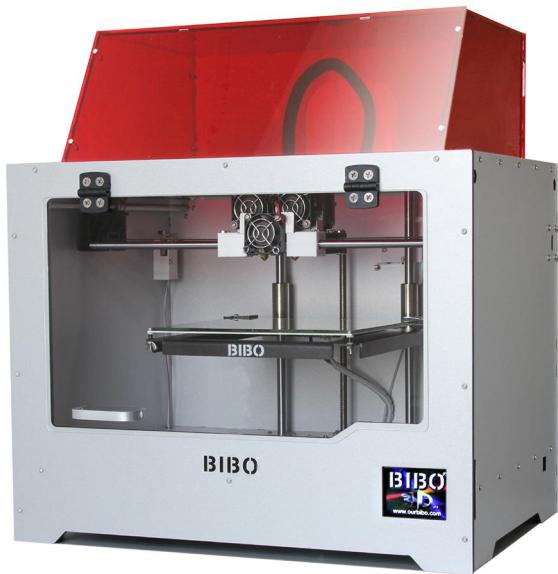


---

## **BIBO 2 TOUCH 3D PRINTER**

### **---Operation Manual**



**SHAOXING BIBO AUTOMATIC EQUIPMENT CO., LTD.**

V3.0a

Please operate the machine strictly according to this operation manual to have a promised warranty from BIBO! As the warranty is based on the 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.....	10
5. Some Notices for Printing.....	29
6. Software Installation.....	32
7. Slicing for A Gcode File.....	35
8. Acrylic Cover Installation.....	51
9. Function Introduction(including copy printing).....	53
10. USB Driver Installation (not necessary).....	57
11. WIFI (optional function).....	60
12. Maintenance/Optimal Operational Tips.....	83

## 1. Caution

Please be careful for human electrostatic especially in winter when touching the motherboard or extruders. The easiest way to solve this problem is that we can touch the metal part nearby by hand first before touching the motherboard or extruders. So the electrostatic usually will be gone before touching the motherboard or extruders.

SD card or USB drive for the printer should be smaller than 16GB. SD card should be class 4.

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

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 230-245 degree 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 or inserting SD card to the printer, the other operation on touch screen will be not responded** as the former action (heating or close heating or reading the sd card) 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.
4. **It is strongly not recommended to replace the parts**, as replacing the parts usually causes the damage or the new problem. Please kindly contact [bibo@ourbibo.com](mailto:bibo@ourbibo.com) first before replacing, as there may be much easier way to solve your problem without disassembling anything. For example, pushing down filament or the metal stick to the heated extruder is much easier then replacing the hot ends for the extruder clog problem.

**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.**



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 also 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 (may already fixed on the filament tube couplers support), 1 filament tube 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 may not run. When you want to print something, please copy the gcode files to SD card.**

**Red acrylic cover panels and clear door may have paper or film 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.

**The packing manual is here:**

<https://drive.google.com/file/d/1-CAvBgYopazYT4ImRduwQrP8KD-b0Q7L/view?usp=sharing>

### 3. Initial Hardware Installation

BIBO 3d printer's working principle is Fused Filament Fabrication(FFF). FFF is the most common method of 3D printing. It works by melting plastic material called filament onto a print surface in high temperature. The filament solidifies after it cools down, which happens instantaneously after it is extruded from the nozzle. 3D objects are formed with the filament laying down multiple layers. 3d printing is similar to building a house.

#### **3D printing involves three steps:**

1. Create or download a 3D model (usually in stl format).
2. Slice the 3D model in the slicing software (repetier host, cura or simplify 3d) and export the 3D file in gcode format file.
3. The printer recognizes the gcode format file and builds the 3D model.

