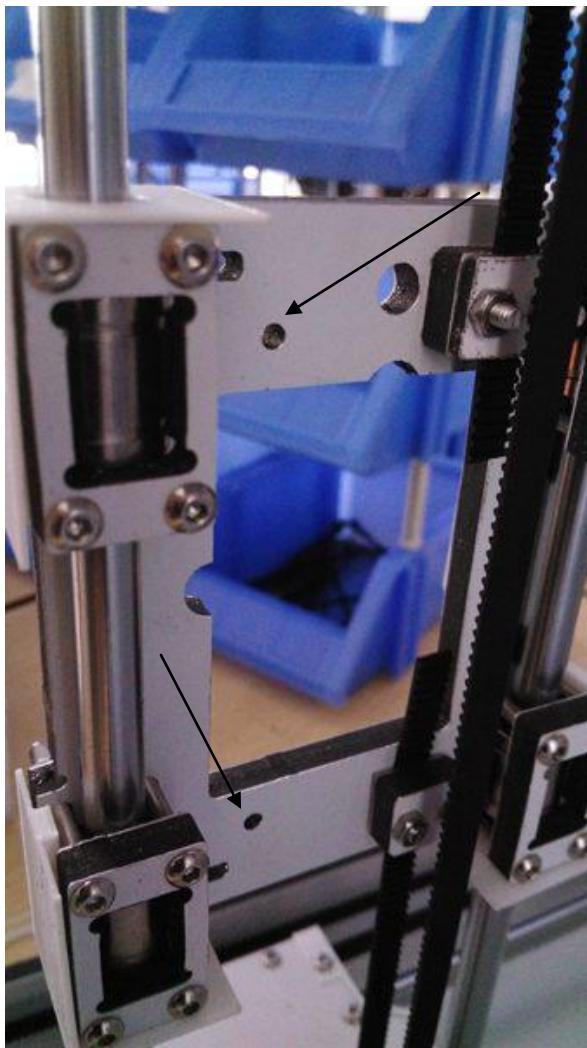
The first step is to take **the machine** from carton , also **taking the parts out from the small carton in the printer first** , then take the **small carton** out from the printer, and **remove all the protection bubble wrap and ribbons**.

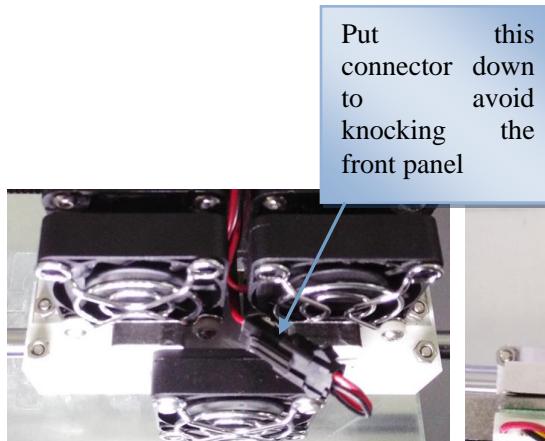
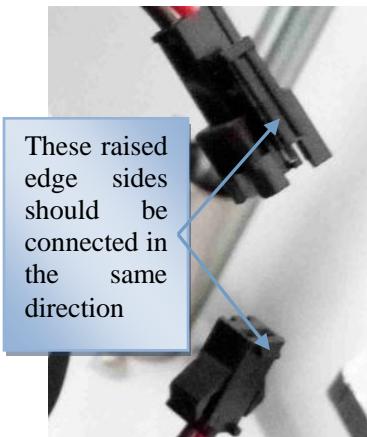
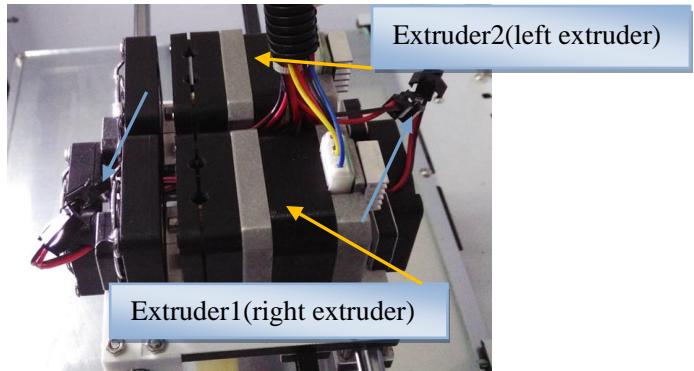
Now raise the build platform by rotating clockwise the lead screw. **Then take out the packed acrylic cover**. Please check the photo below:



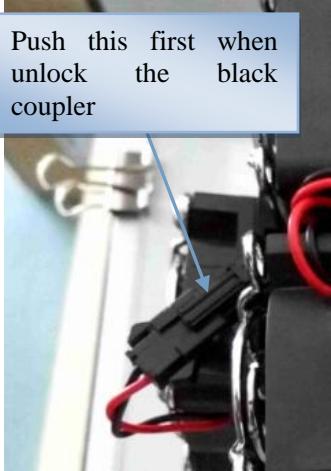
Then install the extruders with two bolts in 10mm or 8mm length (in a bag marked with 10 or 8), and connect two fans with black coupler. See the photo below:



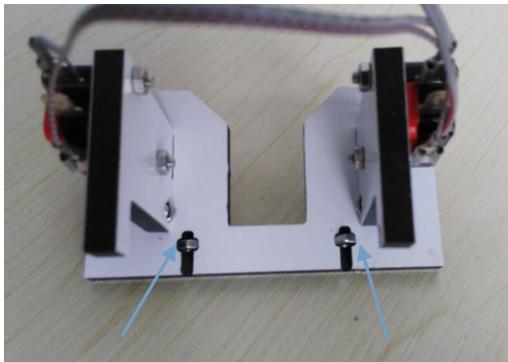




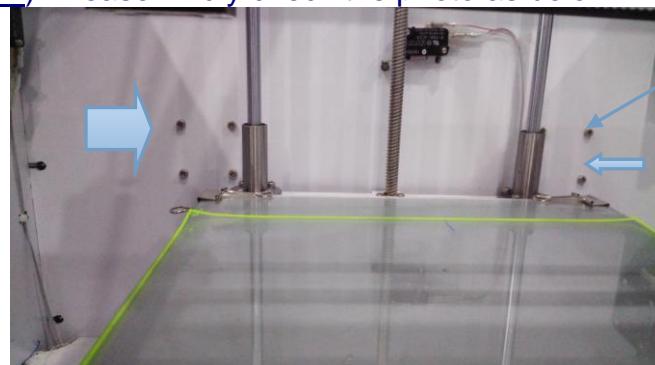
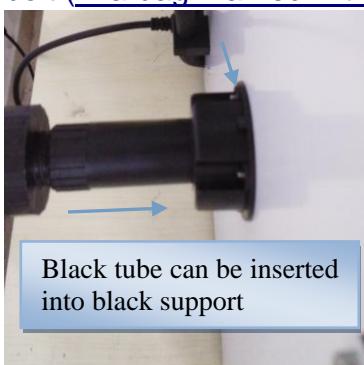
If you want to take the extruders down from the printer, just unlock two black coupler, no need to remove the cooling fan support, the same as you assemble the extruder. When you unlock two black coupler, please push the button first then unlock the black coupler as the photo below:



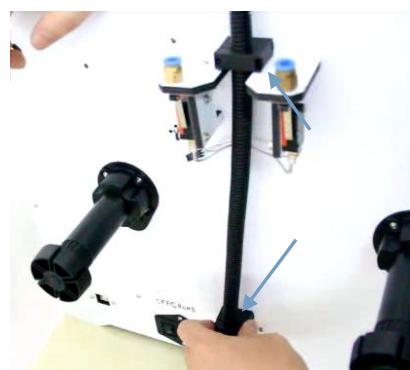
The second step is to install the filament guide tube support on the back of the printer with 16mm length bolts and nuts (bolts in a bag marked with 16 , nuts in a bag marked with 12), and lock the black coupler as the photo below.



The third step is to install filament holders on the back of printer with 12mm length bolt (in a bag marked with 12). Please kindly check the photo as below:

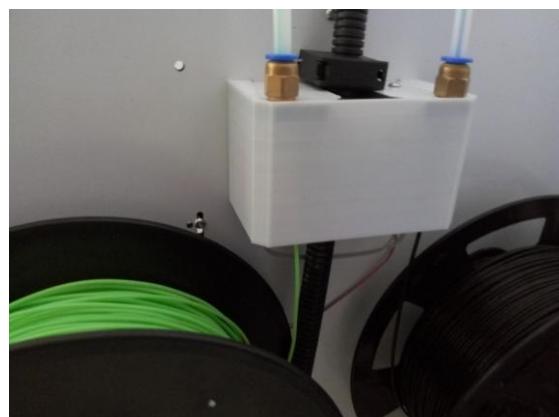


The fourth step is to assemble the extruder wire clamp with 12mm length bolt (in a bag marked with 12), please kindly check the photoes as below(some printers may be assembled already). Let the black wire cable be straight (just only a little tight) . If too loose, the black wire may be not long enough for extruders to travel inside the printer.



Then install the filament guide tube(black or white) to the filament tubes coupler (with blue cap) on the back of the printer. Please insert the filament guide tube to

the coupler directly. If it is hard for you to insert, press the blue cap of coupler and then you can put the filament guide tube into the hole of coupler. In the end put white cover on the filament guide tube support(when you load the filament you can take it off). Please kindly check the photo as below:

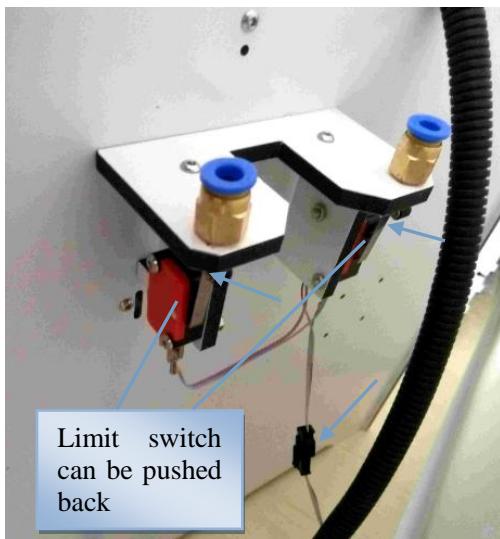


Congratulations! You have completed the initial hardware installation!
For acrylic cover installation, it is in chapter 8. You can install it after first printing test.

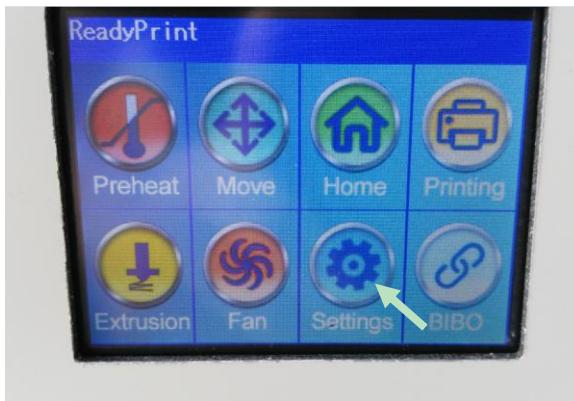
4. First Print

The first step is loading the filament for two extruders

BIBO touch 3D printer has the function of filament run-out detection. **That means when there is no filament loaded in filament guide tube support for two extruders in the back of the printer, you can't print anything or do any laser engraving.** (If you don't want this function or you just want to load only one spool filament for one extruder, you can unlock the black coupler. If you want to **print flexible filament(soft filament)**, you'd better tie the limit switch with tape or rope to push the switch back for better printing quality)



- 1) Turn on the power switch on the back of printer and you will see the main menu:



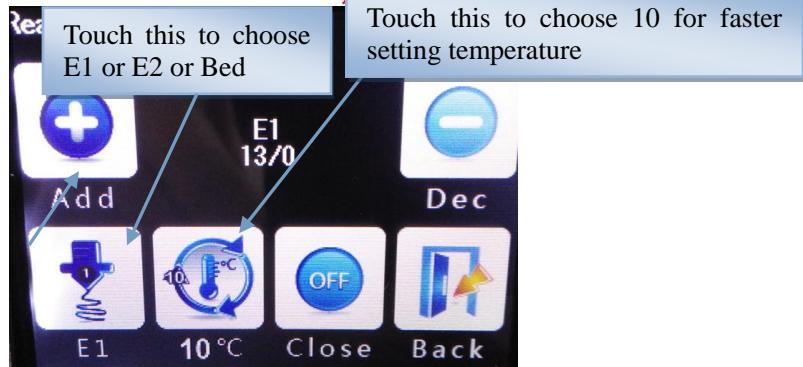
- 2) Now use your fingertip to touch the "Settings" button, and you will see
NOTE: Using fingernail or a pen, not finger bellies or soft part to touch the touch screen, or will let some move button not be responsive. Please kindly check our way to click the touch screen on this link: <https://drive.google.com/open?id=0B76TjeOksIHINI9fZUI3U0xoMTg>



Then touch “Change” menu,
ReadyPrint>...>Filament Change



Then touch “Preheat”,



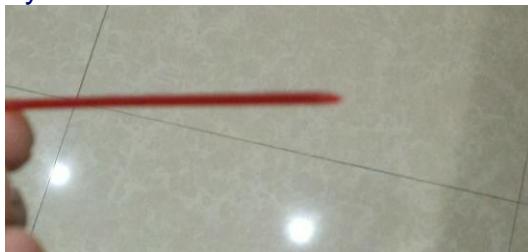
3) You can choose nozzle E1(right extruder) and nozzle2 E2(left extruder), then set the temperature at 220°C by “add” button as usual (this temperature is according to your material type and manufacture’s instruction. For **PLA usually 200°C**, and for ABS usually 220-240°C. It also based on the existing filament in the nozzle). Now you will see two extruder’s temperature is rising.

4) Now we will do the **loading filament step**. Put two spools of filament in the back of the printer (We send one spool of PLA filament, and you can cut some from that spool to have a roll of filament). One runs clockwise and the other one runs counter – clockwise. For not rolled on spool test filament, is the same operation.

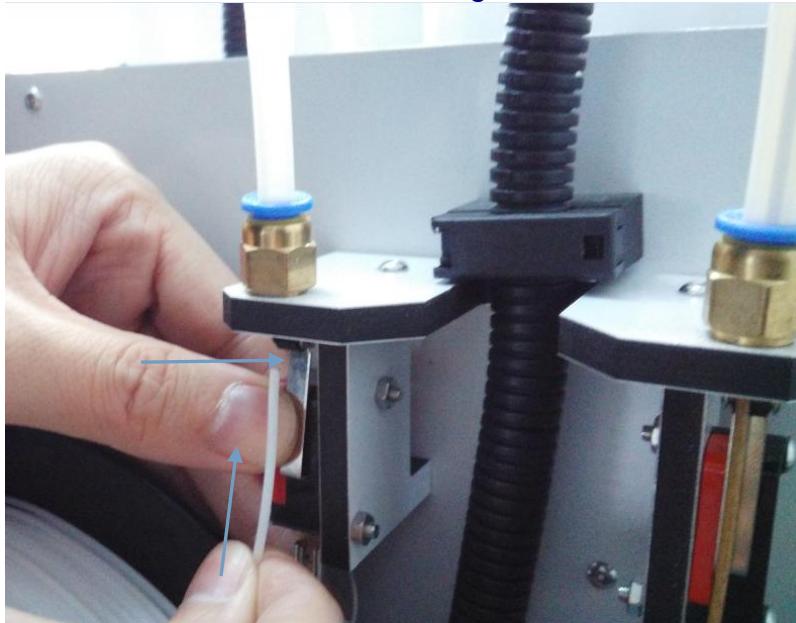


Sharp the filament head with scissor and let the filament as straight as you can to

ensure it come to the hole of extruder smoothly. If it is difficult to put the filament to the hole of nozzle on the way, please rotate the filament in different direction and try more times.



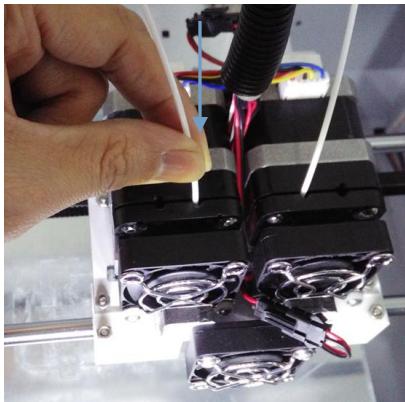
Load the filament to the filament guide tube for two extruders as the photo below:



When the extruders reach 220°C, please touch "back" on touch screen to go to filament change interface, touch "In"



Then insert the filament to the extruder which is heated before.



This is for E2 (left extruder)

If the filament come out from the extruder's brass nozzle, touch "stop" to stop loading filament to this extruder. **It will still loading (extruding) the filament for a while.** And touch "Preheat" to go to the "preheat" interface again



Touch "close" to let the extruder's temperature come down (**if the filament stays in the hot nozzle for long time, the extruder will be clogged. We should cool down the extruder after stopping the loading of filament**).

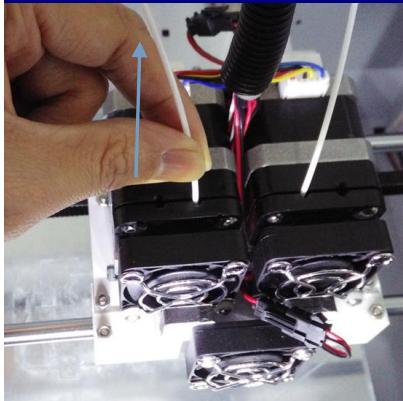


Now loading the filament to the extruder is **finished and successful**.
The printer can only load or unload the filament for one extruder, please do it one by one. During loading or unloading the filament, other operation on the touch screen will be delayed to respond except stop button, the same as heating (if you are heating the extruder, other operation on the touch screen may not be responded. You can restart the printer to solve this).

If you want to **unload the filament**, the operation is the same as loading filament, heating up the extruder to filament melting temperature such as 220°C. The difference is to **load the filament first (In) until the filament come out from the nozzle,**

then stop loading the filament (In), unload the filament (Out) .

Finally stop the filament change and close the heating of the extruder and pull the filament out.



Do not leave the extruder heated for long time, just stay with the printer for heating. The filament might decompose after leaving it at high temperatures, leaving a layer of contaminants behind in the hot-end nozzle, and the extruder will be clogged.

Please don't disassemble the extruder part as the photo below:

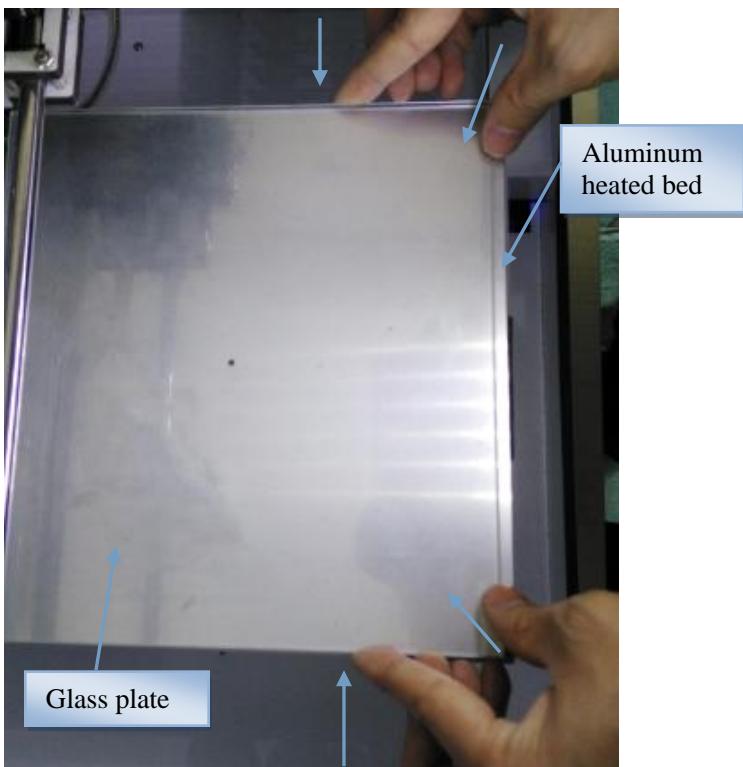


We have calibrated the tension for extruding system. If you disassemble them, the calibration will be broken.