We have the **BIBO parts assembly video and acrylic cover assembly video**, and you can copy it from the SD card which is sent with the printer to you or download from this link:

[https://drive.google.com/file/d/1e02U8qlDgyE1xym4yPbTf\\_fmFDw2x0-b/view?usp=sharing](https://drive.google.com/file/d/1e02U8qlDgyE1xym4yPbTf_fmFDw2x0-b/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.**

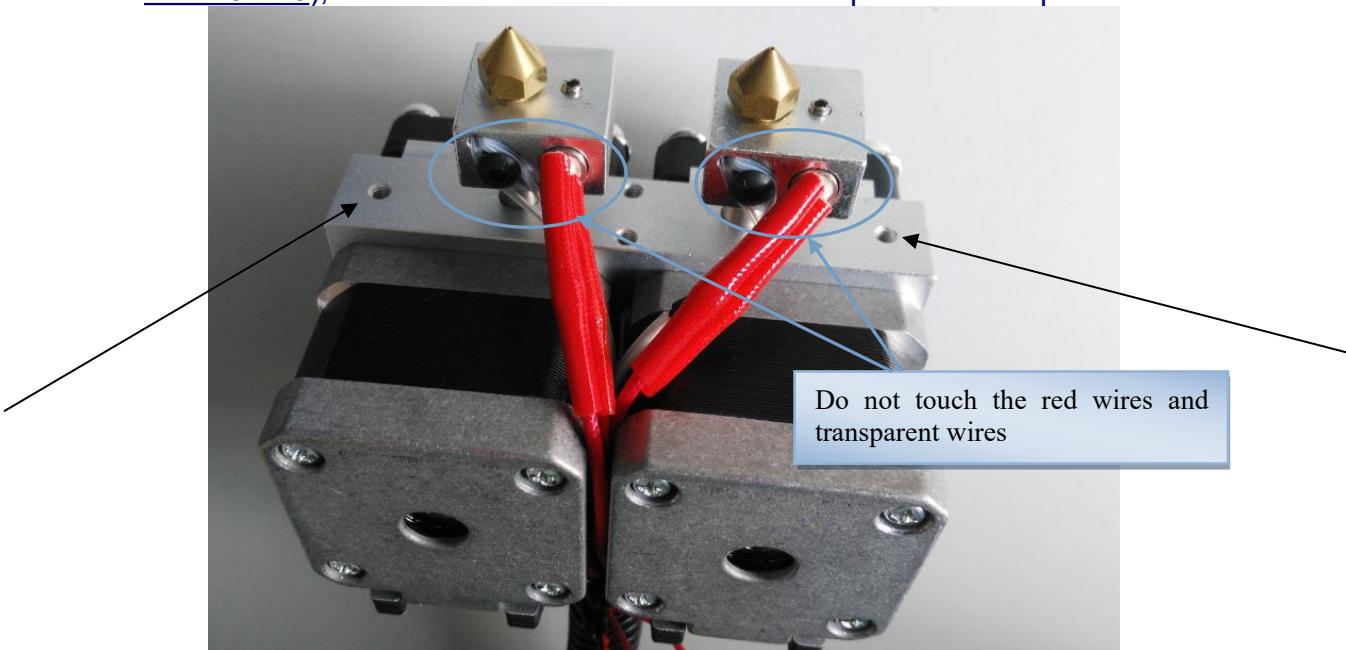
The printer is usually transported by courier. If the printer is all assembled, its volume will be much bigger, and the courier freight will be much more as freight depends on its volume, not actual weight. So we will let big volume part such as acrylic cover (fragile for shipping), filament run-out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