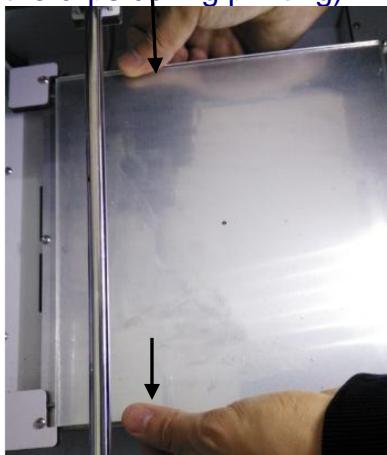
The second step is Leveling the heated bed(not necessary).

After transportation, **only a few printers** have to be leveled the heated bed to ensure your 3D prints stick well to the heated bed. **How to check whether you have to level your heated bed, please follow steps below.**

1.Put the glass plate(packed with acrylic plate) on aluminum heated bed.

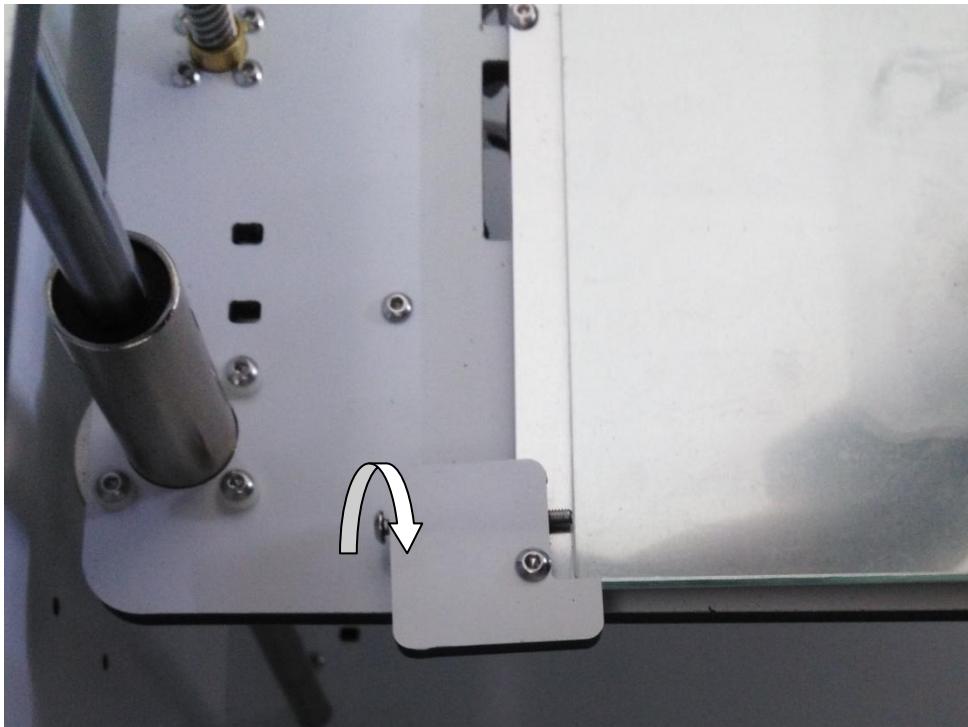


Push the glass plate to the bolts in the back of aluminum heated bed and press it down to install it . Let the glass plate be well contacted with aluminum heated bed by pressing the sides of glass plate as the photo below (**please don't use any clip to fix the glass plate**, or the heated bed will be damaged and nozzle will hit the clips during printing):

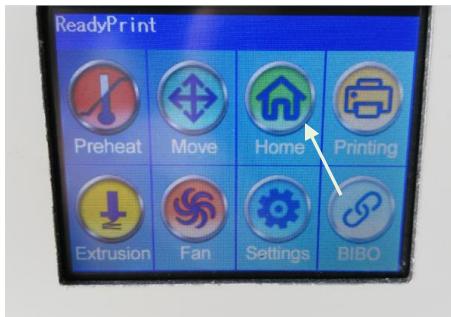


Please also check the glass bed whether can be easily moved in horizontal direction.

To hold the glass bed, you can screw the bolt to fix the glass, but not too tight, just a little contact with the glass as the photo below. **Usually you don't have to do this, as we already did it before delivery.**



2. Start the machine to see the main menu and touch the “Home” button



3. Now use your fingertip to touch the “Home” button to run the homing.

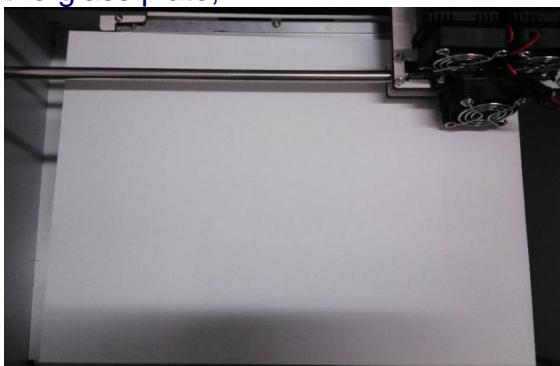


After homing, you can turn off the printer power. **Move extruders' nozzles to be up on heated bed slowly. If nozzles can be moved on heated bed (even the nozzles knock the bed and a little below the bed but still can move on the bed), then you can go to the third step to print the test gcode file first.**



If nozzles can't be moved on heated bed as they are below heated bed glass a lot, you should do following steps to level the bed as below:

Now the printer is still power off, put a paper such as our warranty policy paper on the glass plate,

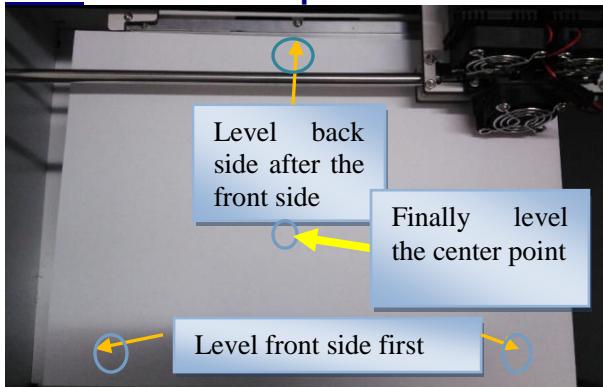


Rotate three wing nuts Clock-wise to let the glass plate down and nozzles can move on the paper.

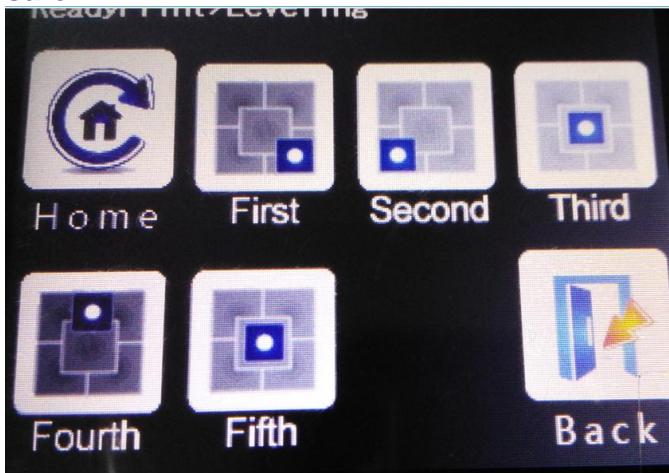


Now move the extruder between the printer nozzles and heated bed to let a A4 paper be very hard to pull through nozzles and the bed. You can adjust three wing nuts and bolts under the heated bed. **first level the front side with two holes and fix in place, then level the back side with one hole, finally level the center of the bed, just in this order to level the bed.** If it is hard to level in 4 points, just ensure the center of the bed leveled first. After printing, when there is some places not stick well for the print, we can apply more glue

there. So the lower place can be even too.



You can also use level menu on touch screen (not recommended), just click first to fifth one by one with fingernail or a pen, checking the paper movement and adjust the wing nuts. This level menu has the bug now, sometimes it will be not responsive. We recommend to move the extruder by hand which is faster and easier, and can feel the obstruction between nozzles and the bed, also safer.

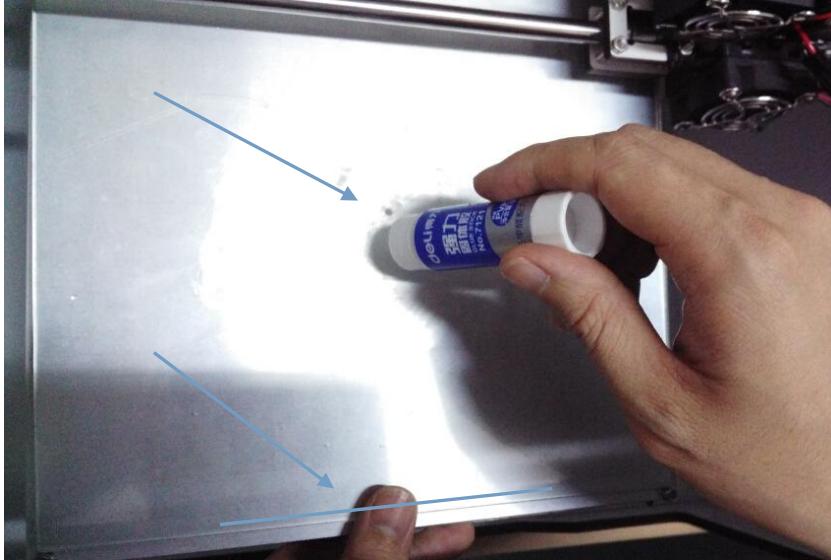


If the heated bed is easily going down during leveling, please move the extruder to the left front corner near you by hand, then turn on the printer power, only do home Z axis. In this way, the heated bed will not come down during leveling as Z axis motor has the power to lock the heated bed, and you can move the extruder everywhere by hand as X Y axis motors are not locked.

The third step is to print from SD card.

To promote stickiness of first layer printing, your machine was delivered with a stick of glue. Using glue is optional but tends to help quite a bit. Don't go crazy with the glue though, more is not better. Clean your glass plate and then spread a bit of the glue onto it. With a damp piece of paper or cloth spread the glue out over the bed and let it **dry** (starting a print that uses the heated bed will make it dry quickly). After the water evaporates you will be left with a very thin layer of glue. Usually you don't have to use glue before every print. You just spread the glue on the place will be printed, and **every print we will set the object to be printed in**

the middle of glass plate and extrude lines in the front side to purge the nozzle. If it is hard to remove the prints from print bed after printing, maybe glue is applied too much in the print area. You can let the prints and glass be in the water, as our glue is soluble in water, and it is very easy to remove the print from glass.



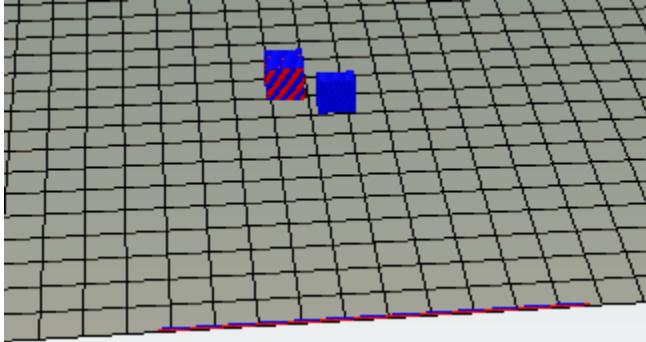
We will give you two test gcode file in SD card's test gcode file sent with the machine to print 1 cm height cuboid, one is for testing the right extruder, and the other is for testing the dual extruder printing(right extruder prints main object, and left extruder prints support). You can insert the SD card to the right panel of priter to choose them to test the machine with our filament sent to you.

y186test dual extruder cube with pla filament 190 degree 15 minutes
 y186test right extruder cube with pla filament 190 degree 10 minutes

2016/4/1 16:06 Repetier-Host
 2016/4/1 16:06 Repetier-Host

311 KB
 78 KB

PS. For dual extruder printing test, the blue one in the photo below is the only object you want to print, the other one in red and blue is wipe & prime tower for better dual printing, so its printing quality is not concerned.



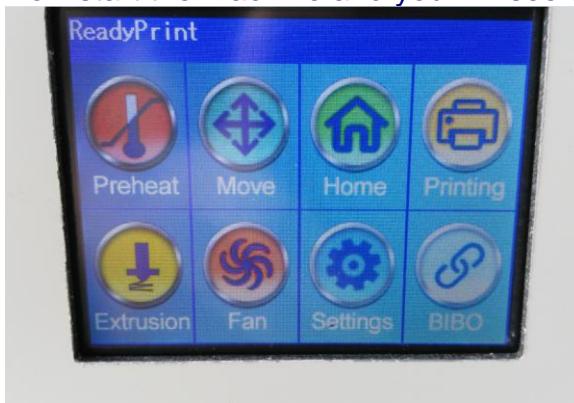
Before the first print, there are something IMPORTANT to read first:

- When the nozzle was heated up, you will see the filament will come out from the nozzle, please cut the coming out filament and clean the nozzle with a insulated scissor or Diagonal Cutting Pliers before the nozzle temperature arrive at target temperature and start to print. If not, the coming out filament

may be stick to the nozzle or first layer during printing, and the first layer printing will come to be failure. To avoid the nozzle drag the print, please clean **two** nozzles first before printing started.

2. Sometimes, you hear extruder gear clicks or nozzle is knocking the print even knocking off the print, this may be caused by that the nozzle is too close to the heated bed. Please level the heated bed again. **Nozzle knocking off the print can also happens when printing some special object** such as overhangs as the end of overhangs is easily warping. The other nozzle will knock the warping print, and it is the disadvantage of dual extruders 3d printer. But you can **preheat the other extruder which is not for printing and load the filament first then finally unload the filament before your printing**. In this case, the other nozzle not for printing will be always hot, even it hits the print, it will not ruin the print.
3. When it start printing, please keep your eyes on the heated bed to see the first layer. If the first layer is printed successfully, your printing has totally been succeed in half. If the first layer can't be well stick to glass plate as the nozzle is too far away from the glass plate, you should level the heated bed again.
4. Please don't print in too slow speed (below 20mm/s). The stepper motor running too slow will cause sympathetic vibration and noise, and the motor driver will get more heat for long time printing. To protect itself, the printing will be stopped.

Now start the machine and you will see the main menu



Press "Printing" button and then select the gcode file. Now 3D printing will be automatically started. If you can't print from SD card, please remove all the files in the SD card to the computer. Then format the sd card on the computer into FAT32, finally copy the test gcode to SD card. Usually the SD card will work on the printer.



Now your first print test is finished.

5. Some Notices for Printing

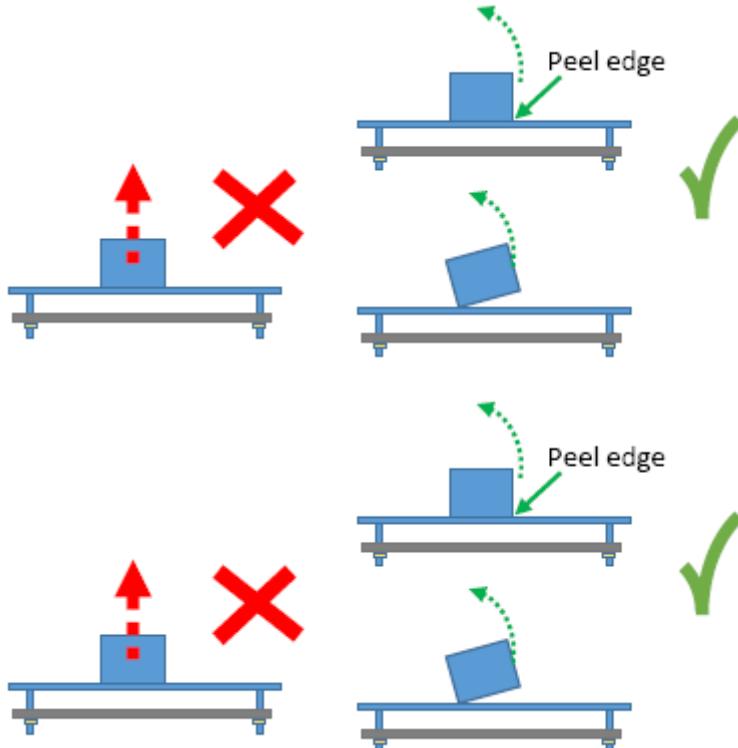
1. Some filament were not good winded on rolls in order by filament manufacturer or the user, so it will be tied a knot. See the photo as below:



So please keep an eye on the filament roll and loose the filament if there is a knot. If not, you will see the base of printed object is ok, but the top part is bad in printing.

2. When you hear extruder gear clicks and no filament come out from the nozzle during printing, it may be caused by the filament is broken away into two pieces in the extruder. Some filaments has good hardness but too fragile. Now you should unload the filament out. Sharp the filament head with scissor and let the filament as straight as you can to ensure it come to the hole of extruder smoothly. Loading the filament into the extruder again.

3. Remove object after print completes, please Let bed cool down; Don't vertically pull object away from print bed; Start peeling at edges by Diagonal Cutting Pliers or something else ; After peeling edge, try to twist it from bed. You can take the glass plate down form the printer for easier removal. Sometimes it is hard to remove the print, please heat the glass bed to 70 degree, and it will be easier to remove the print. Please wear glove to hold glass bed as it is very hot.



4. Please put the printer center on a level table, not the edge of table. The printer may be moving during fast speed printing especially for smooth desktop. So please let the printer be horizontal, and have good friction to the desktop.

5. SD card may not be detected when you insert it to the printer which is power on, you can refresh it or power off the printer and restart the printer.

6. When start the printer, you will hear the voice "zhi" of motor lock. It is normal, the voice will disappear after some time.

7. When the printing is done, please do not turn off the printer at once until the nozzle temperature be under 50 degree Celsius, or the extruder will be jammed. When you stopped the printing through touch screen, the nozzles' temperature may be held and extruder fans will be always running in that case. If you want to continue to print, it is ok for you to save printing time without heating again. If you don't want to print any more, you can turn off the printer then power on it again, waiting the nozzle temperature be under 50 degrees.

8. When you start the printing, the touch screen will show you the nozzle's target temperature, but the nozzle's actual temperature come down even to room temperature. Please don't worry, **the printer will heat the bed first**. After bed temperature comes to the target temperature, the nozzle will start to be heated and its temperature will rise up to the target temperature. During heating, operation on the touch screen may not be valid or the screen be dead. You can wait the heating ended to operate or **power off the printer and power on again** to solve this problem.