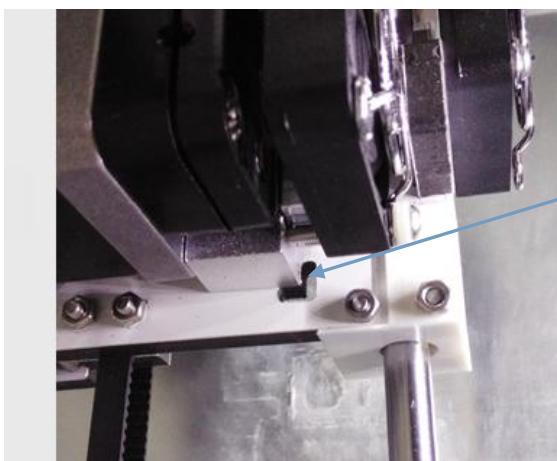
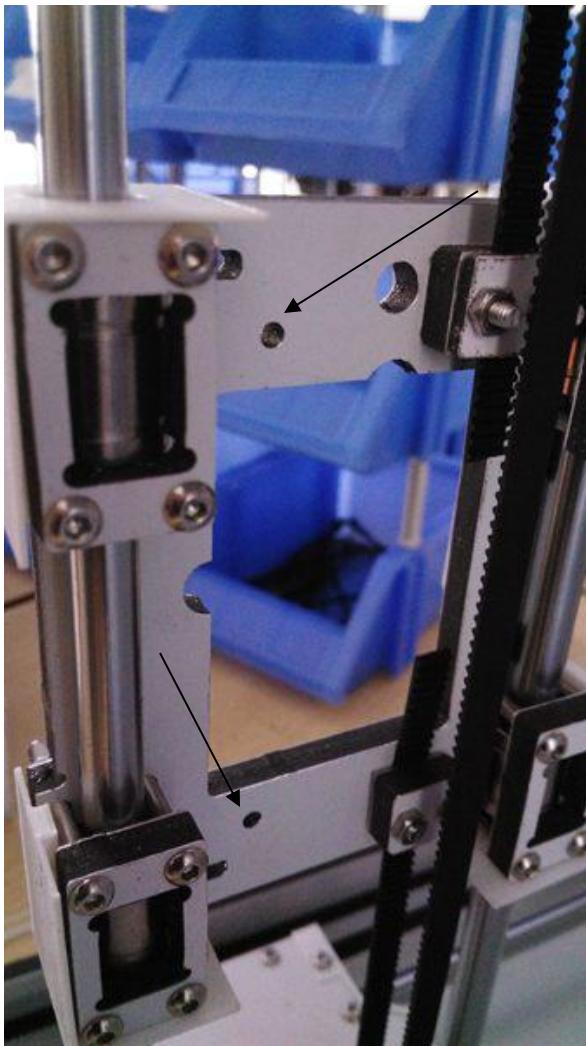
The first step is to take **the machine** from carton , also **taking the parts out from the small carton in the printer first** , then take the **small carton** out from the printer, and **remove all the protection bubble wrap and zip ties**.

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

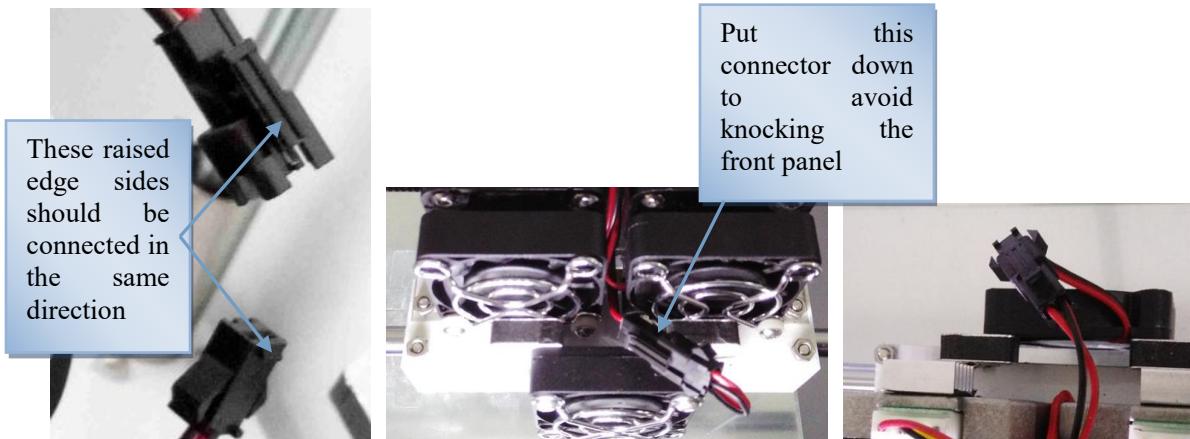
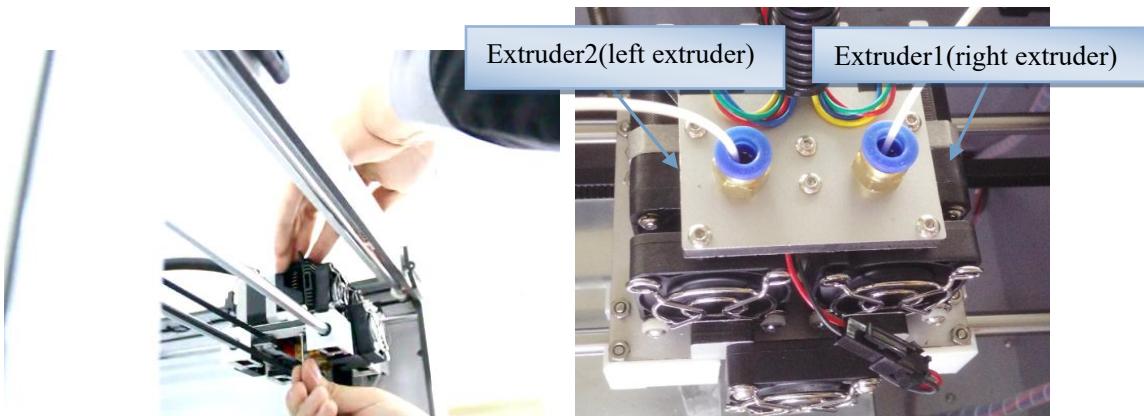