9. If you want to print flexible filament, Best material to choose for our machine is PLA flexible 1.75mm or TPU 1.75mm, and setting slow speed such as 20-30 mm/s.

10. When printing, please don't insert usb cable, or the printing will be paused for a while then continued. It will affect the printing quality.

12. In summer, the room temperature will become high, so we suggest you to make sure the printing condition under 30 degrees. If the room temperature is warm and there is no wind, you can also try printing without the acrylic cover.

13. There is some u disk can't be detected by the printer when inserted. You can insert the u disk then restart the printer, then it will work. Meanwhile in the settings menu on touch screen there is the option for SD or U disk. Please choose U disk if you want to print from U disk.

6. Software Installation

Repetier-Host and **Cura** are the best software to use with BIBO 3D printer. They can process your CAD files (STL files) and make them printable.

For software-**Cura**, please check another **operation manual in SD card's "BIBO maker touch cura settings" folder**, **Mac system users are recommended to use Cura**, as it is more friendly than Repetier-Host.

To install Repetier-Host on your PC take the following steps:

1. Copy the installation document of Repetier-Host in the SD card sent with the printer to your PC. You can also download the latest version of Repetier-Host from <http://www.repetier.com/downloads/> .

2. Run the installation document of Repetier-Host and follow the installation instructions. During installation it will ask to install a driver. This is recommended to do. For how to install and use Repetier-Host on your PC for windows, Linux, check the link below:

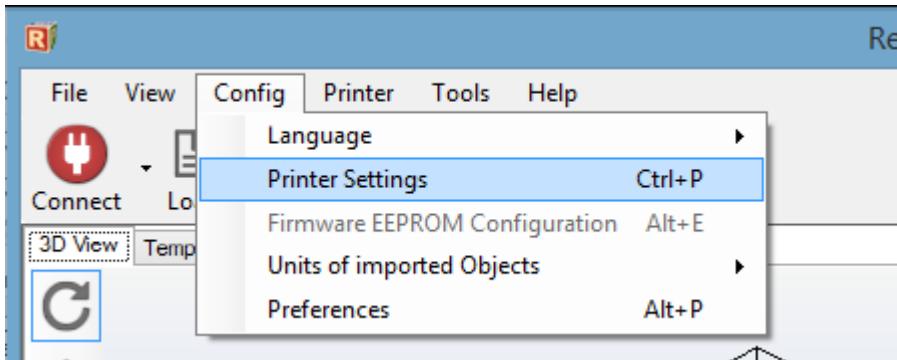
<http://www.repetier.com/documentation/repetier-host/>

For Mac users, please kindly check the link below:

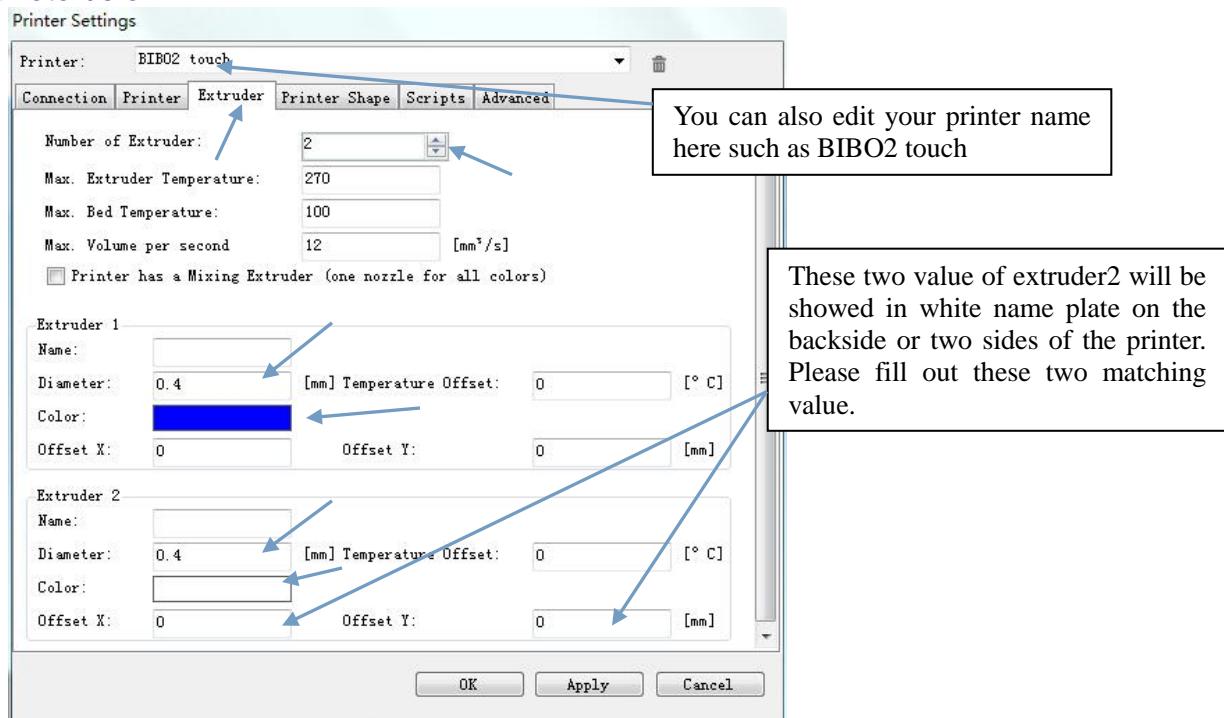
<http://www.repetier.com/documentation/repetier-host-mac/>

3. Start Repetier-Host.

4. Go to Config -> Printer settings.

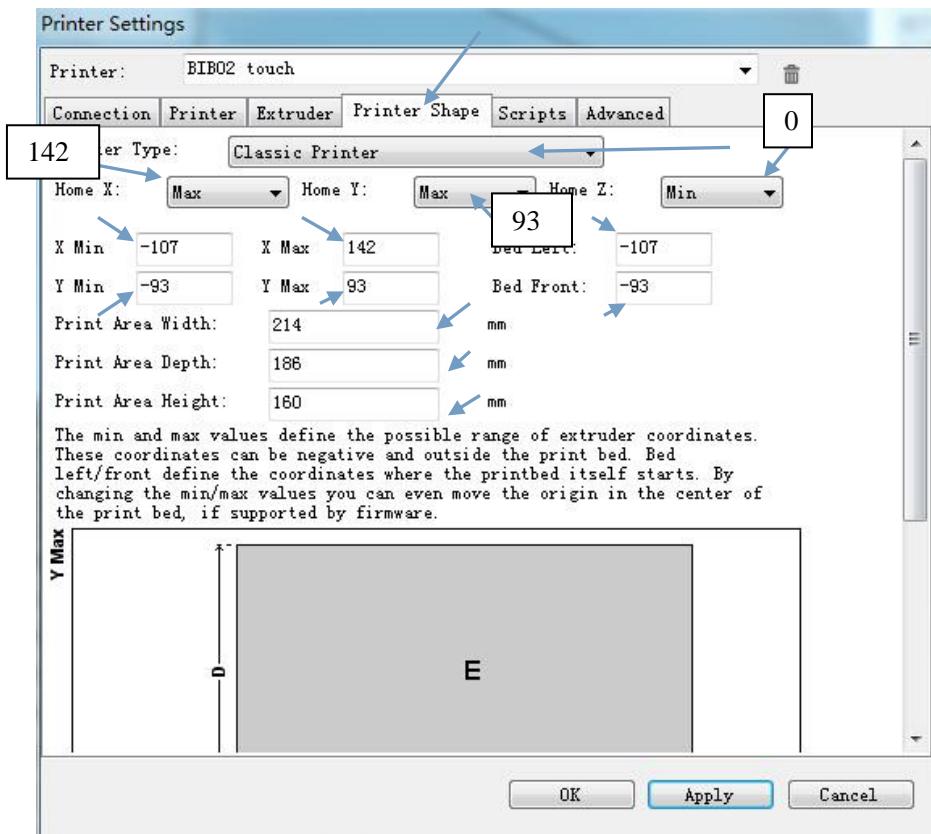


5. Once you see the printer settings window, please fill out the parameter as photo below:

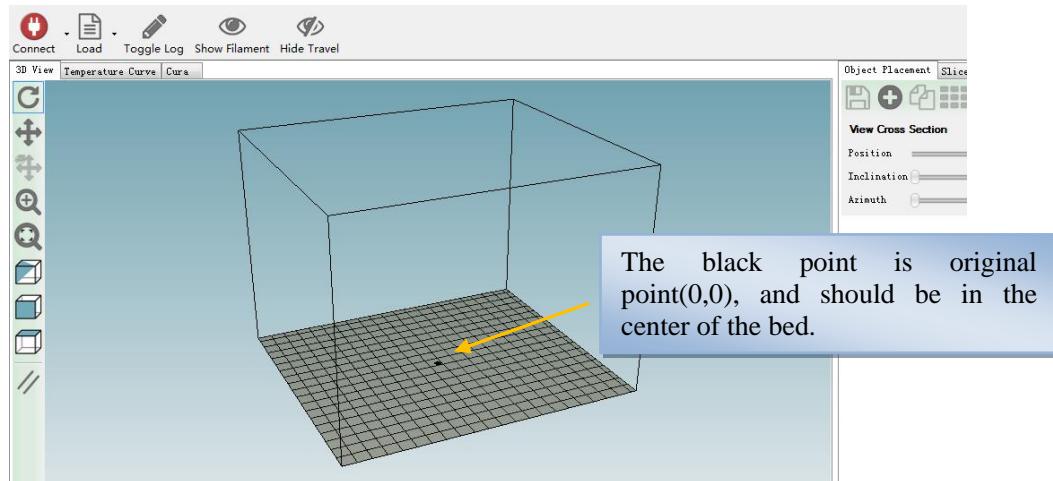


PS. Extruder 1(the right one near x axis motor) is the first extruder. Extruder 2(the left one) is the secondary extruder. For Printing PLA, left extruder is better than right extruder.

If your printer has a 350°C optional extruder, that means your extruder 2 can be heated up to 350°C. Please **do not use 350°C optional extruder to print PLA**, the extruder will be jammed as there is no teflon tube in the hot end. **To print in high temperature, Usually need low printing speed, enclosed environment, warm room temperature, high bed temperature and patience on printing experiment.** Meanwhile, when you change filament for 350°C extruder, please **heat up the extruder to the old filament's printing temperature, let the old filament come out from the nozzle completely**, then heat up the extruder to the new filament's printing temperature to upload the new filament. This will reduce the jam risk.



After configuration of the printer, you will see the photo below. If the black original point is not in the center of the printer, please check the configuration photo above.

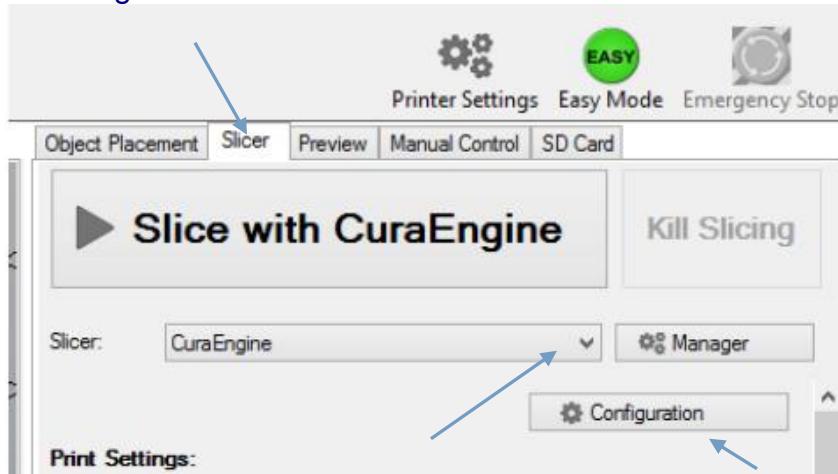


7. Slicing for A Gcode File

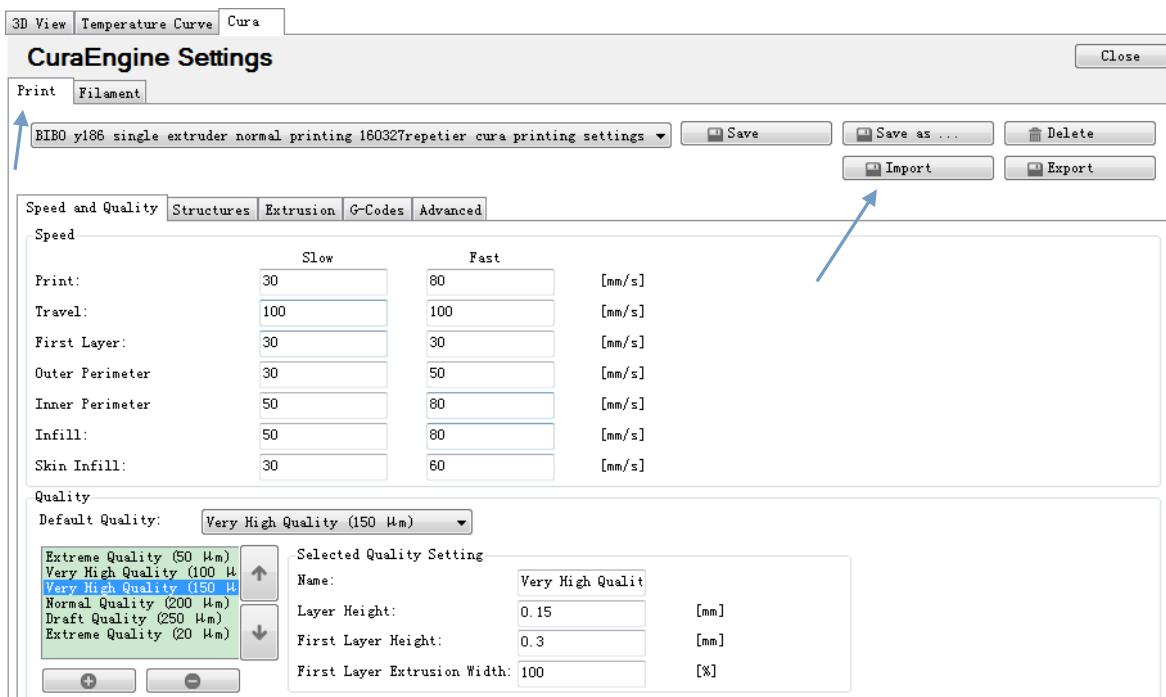
Repetier-host is an open-source software and is supplied software with our printers. We will give you repetier-cura settings, and you can import these settings to repetier-host to slice the printable object.

Single extruder printing:

Now start the Repetier-host, click the Slicer tab, and choose the slicer to be CuraEngine.



Then click Configuration, choose the Print tab. Click Import button, then import BIBO y186 **single extruder normal** printing 160327repetier cura printing settings.rcp from SD card's "BIBO maker and touch repetier cura settings" folder. You will see our printing settings as the photo below:



3D View | Temperature Curve | Cura |

CuraEngine Settings

Print Filament

BIBO y186 single extruder normal printing 160327repetier cura printing settings

Speed and Quality Structures Extrusion G-Codes Advanced

Infill

Shell Thickness: 0.8 [mm]
Top/Bottom Thickness: 1 [mm]
Infill Overlap: 0 [%]
Infill Pattern: Grid
 Solid Top Infill Solid Bottom Infill

Support

Support Pattern: Grid
Overhang Angle: 20 [°]
Fill Amount: 30 [%]
Distance XY: 0 [mm]
Distance Z: 0 [mm]

Skirt and Brim

Skirt Line Count: 0 Brim Width: 4 [mm]
Skirt Distance: 3 [mm]
Minimum Skirt Length: 150 [mm]

Raft

3D View | Temperature Curve | Cura |

CuraEngine Settings

Print Filament

BIBO y186 single extruder normal printing 160327repetier cura printing settings

Speed and Quality Structures Extrusion G-Codes Advanced

General Extruder Settings

Spiralize Contour Enable Retraction Perimeter before Infill
Retraction Speed: 50 [mm/s]
Retraction Distance: 5 [mm]
Minimum Travel before Retract: [mm]
Minimum Extrusion before Retract: 0.02 [mm]
Z Hop: 0 [mm]
Cut off Object Bottom: 0 [mm]
Nozzle Diameter: 0 [mm or 0 = use value from "Printer Settings"]
Minimize Crossing Perimeters: Always

The slicer also uses parameters set in "Printer-Settings"->"Extruders"!

Multi Extruder Settings

Create Wipe and Prime Tower Create Ooze Shield
Support Extruder: Any Extruder
Retraction on Extruder Switch: 16 [mm]
Wipe and Prime Volume: 20 [mm³]
Volume Overlap: 0.05 [mm]

3D View | Temperature Curve | Cura | [Help](#)

CuraEngine Settings

[Print](#) [Filament](#) [Close](#)

BIBO y186 single extruder normal printing 160327repetier cura printing settings [Save](#) [Save as ...](#) [Delete](#) [Import](#) [Export](#)

Speed and Quality | Structures | Extrusion | G-Codes | Advanced

```

Start G-Code
End G-Code
Before Extruder Switch
After Extruder Switch

Create Default
You can add dynamic values,
that get replaced during
slicing.

Temperatures:
{TEMPO}, {TEMP1}
{BED}

Speeds:
{Z_TRAVEL_SPEED}
{TRAVEL_SPEED}

You can also add a line
only if an extruder or bed
is used. Therefore add one
of these codes at the
beginning of the line:
{IF_BED}
{IF_EXTO}

Use "Create Default" to get
a good starting script that

```

```

;Startcode BIBO printers
G90 ; absolute mode
G21 ; metric values
M82 ; Extruder in absolute mode
M107
G91
G1 Z1 F100
G90
G28
G1 Z1 F100
{IF_BED}M190 S{BED}
{IF_EXTO}M109 T0 S{TEMPO}
{IF_EXT1}M109 T1 S{TEMP1}
T0
M302
G92 E0
G28
G1 Y0 F300 E0
G92 E0
{IF_EXTO}T0
{IF_EXT1}T1
G92 E0
G1 X-15.0 Y-92.9 Z0.3 F3000.0 ; move to start-line position
G1 X15.0 Y-92.9 Z0.3 F1500.0 E3 ; draw 1st line

```

3D View | Temperature Curve | Cura | [Help](#)

CuraEngine Settings

[Print](#) [Filament](#) [Close](#)

BIBO y186 single extruder normal printing 160327repetier cura printing settings [Save](#) [Save as ...](#) [Delete](#) [Import](#) [Export](#)

Speed and Quality | Structures | Extrusion | G-Codes | Advanced

```

Start G-Code
End G-Code
Before Extruder Switch
After Extruder Switch

Create Default
You can add dynamic values,
that get replaced during
slicing.

Temperatures:
{TEMPO}, {TEMP1}
{BED}

Speeds:
{Z_TRAVEL_SPEED}
{TRAVEL_SPEED}

You can also add a line
only if an extruder or bed
is used. Therefore add one
of these codes at the
beginning of the line:
{IF_BED}
{IF_EXTO}

Use "Create Default" to get
a good starting script that

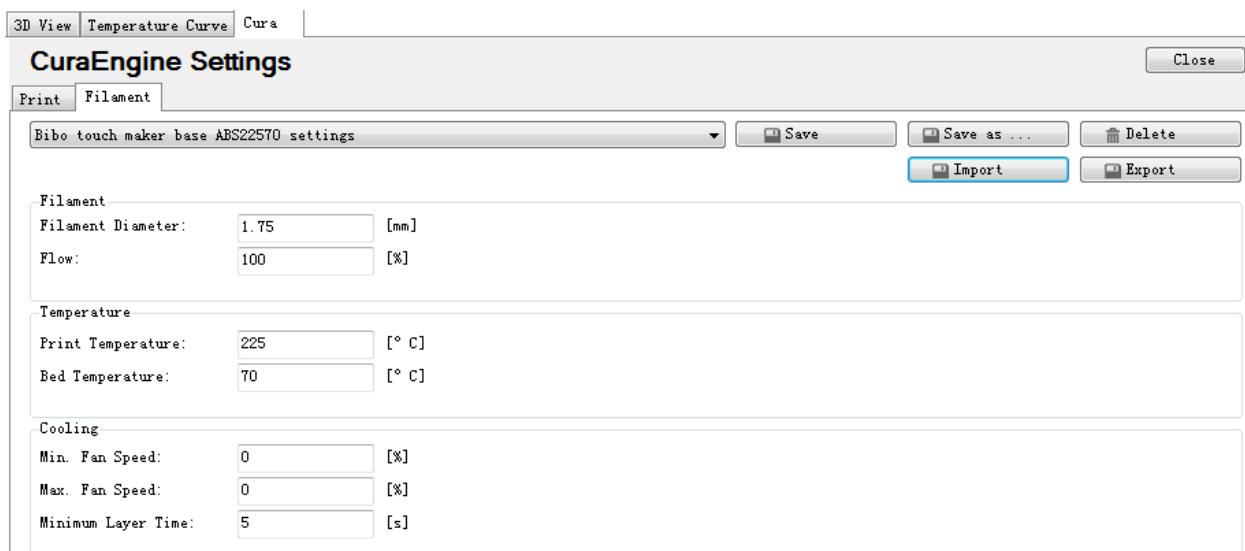
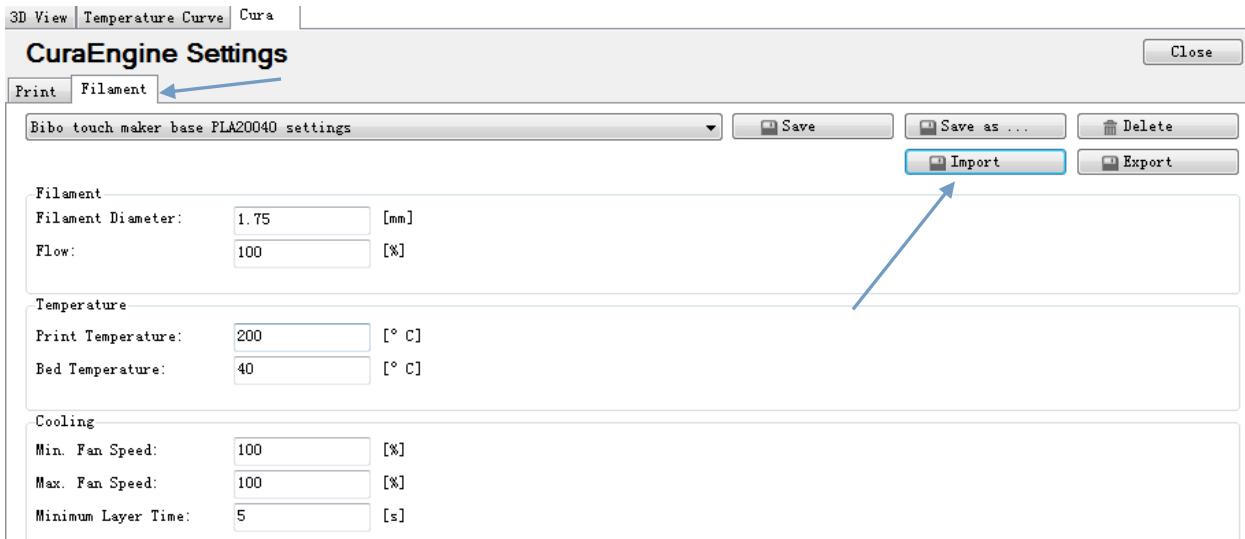
```

```

;BIBO End GCode
M107
G91 ; Relative positioning
{IF_EXTO}T0
{IF_EXTO}G1 E-1 F300; Reduce filament pressure
M104 T0 S0
{IF_EXT1}T1
{IF_EXT1}G1 E-1 F300; Reduce filament pressure
M104 T1 S0
G1 X-20 Y-20 F3000
G28 X0 Y0
G90 ; Absolute positioning
G92 E0 ; Reset extruder position
M140 S0 ; Disable heated bed
M84 ; Turn steppers off
M117 BIBO Print complete

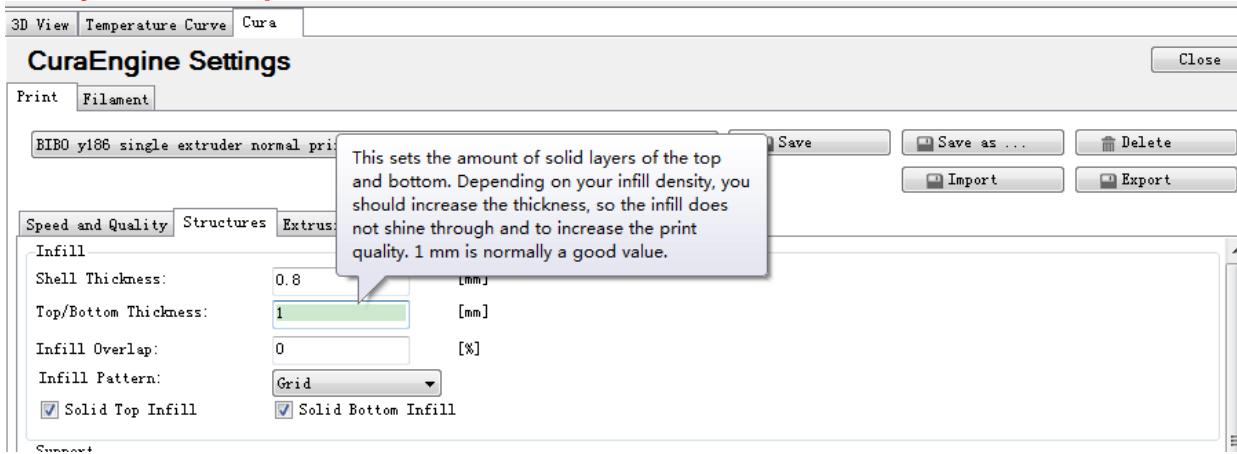
```

Meanwhile, choose the Filament tab. Click Import button, then import the Bibo touch maker base PLA20040 (maybe 20030, depends on filament) settings.rcf. from SD card's "BIBO maker and touch repetier cura settings" folder. You will see our printing settings as the photo below:



Repetier-host is an open-source software, which has better function in windows system than Mac, as most softwares. So we have cura, simplify 3D, printrun and makerware for optional choose. Just let the machine setting and slicer settings to be simillar to repetier-cura settings, and your printing will be successful(slicer setting is very important for printings). Cura settings for BIBO have been saved in SD card's "BIBO maker touch cura settings" folder, please kindly check them.

The settings we supply for you is just for reference. Different filament, different structure of printing object or even different room temperature, will cause different printing quality. So you can also set your filament settings or Print settings according to different conditions. **For every setting meaning, you can click the frame and repetier-host will show what does this setting mean. Please kindly check the photo below:**



Usually, with our glue on the heating bed, bed temperature 70 degree Celsius is ok for printing **ABS**(let nozzle cooling fan be off), and room temperature is ok for **PLA**(let nozzle cooling fan be full on).

Printing ABS, please apply more glue on the bed, with acrylic cover and door on. Printing temperature for ABS usually is 230 degree, but depends on filament supplier's advice. ABS is more difficult to print as it will warp especially for big size object. Usually you can choose to print object with **raft** as support type in settings in cura or repetier host. Printing speed not faster than 50 mm /s is better for quality.

Printing PLA, there **will be more risk to have the extruder clogged**. As PLA will expand when heated in the first time, but it will be hard to be melt again even you use higher temperature. So sometimes it expand forever and let the extruder be clogged. That's why we recommend you not to let filament stay in the hot extruder long time if not printing. Printing speed not faster than 80 mm /s is better for quality. **Printing PLA not more than 200 degree, usually 180-190 degree is the best**. Setting a printing temperature above 230-245degree for PLA is starting to get into bad territory as the plastic will start to change properties if left in the nozzle for too long and can cause clogs.

Printing travel speed(not printing speed) 100 mm/s is good for printing quality, if faster travel speed, you may need to tighten belt set screw after sometime printing.

There are two articles of printing tips in SD card, please kindly check them. BIBO 3D printer uses the same slicer software Cura as Ultimaker, and the machine

working principle is almost the same. So the two articles will be helpful to you.
Here is also attach the link for you:

<http://support.3dverkstan.se/article/23-a-visual-ultimaker-troubleshooting-guide>

<http://support.3dverkstan.se/article/30-getting-better-prints>

If the layer is thinner, the printing temperature should be lower;

If the printing speed is faster, the printing temperature should be higher and the layer should be thinner;

If there is stringing and overhang, the printing temperature should be lower;

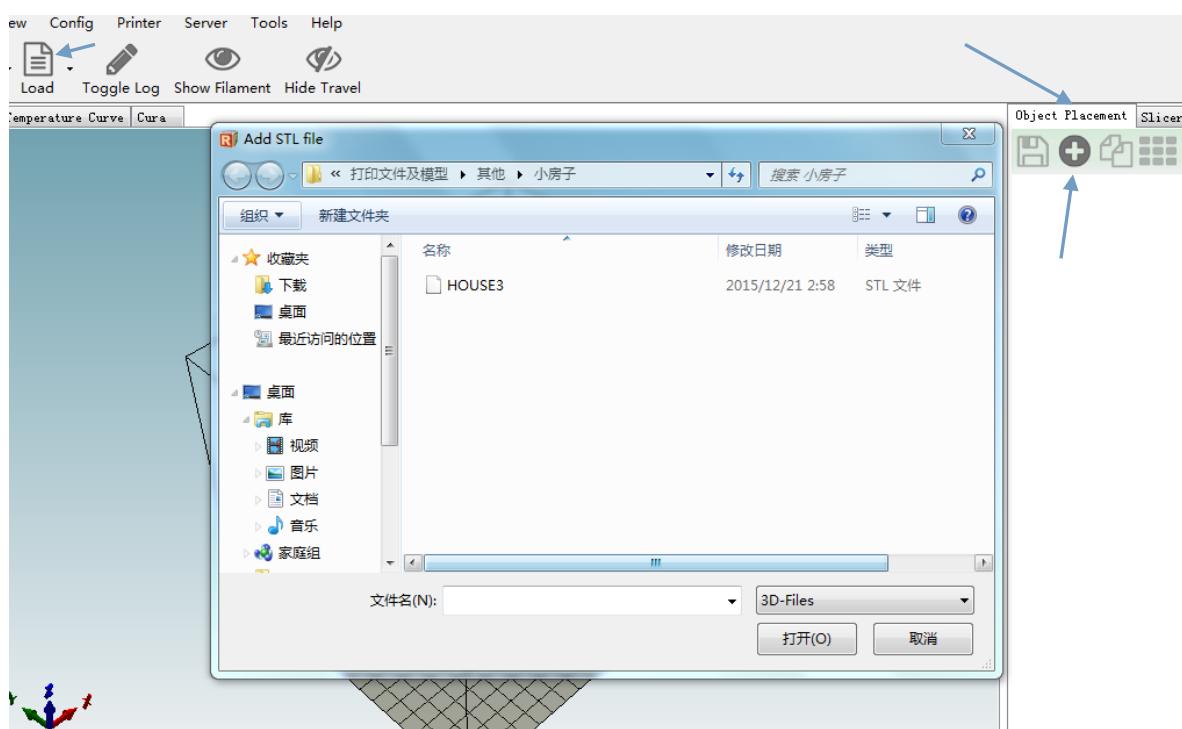
If there is warping or first layer not sticking, please use more glue and higher bed temperature, or printing nozzle a little closer to the bed;

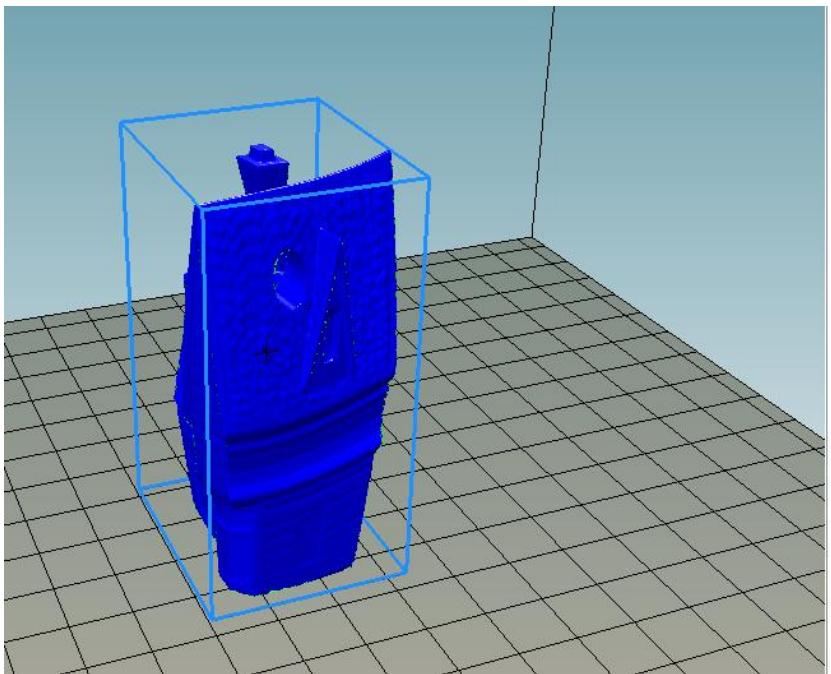
If there is shifted layers or prints are leaning, please lower the speed, or check the belt pulley is tight enough to the rod and belt in suitable tension.

Now you can load the object first. click the “Object Placement” tab, and click “Load” in the left or “+” in the right. Then choose the object you want to print.

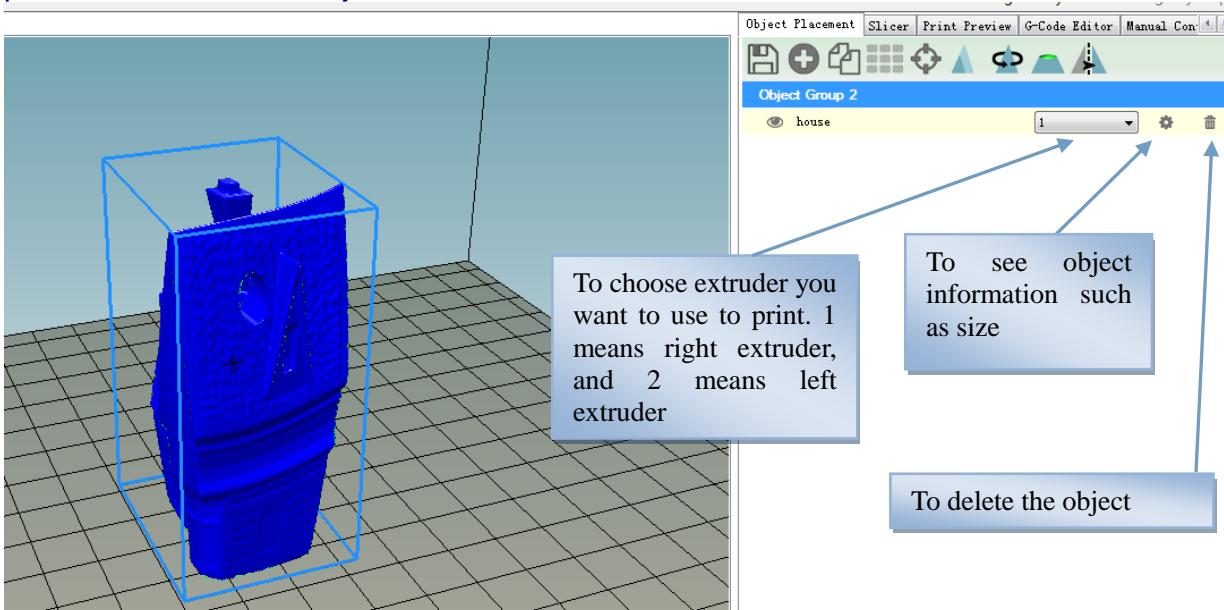
For object you want to print can be downloaded free from

<http://www.thingiverse.com/> or <https://www.youmagine.com/>

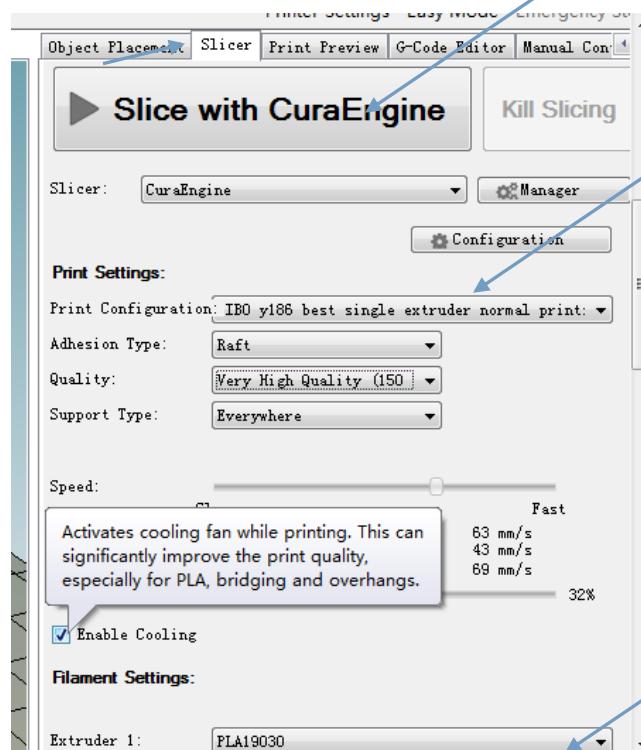




You can center the object on the bed by clicking “” icon. You can also rotate the object by “” icon. When you move the mouth arrow to every icon, repetier-host will show what this icon mean. What you want to do is let object more printable, such as center the object, less support to be printed to hold the object you want to print. Meanwhile, the object should be smaller than 214mm*186mm*160mm.

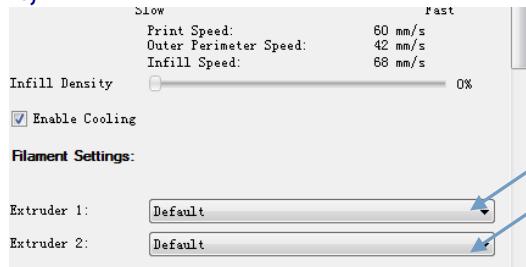


Then go to Slicer tab and choose the right settings. For every settings, you just move the mouth arrow above every settings, and the meaning will be showed. Such as you move mouth arrow above “Enable cooling”, it shows the information as below. When printing ABS, please don't choose Enable cooling.