Then install the extruders with two bolts in 10mm or 8mm length (in a bag marked with 10 or 8), and connect two fans with black coupler. See the photo below:

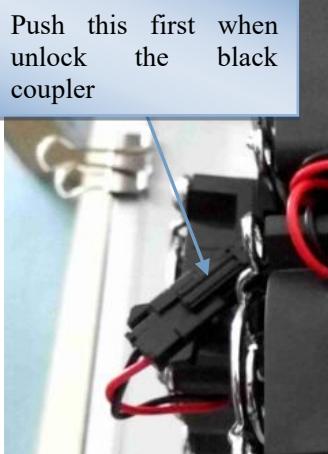




Let this aluminum bar be put in  
this position

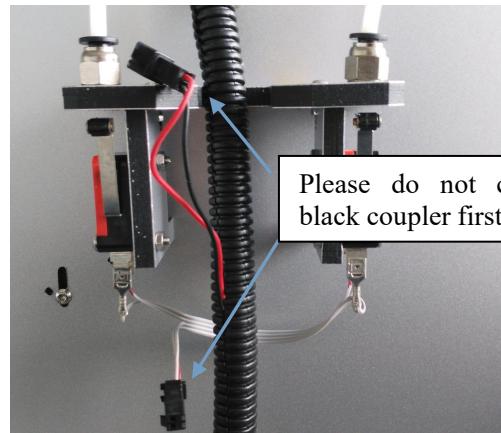
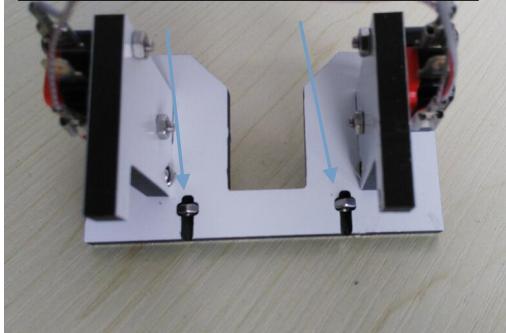


If you want to take the extruders down from the printer, just unlock two black coupler, no need to remove the cooling fan support, the same as you assemble the extruder. When you unlock two black coupler, please push the button first then unlock the black coupler as the photo below:



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), and **do not connect the black coupler first** as the photo below.