For single extruder printing, Print Configuration should be chosen “BIBO y186 single...”. If choose “BIBO y186 dual”, the printing will be not started as it is for dual extruder printing.
Usually we choose “normal printing” especially for PC, ABS printing, PLA can be chosen “fast printing” or “very fast printing”, but the printing quality will be not better as “normal printing”

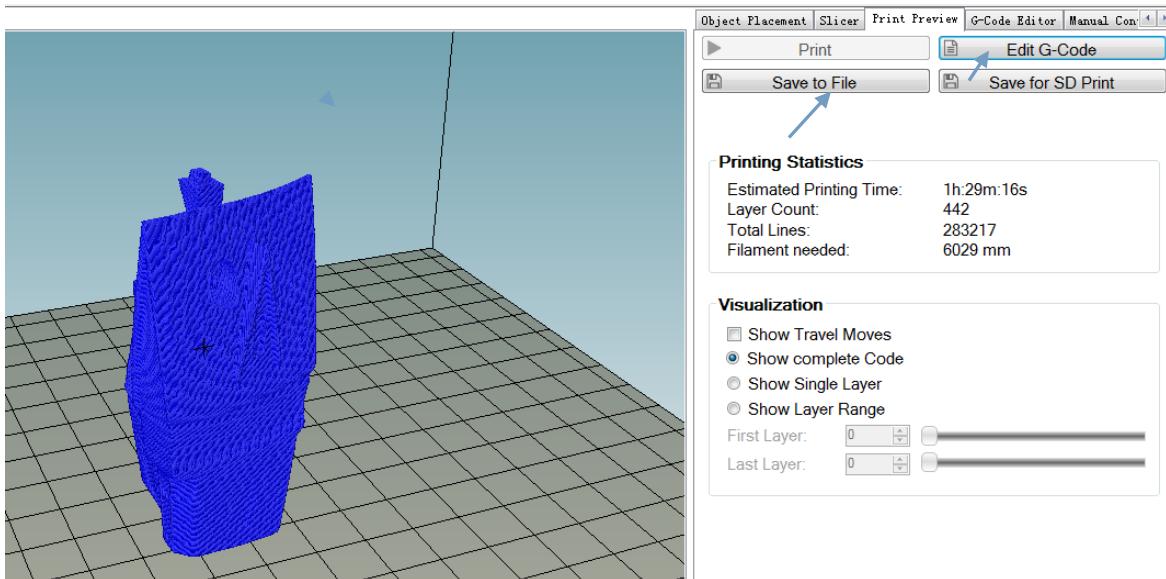
Sometimes you choose the printing temperature in repetier host, but it has a different printing temperature actually during printing. This problem is caused by repetier host's bug. **You can choose the default filament setting first and slice it,**



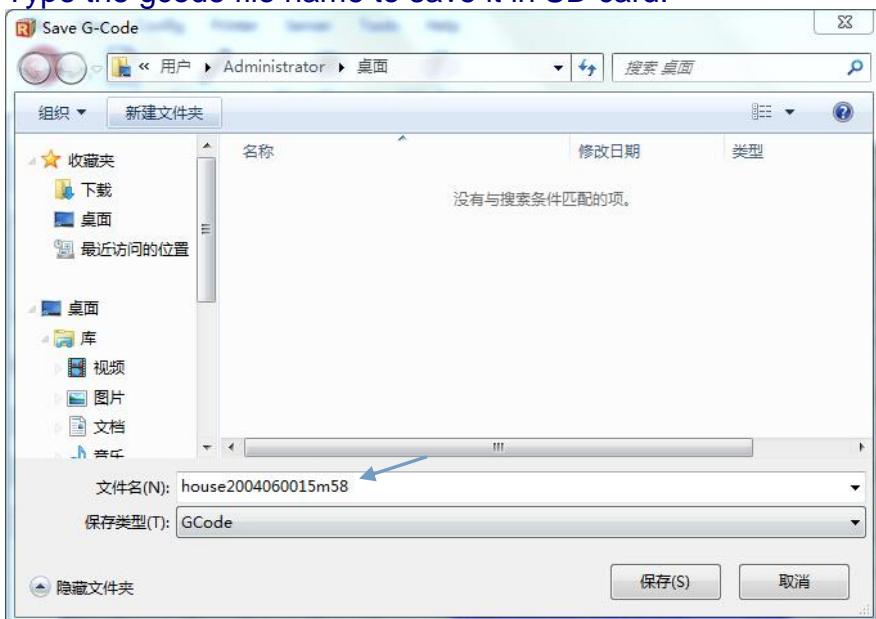
then choose the filament setting you want and slice it. Now your settings or choice will be valid for the slicing. If still not ok, just restart repetier host and it will be ok.

After finishing the settings, click “Slice with CuraEngine”.

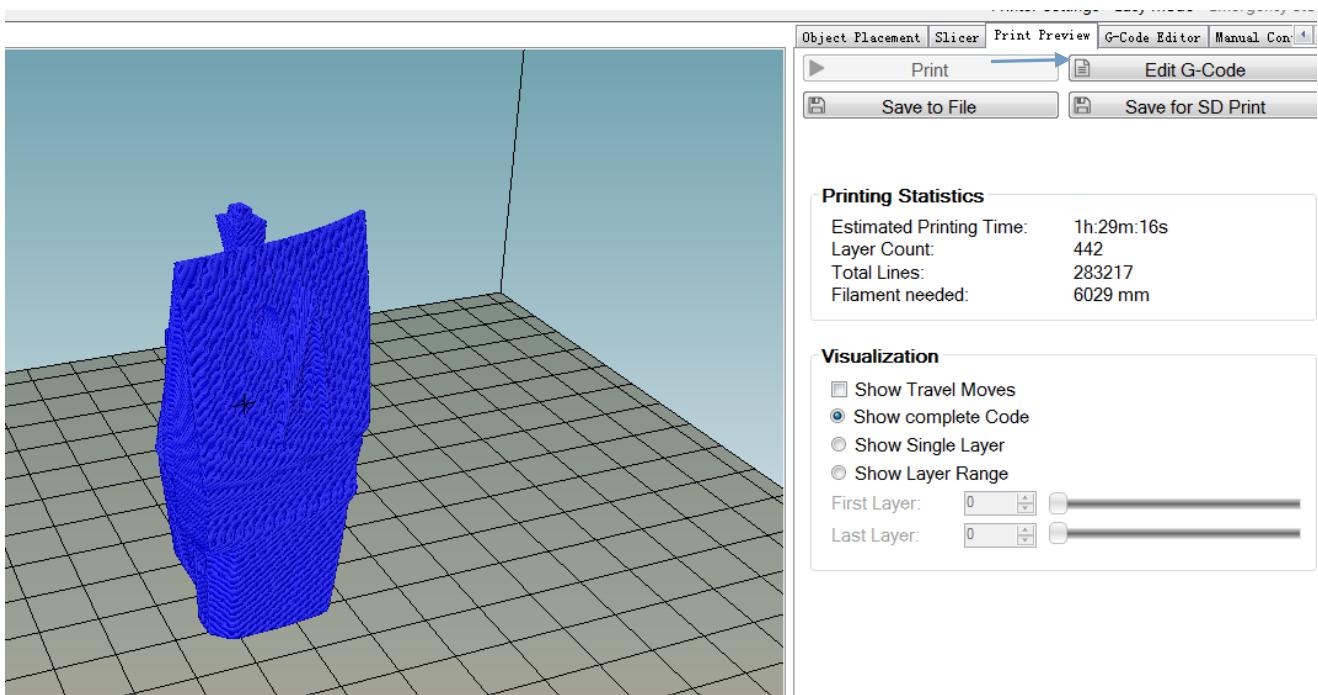
Then go to Print Preview tab, click “Save to File” and you can see as the photo below:



Type the gcode file name to save it in SD card.



Meanwhile, you can click "Edit G-Code" to edit gcode file.



In the G-Code Editor tab, you can edit or save the gcode file on PC or SD card.

Preview G-Code Editor

```

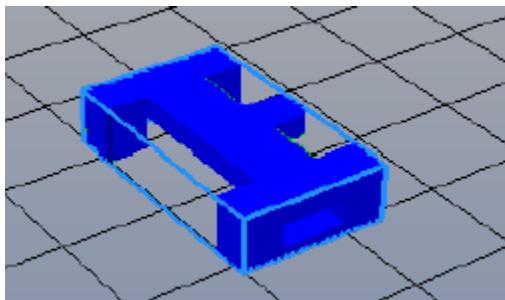
1 ;Generated with Cura_SteamEngine 14.12
2 ; Default start code
3 G28 ; Home extruder
4 G1 Z15 F100
5 M107 ; Turn off fan
6 G90 ; Absolute positioning
7 M82 ; Extruder in absolute mode
8 M190 S30
9 ; Activate all used extruder
10 M104 T0 S180
11 G92 E0 ; Reset extruder position
12 ; Wait for all used extruders to reach temperature
13 M109 T0 S180
14 ;Layer count: 24
15 ;LAYER:0
16 M107
17 G0 F3600 X98.200 Y66.200 Z0.400
18 ;TYPE:SKIRT
19 G1 F1800 X115.800 Y55.200 F1.17076

```

So now you can print the object with a SD card on printer directly, the same steps as First print in Chapter 4, and the difference is you don't have to level the bed again.

Use second extruder for support material

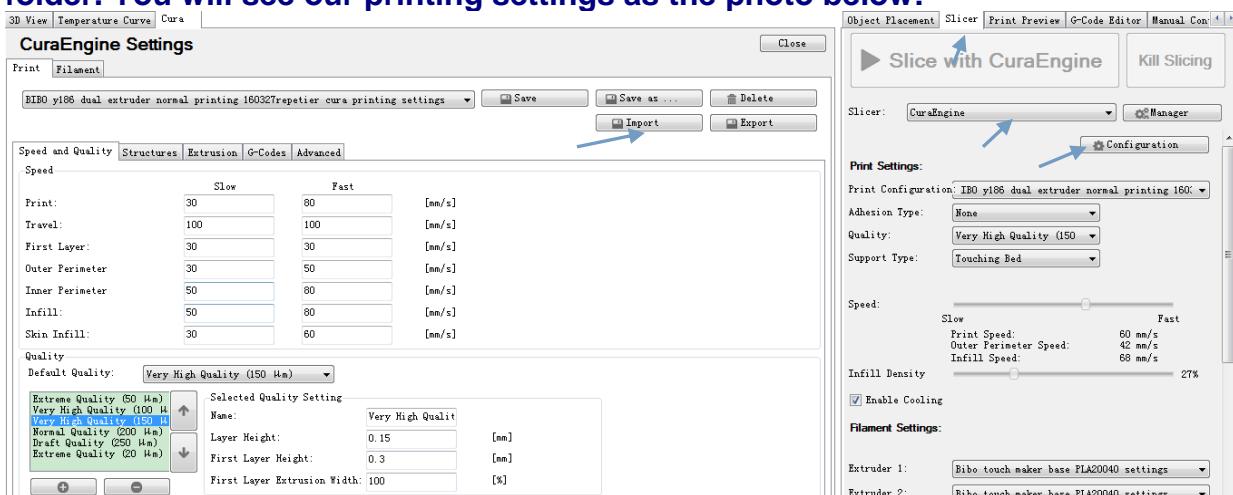
We use this object for reference. This can only be printed well with support material

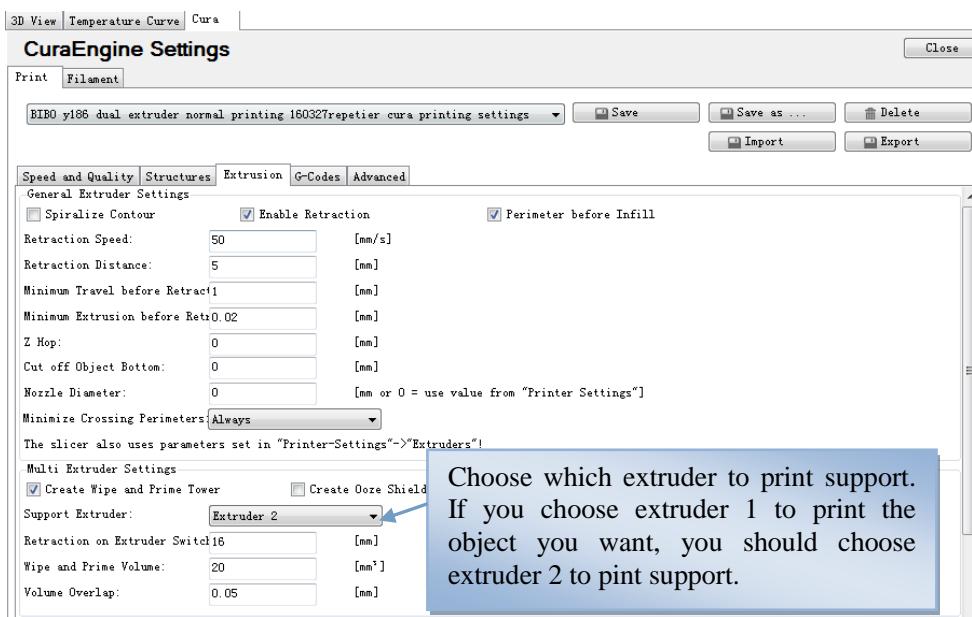


Go to the slicer tab and choose the following:

Slicer : CuraEngine

Then click Configuration, choose the Print tab. Click Import button, then import BIBO y186 **dual extruder normal** printing 160327repetier cura printing settings.rcp from SD card's "BIBO maker and touch repetier cura settings" folder. You will see our printing settings as the photo below:





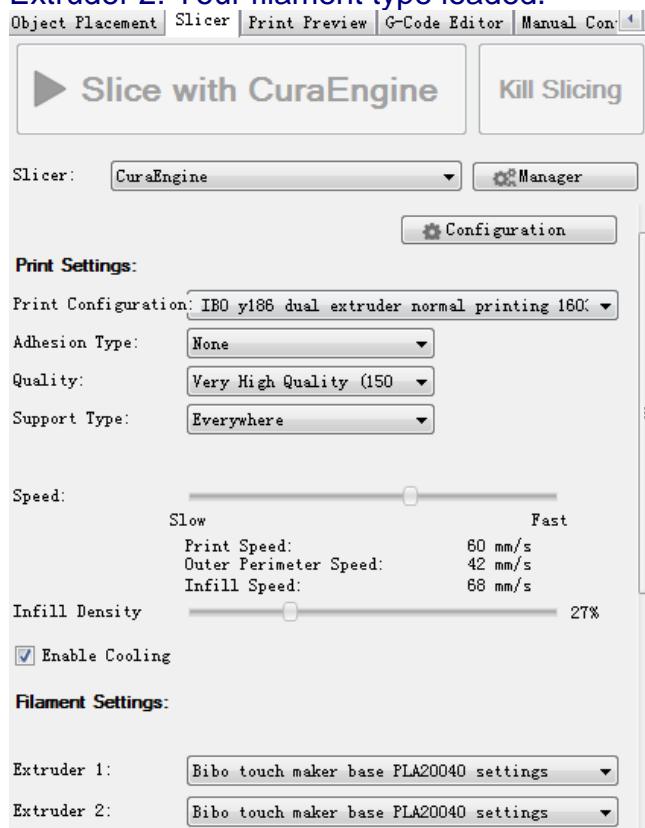
Adhesion Type: ...

Quality: ...

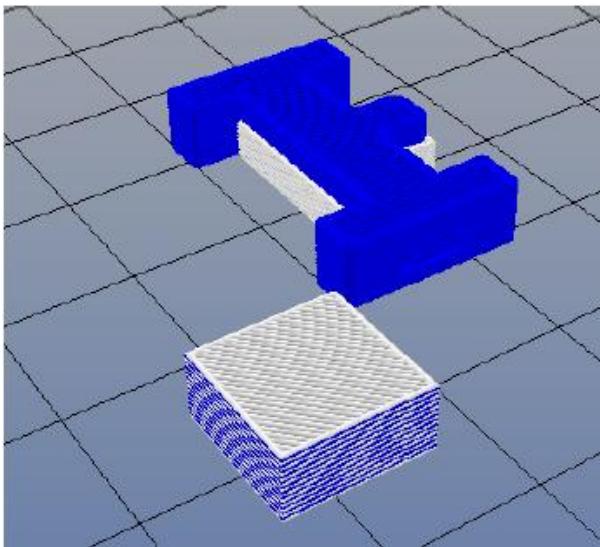
Support type: Everywhere

Extruder 1: Your filament type loaded.

Extruder 2: Your filament type loaded.



Press slice, this should be the result.

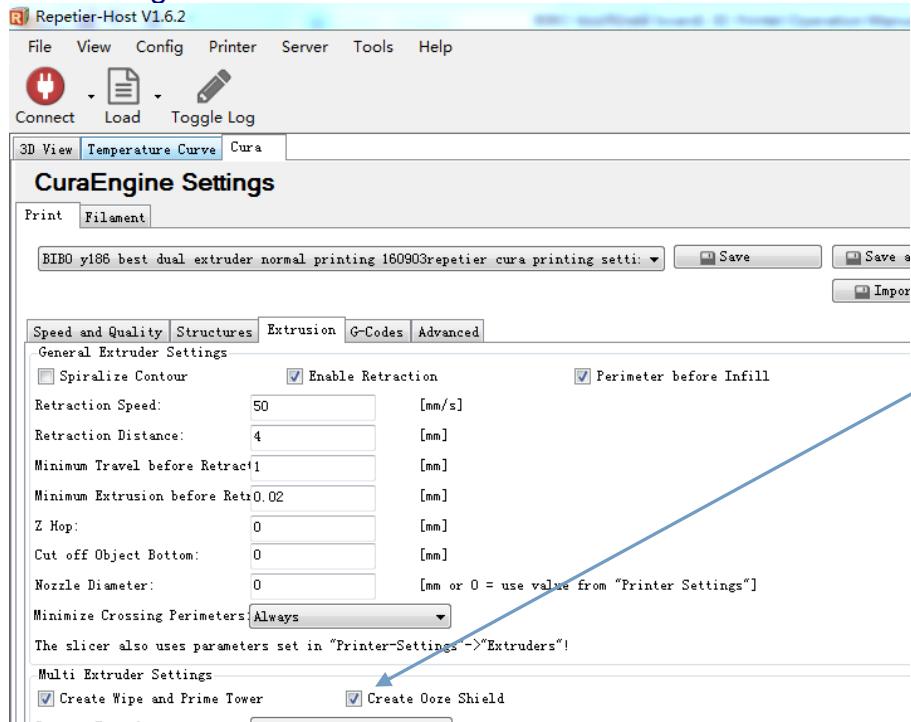


The white colored lines are the support material and the blue lines are the object itself.

Next to the object you see a square tower, this is used to clean the nozzle when switching.

Now our new slicing settings add Ooze shield for dual extrusion. It can catches any oozing from the unused nozzle in dual extrusion. Please take this shield away after printing.

The setting is here:



The ooze shield is like this:

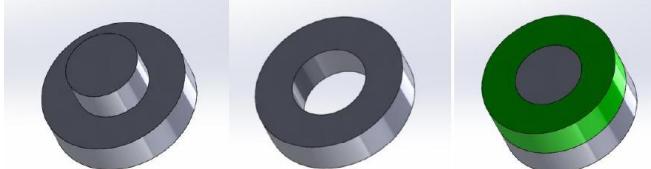


Assign an extruder to an object.

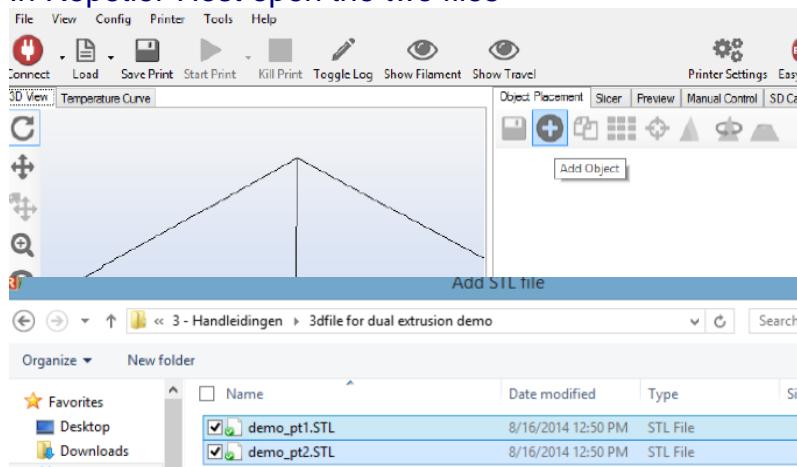
This short tutorial will show you how to **print two objects in a single print with different extruders**.

You basically need two STL files which are created in the same coordinate system and with exact needed distance with respect to each other.

For this example two simple parts are created, see picture below.

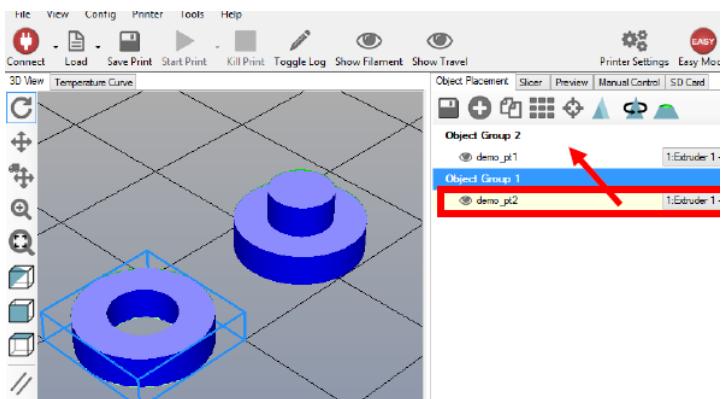


In Repetier-Host open the two files

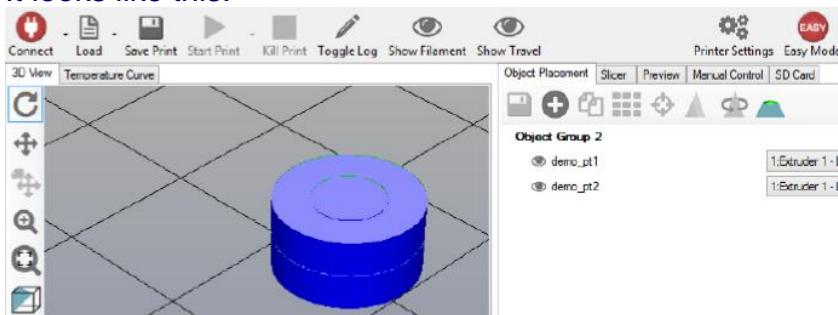


It should now look like this:

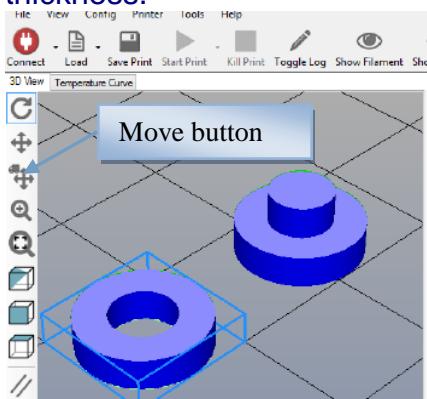
Select demo_pt2 and drag it into object group 2



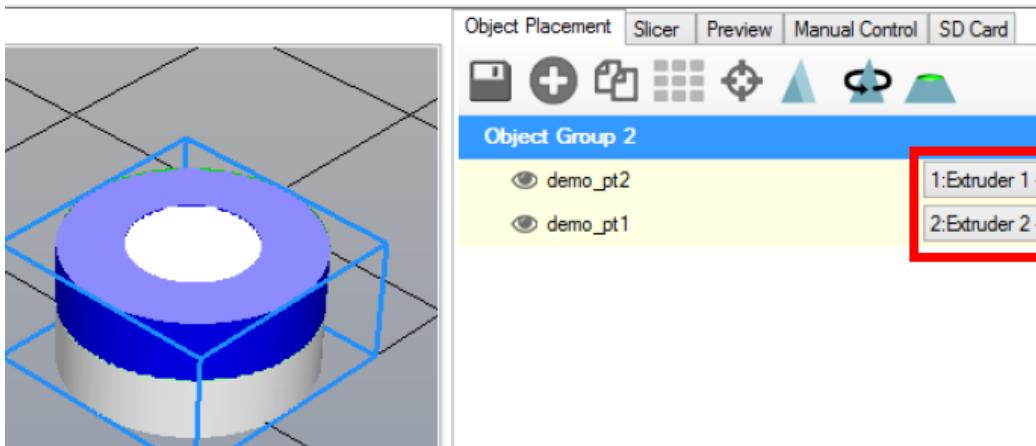
It looks like this:



If two objects are **not in the same coordinate system**. You can use **move button** to let two objects get together. First center two objects, then **select the left object and move it in Z coordinate** for right object's bottom cylinder thickness.



Select which object you want to print with which extruder.

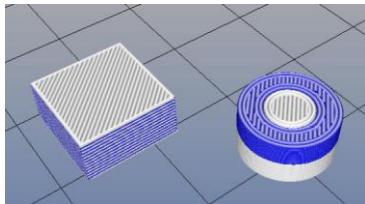


Goto the Slicer tab and select at least the following:

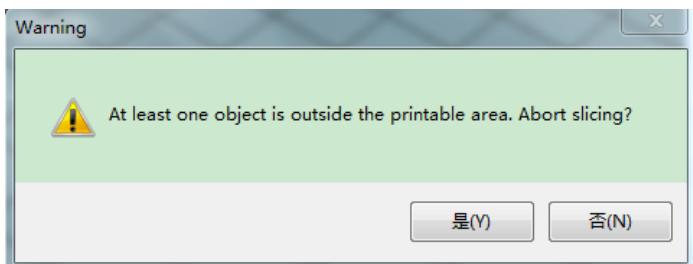
Print Configuration: BIBO dual extruder repetier cura printing settings

Then press the Slice button. The result should look something like this:

The left is a square used for purging when switching from one extruder to the other.



When you use the second way to assign the extruder by moving coordinate, you click slice, there will be window below:



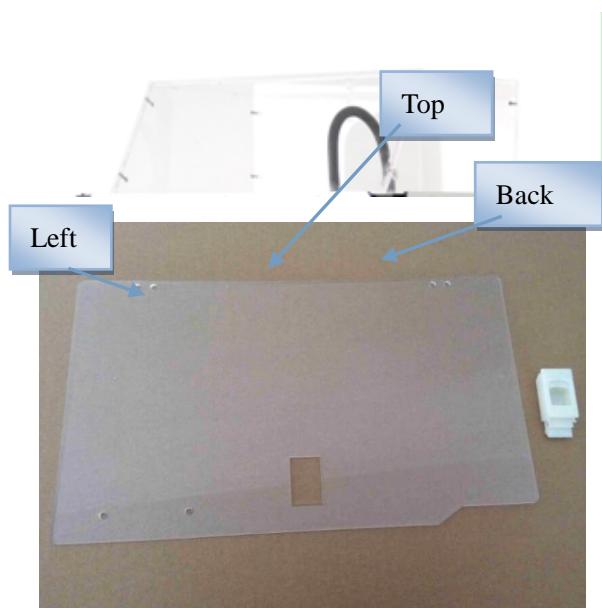
You choose no will be ok.

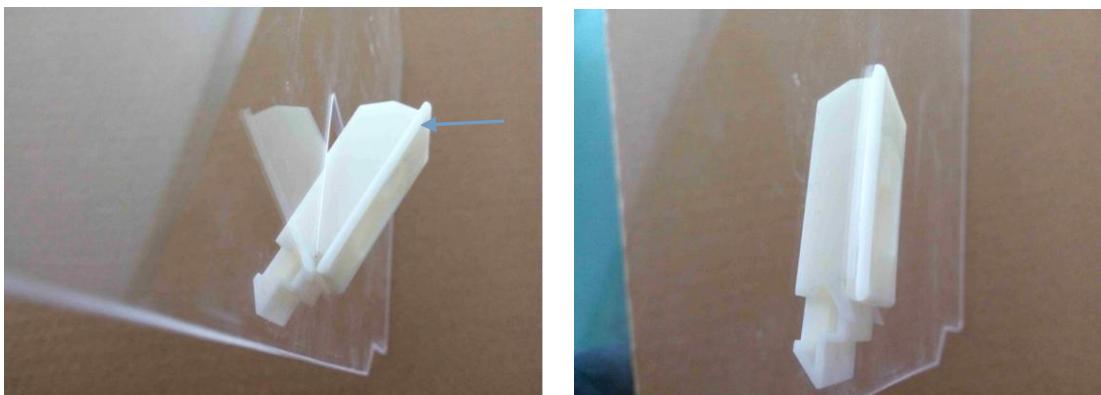
8. Acrylic Cover Installation

Acrylic cover is necessary for printing ABS, PC, Nylon...

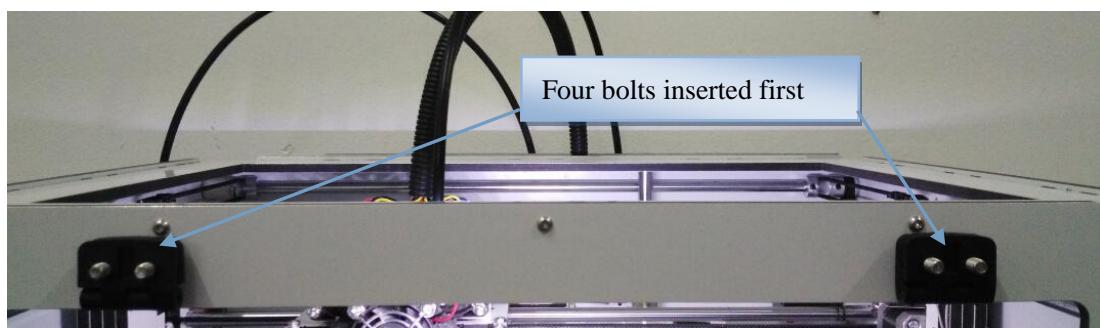
We also have finished **acrylic cover assembly video**, and you can copy it from the SD card.

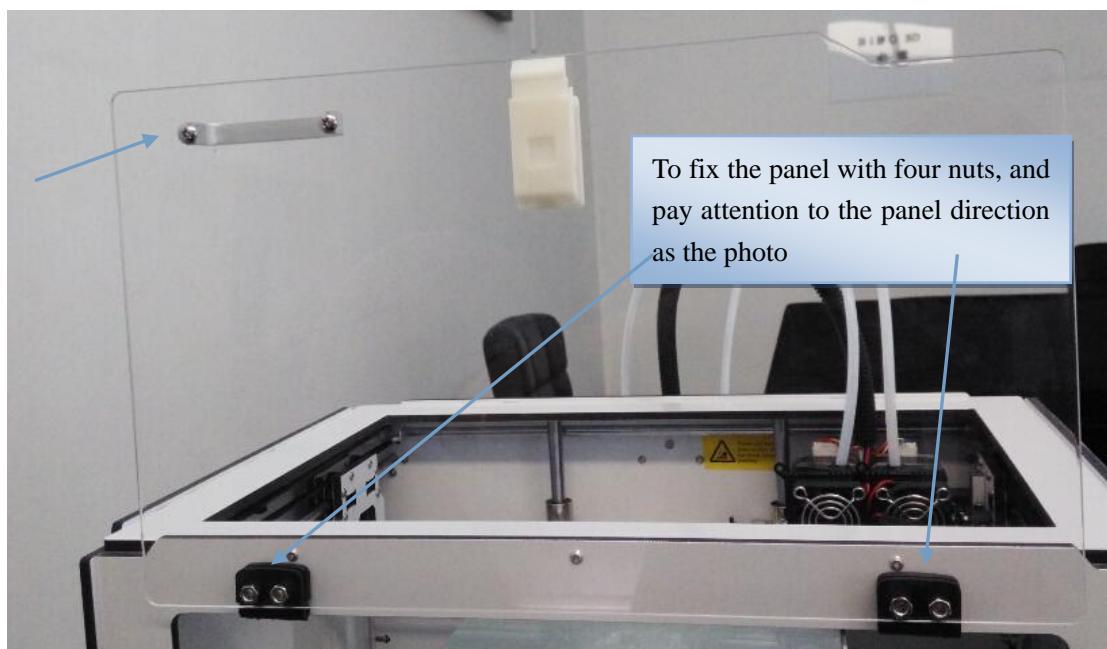
The assemble order is back side, left side, right side, top side and front side in the end. Please screw bolts not tight first, after installed all five panels , you can finally tighten the bolts, but still carefully not too tight (acrylic panel is very fragile). Assemble the acrylic cover as the photo below.

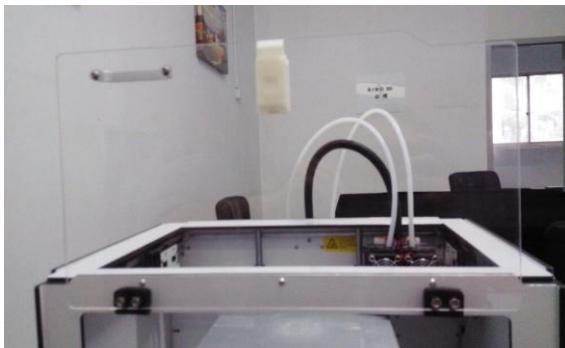




To fix this white clamp should be more carefully. Acrylic sheet is very fragile and will be broken.







Acrylic door down to cover

Acrylic door up to open

9. Function Introduction

Copy printing means that you can let two extruders printing at the same time for two same objects, to cut the printing time at least in half. **The object size should be smaller than 33mm in X axis direction.** When you generate a single extruder printing gcode file, just open this file by repetier-host or notepad.

For example, you generate an **extruder 1(right extruder) 's printing gcode file**, you should add

M420 S1

M109 T1 S190

In the first and second line in the gcode file.

If you generate an **extruder 2(Left extruder) 's printing gcode file**, you should add

M420 S1

M109 T0 S190

In the first and second line in the gcode file.

T0 means extruder1(right extruder), T1 means extruder2(left extruder).

S190 means the copy printing extruder's printing temperature is 190 degree. If you want it be 200 degree, you just revise it be S200.

The adding photo is below:

```
copylcm cube - 记事本
文件(F) 编辑(E) 格式(O) 查看(V) 帮助(H)
M420 S1
M109 T0 S190
;Generated with Cura_SteamEngine 15.01
;Startcode BIBO printers
G90 ; absolute mode
G21 ; metric values
M82 ; Extruder in absolute mode
M107
G91
G1 Z1 F100
```

Meanwhile, you should add

M420 S0

In the last line of gcode file to disable this function.

```

new1dcm cube19045 - 记事本
文件(F) 编辑(E) 格式(O) 查看(V) 帮助(H)
G1 F3120 X4.199 Y2.305 E134.84725
G0 F6000 X4.199 Y2.871
G1 F3120 X2.871 Y4.199 E134.92533
G0 F6000 X3.437 Y4.199
G1 F3120 X4.199 Y3.437 E134.97013
G0 F6000 X4.199 Y4.002
G1 F3120 X4.002 Y4.199 E134.98171
M107
G1 F3000 E129.98171
G1 Z5.250
G0 F6000 X4.002 Y4.199 Z10.000
;BIBO End GCode
M107
G91 ; Relative positioning
M104 T0 S0
T1
G1 E-1 F300; Reduce filament pressure
M104 T1 S0
G1 X-20 Y-20 F3000
G28 X0 Y0
G90 ; Absolute positioning
G92 E0 ; Reset extruder position
M140 S0 ; Disable heated bed
M84 ; Turn steppers off
M117 BIBO Print complete
M420 S0

```

When you stopped the copy printing and want to do other printing (not copy printing, just single or dual extruder printing), you should power off the printer after stopped the copy printing, then power on the printer to start a new printing. This is because copy printing function will be still valid and will let two extruder extrude at the same time after you stopped the copy printing, causing extruder have click noise because of low temperature printing. This is also why we add M420 S0 in the end of gcode file to let copy printing function be not valid.

Continue printing after power cut means if the printer power is cut during printing(should be more than 50 G-code executed after nozzle and bed heating), when the power is back to the printer, the printer will show you the interface to continue the last printing. If you just start the printing, maybe heating or just print a little seconds, when you turned off the power, the function of continue printing after power cut will not be valid. The reason is mentioned in the first (since 50 G-code executed after nozzle and bed heating) . Meanwhile, this function is only valid when printing from SD card.

Filament run-out detection means Printing will be paused automatically after filament run out, ensuring your designs come out intact every time.

If you want to receive the printing status message from the printer, please kindly check the link below:

<http://www.repetier.com/documentation/repetier-host/repetier-informer-app/>

If you want to control the 3D printer from everywhere, you can also use software Teamviewer to control your 3D printer by connecting 3D printer to PC (details are in the next chapter). This software can be downloaded on <https://www.teamviewer.com>

For webcam(optional function), there is another manual with the machine, please kindly check it.

10. Driver Installation (not necessary)

If you want to print by connecting the computer with USB cable(not recommended, printing from SD card is more stable especially for long time printing), you have to install the driver on the computer for the printer.

1. Plug in the power supply cable.
2. Plug the USB cable into the computer
3. Your operating system should find the correct drivers automatically, or download them automatically with Windows update.