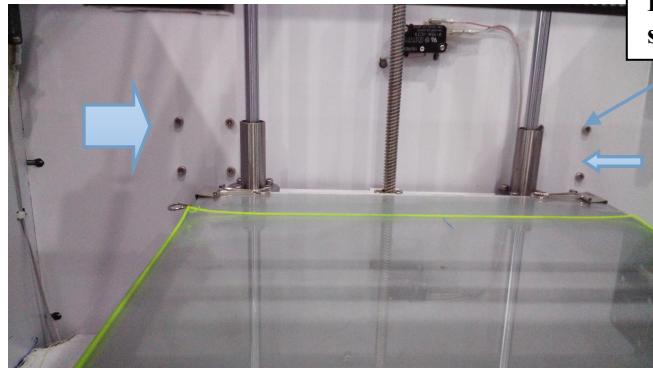
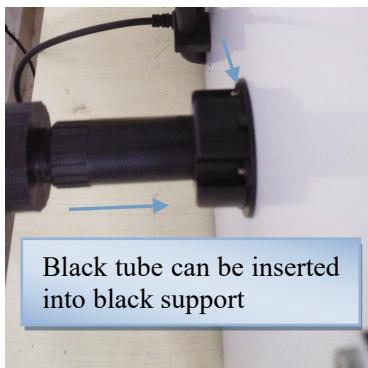
Please press two nuts by two fingers during assembly to avoid nuts falling down.



The third step is to install filament holders on the back of printer with two big white bolts.

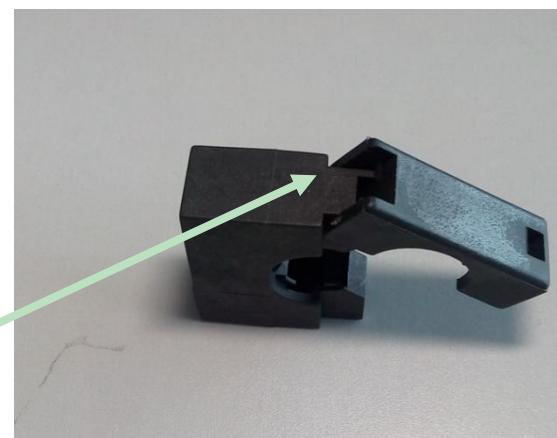
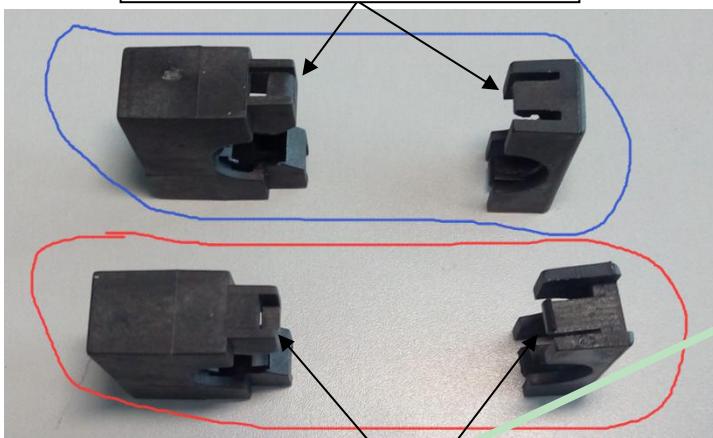


Some printers' filament holders are fixed in a different way, with 12mm length bolts (in a bag marked with 12). Please kindly check the photo as below:

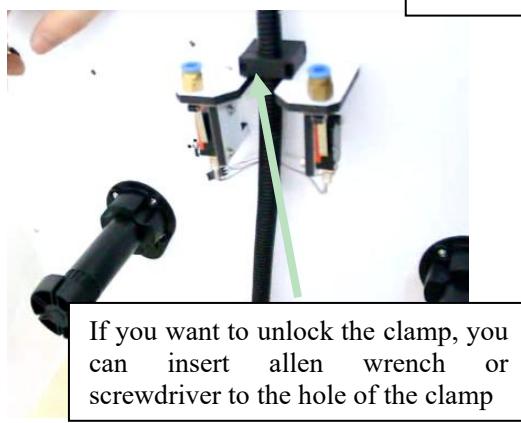


The fourth step is to assemble the extruder wire clamp with 16mm or 12mm length bolt. We have sent you two clamps. One is to be assembled on the top side of the printer's back panel, and **the other one is for spare**. Please kindly check the photoes as below. **Let the black wire cable be straight (just only a little tight)** . If too loose, the black wire may be not long enough for extruders to travel inside the printer.