If the drivers are not found automatically, then copy the driver from the SD card sent to you or download drivers from here:

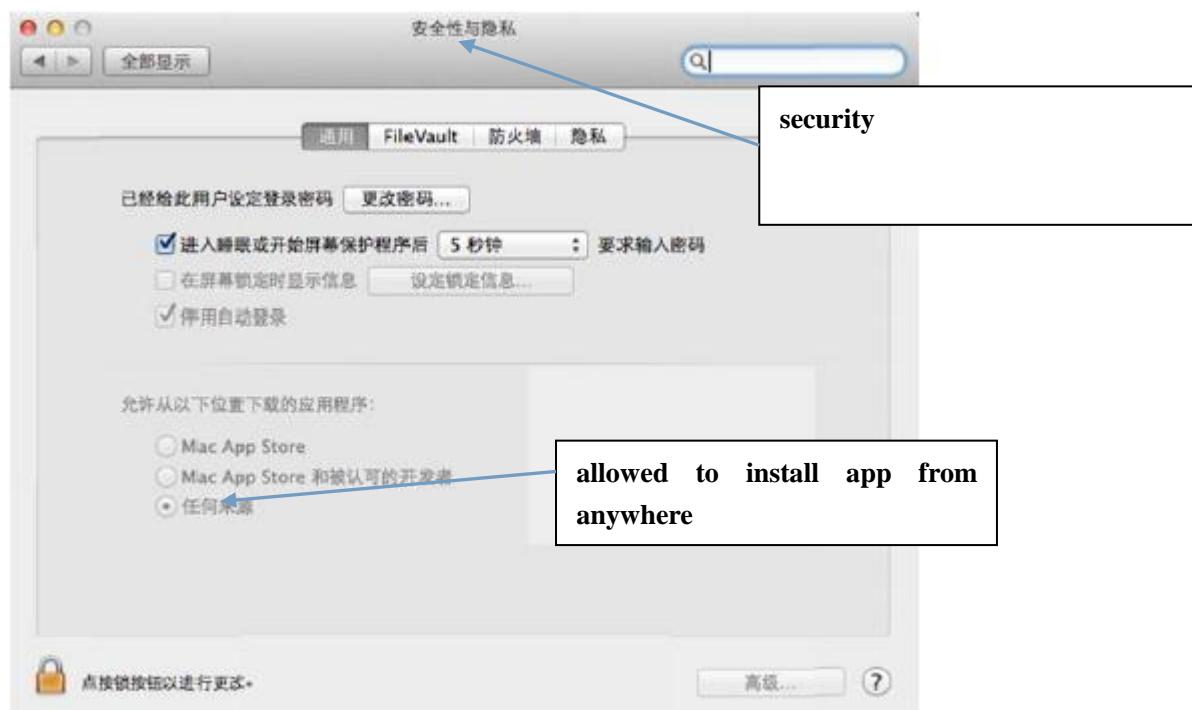
http://wch.cn/download/CH341SER_EXE.html (for windows)

http://wch.cn/download/CH341SER_LINUX_ZIP.html (for Linux)

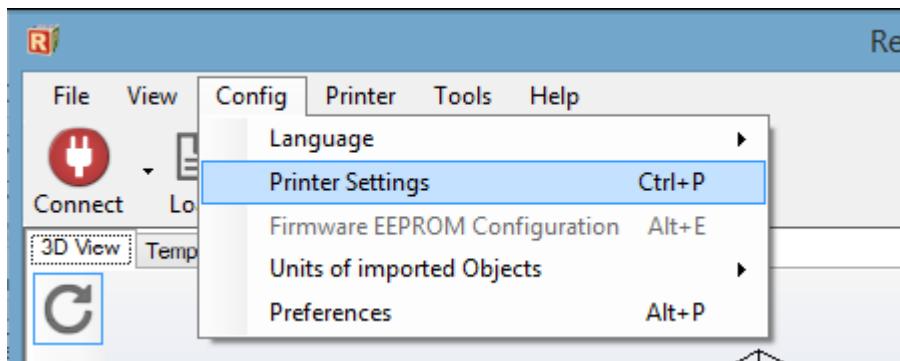
http://wch.cn/download/CH341SER_MAC_ZIP.html (for Mac)

Then install it.

4. For Mac system, if the system is over OSX 10.8, please lower the security level of the system, allowed to install app from anywhere before installation, or the driver can't be installed. Please check the photo as below:



5.After installation, open Repetier-Host. Go to Config -> Printer settings.
After installation, open Repetier-Host. Go to Config -> Printer settings.



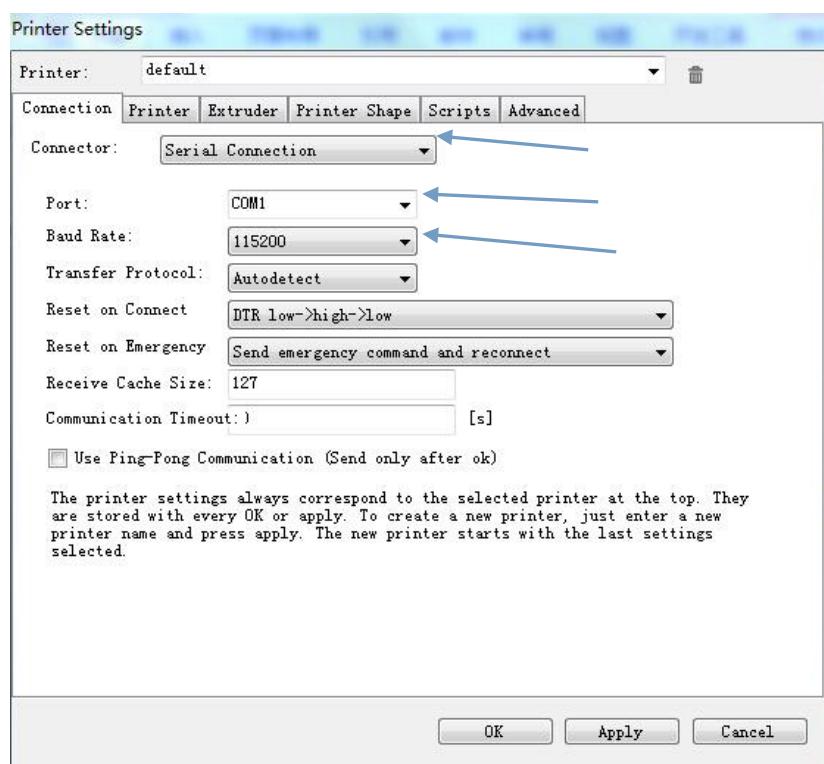
6. Once you see the printer settings window, please fill out the parameter as photo below:

PS. Connector should be chosen “serial connection”

Choose the COM-port which belongs to your printer. If no port shows up, it could because of the following:

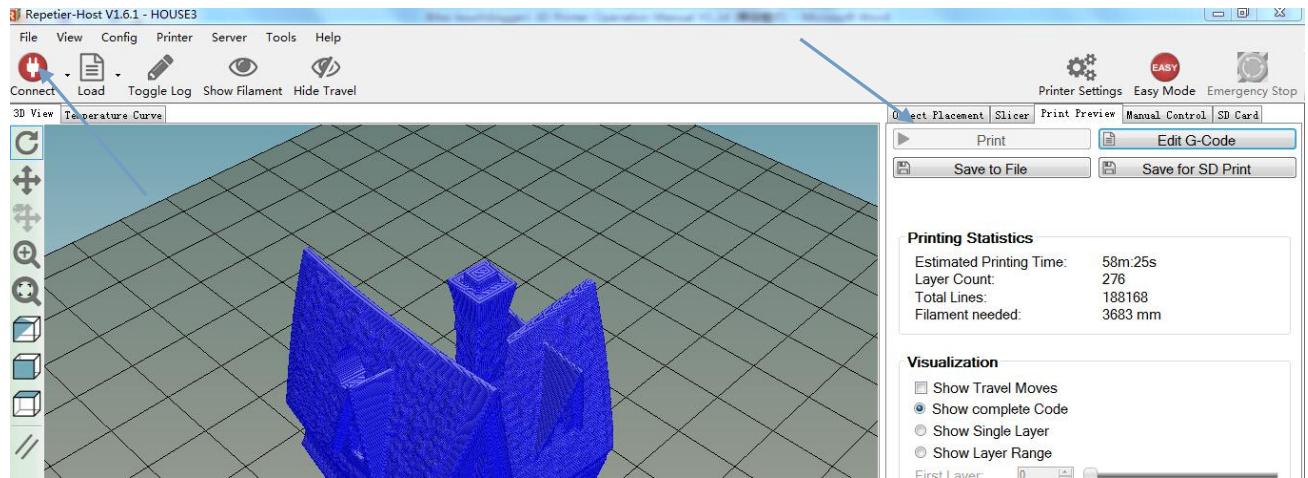
- your printer might not be plugged into the USB port or no power supplied.
- The driver is not installed, or currently installing by windows.

If this is not the case, please install the driver manually again.



NOTE: what COM-port is present for your printer after installation. If there are more than one COM ports available, unplug the USB cable of the printer and then re-plug it in again. Check what port number is appearing and disappearing. This port number is the port present.

7. Click “Connect”, then this button will be turned to green, it means connected successfully. Then click “Print”(you already sliced the object), and the printing will go.



11. WIFI (optional function)

The BIBO printer has the optional function WIFI in AP mode or Client(STA) mode. Client mode is easier. The WIFI module can work on normal 2G wifi in client mode, and 5G WIFI can't be connected. You can use AP mode instead.

Wifi in AP mode (BIBO 3d printer will be a router to send wifi signals)

1. The WIFI mode selection :

To use WIFI in AP mode, please copy the file mks_config.txt in SD card's "**WIFI configuration file for touch**" folder to your PC and save it. Then open this file, Change the value after the colon of "cfg_wifi_mode" into the Wifi network you need ("1" is AP mode, "0" is client (STA) mode.

AP mode as a router to generate the wifi signal, client mode to receive and connect the wifi signal nearby. If in AP mode, please let the wifi name be different from the nearby wifi's name.) .

Choose 1 for AP mode

2. WIFI configuration instruction on AP module

A. WIFI name: change the WIFI name after the colon of the field

“cfg_wifi_ap_name” into the hot name you need . For example, i need the wifi name to be 2166

B. WIFI password: change the WIFI password after the colon of the field

“cfg_wifi_key_code” into password of WIFI hotspot, and save it. For example,

i need the wifi password to be jinjinkeji

3. Now save the file and copy **mks_config.txt** file directly to the SD card and insert the SD card to the printer. Power on the printer(this step is very important, should insert the SD card first then power on the printer), and after a short while of touch screen booting, the wifi configuration will be valid(there will be a file named config_cur in SD card).

4. Press "WIFI" button on settings menu by touch screen, waiting WIFI name and KEY to be turned to yours, and the state be connected. Sometimes this will take about 2 or 3 minutes (you can touch back and then touch WIFI to refresh the wifi state).



5. Let your PC or smart phone connect the wifi name you just wrote for your printer. For example, i choose to connect the wifi "2166" and type the wifi password jinjinkeji, then my PC connect the printer's WIFI .

6. Use Printrun to control printer through WIFI

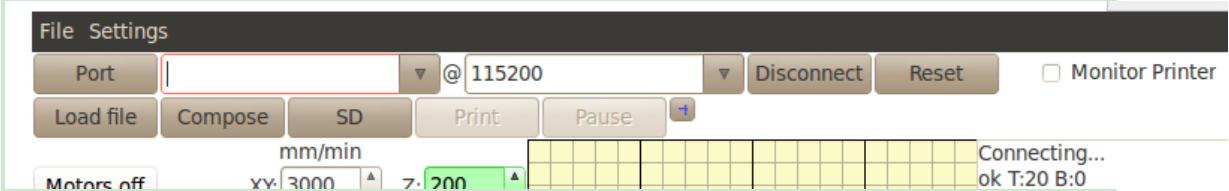
For Windows, Linux and Mac users, please kindly visit

<http://reprap.org/wiki/Printrun>

and download the printrun.

Type wifi IP:8080 at port (for example, 192.168.3.100:8080)

Baudrate should be 115200, then connect



Wifi in client mode (BIBO 3d printer can connect your wifi router)

In client mode, the operation is almost the same as AP mode. But there are

Tel:+86 575 88918283

Url: www.ourbibo.com

Fax:+86 575 88489635

E-Mail: bibo@ourbibo.com

Mobile:+86 133 25759139 50

Skype: [bradenking0202](#)

two difference as below:

A. In mks_config.txt file, revise the "cfg_wifi_mode" to be 0 , >cfg_wifi_ap_name:216 to be your wifi name (your wifi name to replace 216). Also revise >cfg_wifi_key_code:jinjinkeji to be your own wifi password(your wifi password to replace jinjinkeji). Now save the file and copy it to the SD card and insert the SD card to the printer. Power on the printer(this step is very important, should insert the SD card first then power on the printer), and after a short while of touch screen booting, the wifi configuration will be valid(there will be a file named config_cur in SD card).

(If you want fixed ip for wifi unit, please kindly revise the >cfg_ip_dhcp_flag:1 to be >cfg_ip_dhcp_flag:0 in mks_config.txt , also type your gate, mask and ip address you want, then save it.)

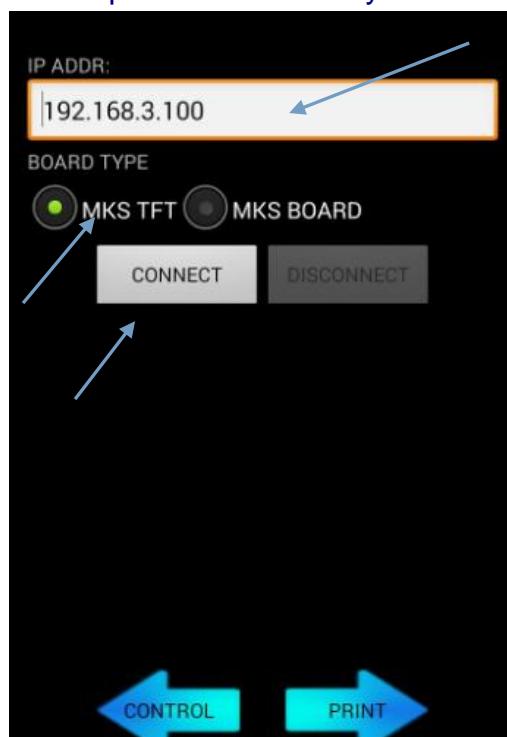
B. There is no need to let your PC to connect the wifi of BIBO 3D printer, just let your PC or smart phone connect the wifi which is BIBO 3D printer also connected.

Use APP to control printer through WIFI (Android or IOS)

1. Android

There is an APP install file in the SD card's "**Touch Wifi APP for Android**" folder for Android phone users. Please kindly install it first on your phone.

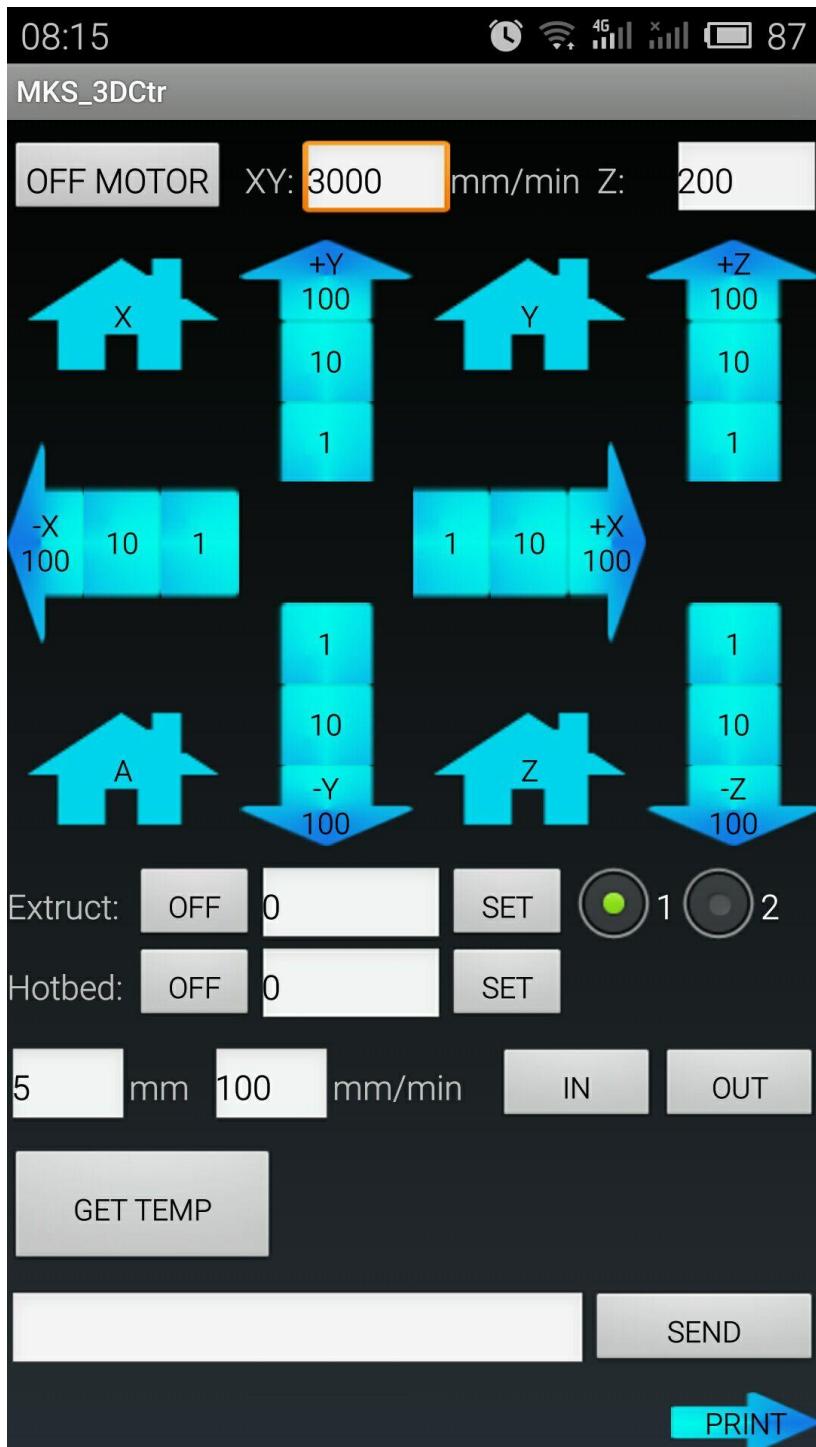
Then open this APP and you will see:



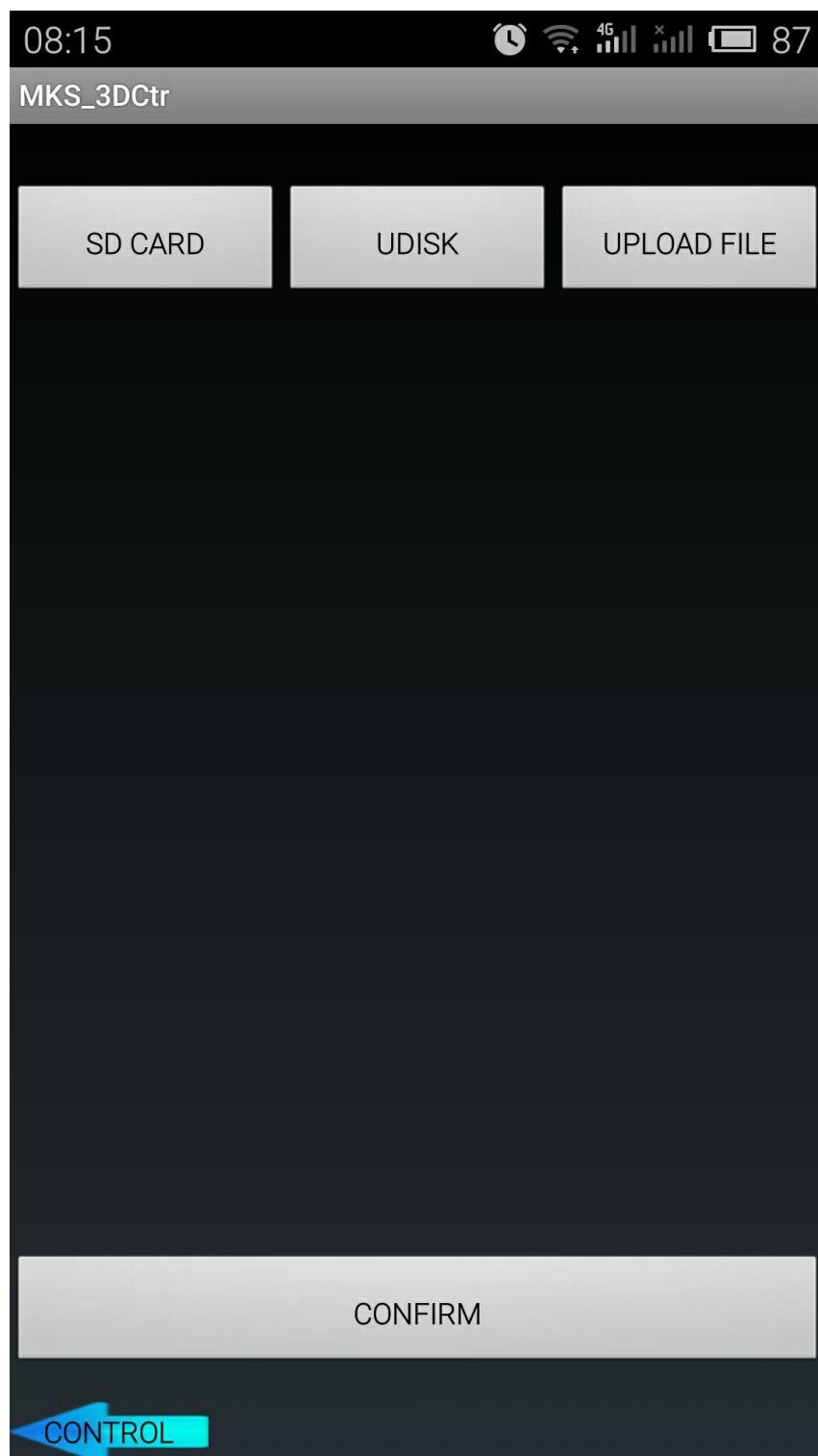
Type the **WIFI IP showed on your touch**

screen, choose MKS TFT board type and click connect button.(If in WIFI AP mode, please let your smart phone connect the BIBO 3D printer's wifi first, to ensure that your phone and the printer are in the same LAN)

For control the printer, please choose left button "control" and you will see:



For Printing, please choose the button such as “SD CARD” and choose the gcode file to start the print.



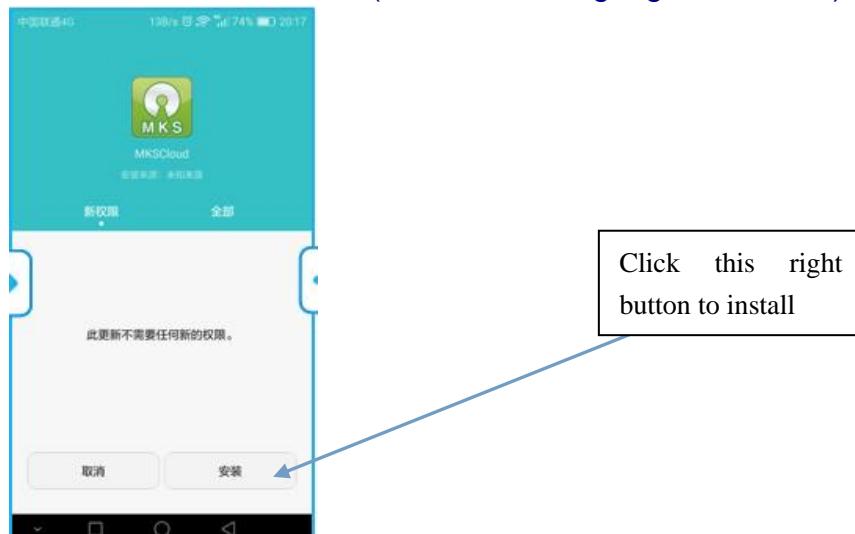
2. IOS

Please search mkscloud in APP Store or scan the QR code below:



(Android4.0.3/IOS8 or newer)

Now install the mkscloud (The default language is chinese)

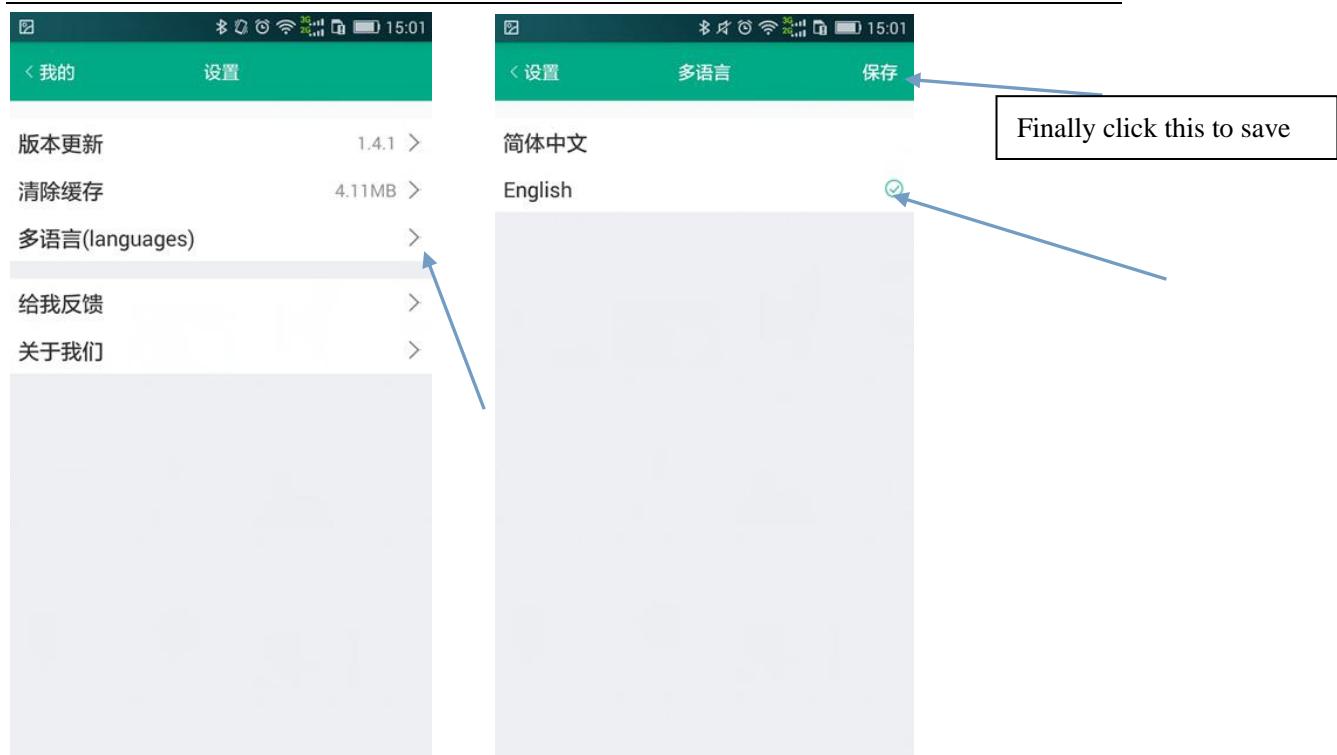


Set the language to be English (click the buttons as below)

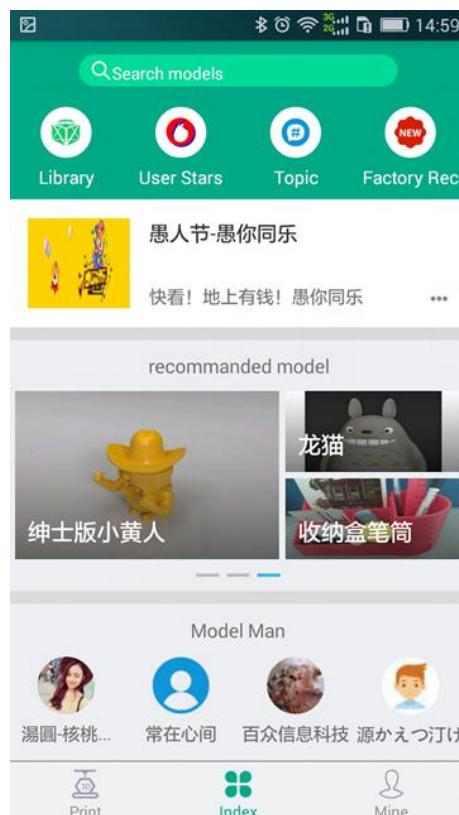




Shaoxing Bibo Automatic Equipment Co., Ltd.



Now mkscloud is in English. The APP can't be registered without Chinese phone number, but you can still use the APP smoothly.

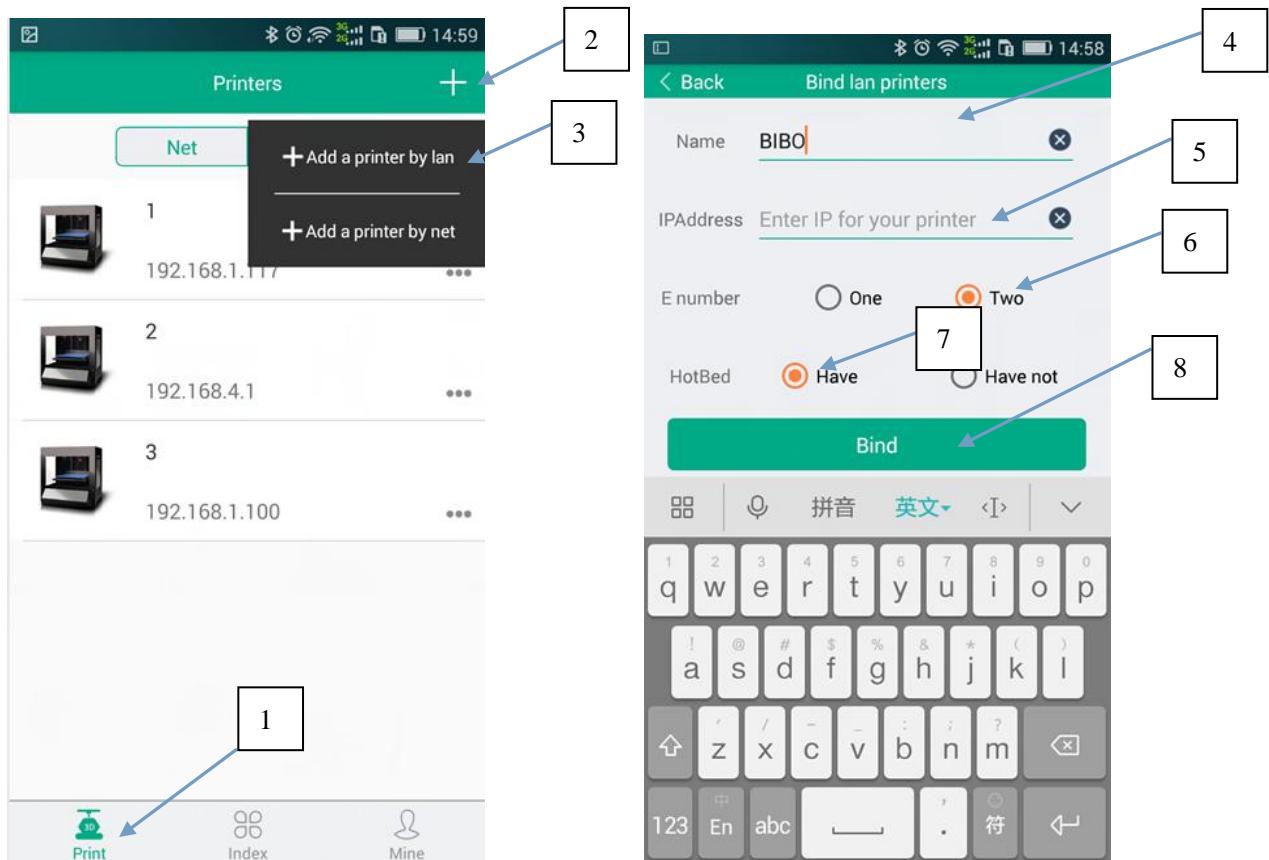


Bind the printer in LAN (Net is not open yet except in China)
Type the **WIFI IP showed on your touch screen**.(If WIFI is in AP mode,
please let your smart phone connect the BIBO 3D printer's wifi first, to ensure
that your phone and the printer are in the same LAN)

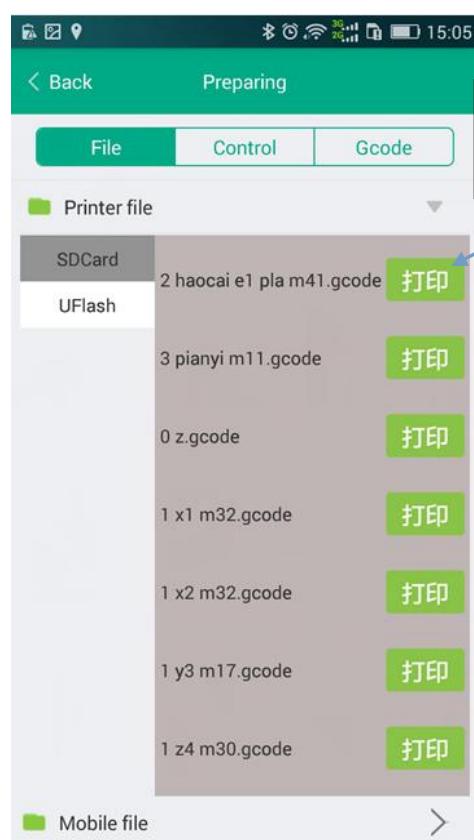
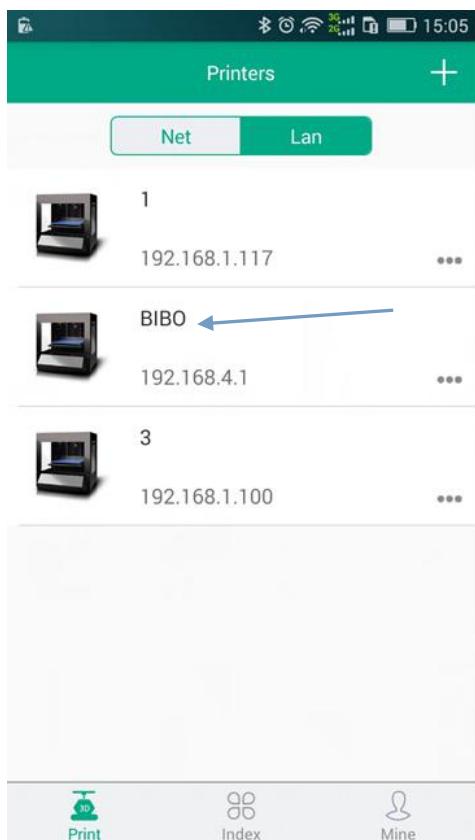
Tel:+86 575 88918283
Url: www.ourbibo.com

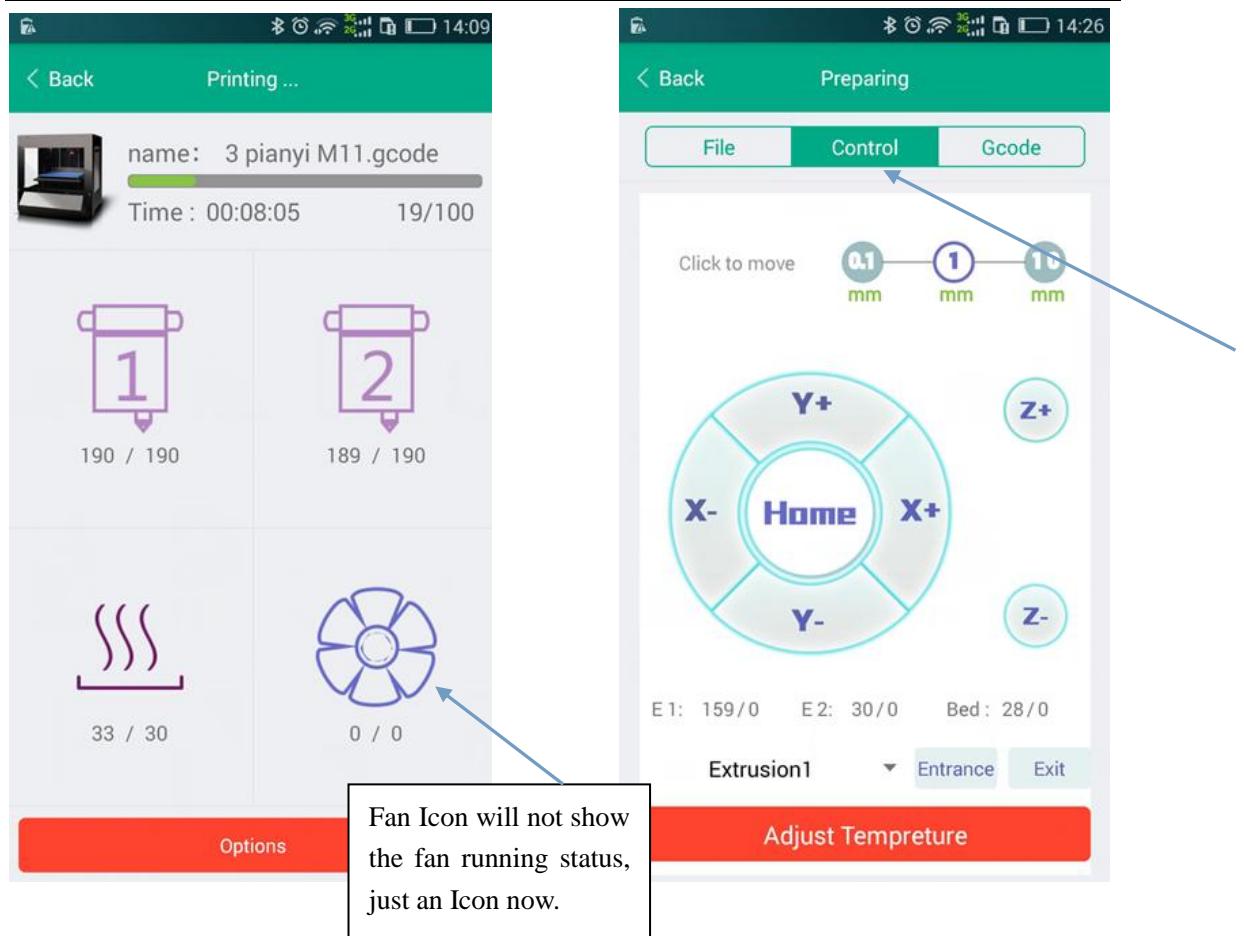
Fax:+86 575 88489635
E-Mail: bibo@ourbibo.com

Mobile:+86 133 25759139 55
Skype: [bradenking0202](#)



Click the printer name





12. Maintenance/Optimal Operational Tips

The BIBO printer is an easy product to maintain and clean. To keep enjoying your BIBO printer and ensure optimal print performance, regular maintenance is very important.

General tips for optimal operation

- Make sure the filament enters the hot-end is clean from dust. You can do this by letting it run through a piece of sponge before it enters the hot-end. You will be surprised how much dust will accumulate during a 5 hour print. That would have all entered the hot-end and formed a layer of contaminants, which would interfere with the heat going into the filament.
- Do not use too high temperatures. They will make the filament decompose and contaminate the inside of the nozzle.
- Do not leave the hot-end heated for hours. The filament might decompose after leaving it at high temperatures, leaving a layer of contaminants behind in the hot-end nozzle.
- Use good quality filament.

Cleaning

- To ensure good bed adhesion of prints we recommend cleaning the heating bed with spirit, alcohol, thinner or nail polish remover. We recommend doing

this before each print, especially for long prints.

- Be careful not to get any liquid onto the electric board, as it can damage the electronics permanently.
- Dust the printer off with a moist piece of cloth.

Regular maintenance

- Lubricate the vertical spindle and the linear railings with the sewing machine oil; we recommend doing this every 3 months.
- After some time the belts might lose a little tension. Tighten them until a healthy tension. The bolts and nuts on the printer will get loose too such as on the x axis carriage, please check it usually and tighten them .