This two side should be locked later



This two sides should be locked first



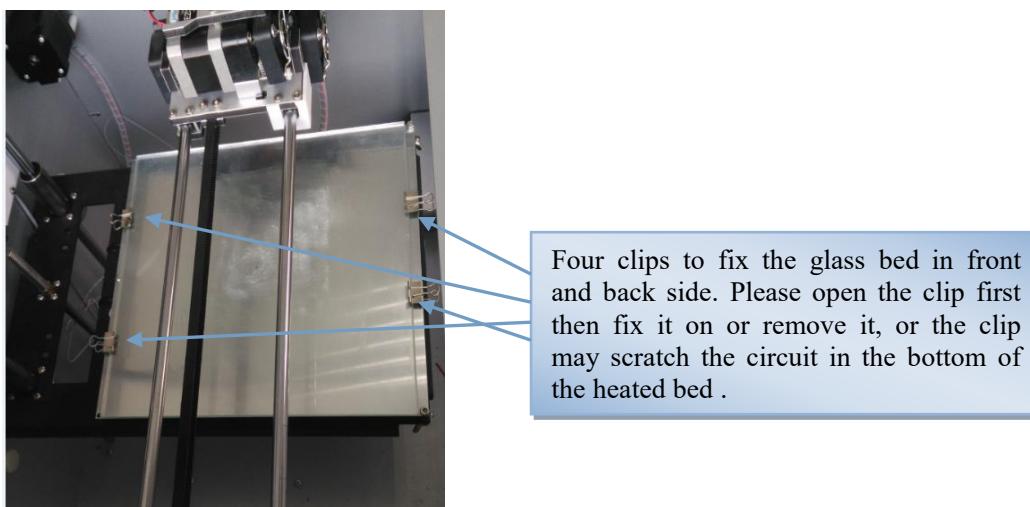
Congratulations! You have completed the initial hardware installation! For acrylic cover installation, it is in chapter 8. You can install it after first printing test.

#### 4. First Print

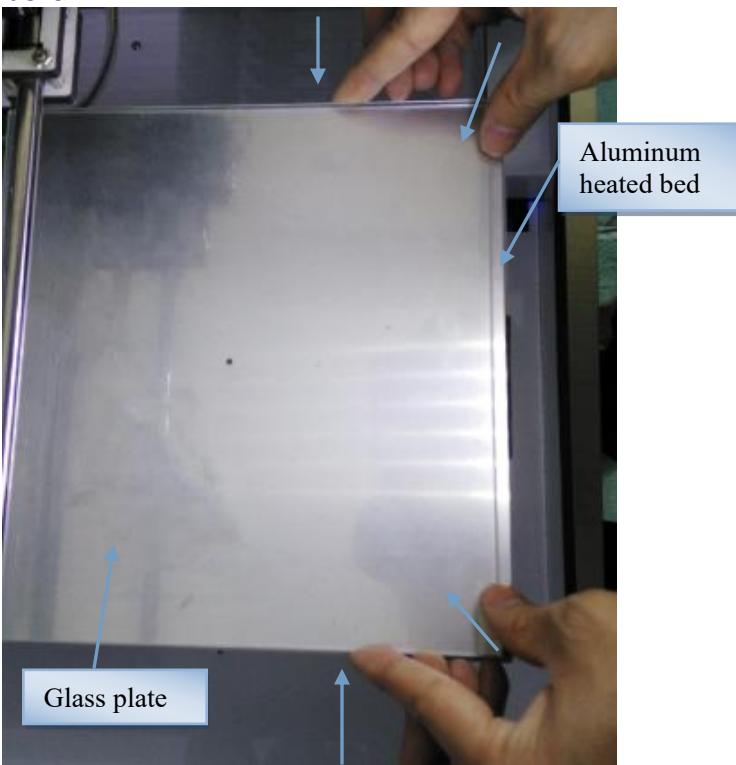
The first step is Leveling the heated bed(not necessary).

After transportation, **only a few printers's beds** have to be leveled to ensure your 3D prints stick well to the heated bed. **How to check whether you have to level your heated bed or not, please follow steps below.**

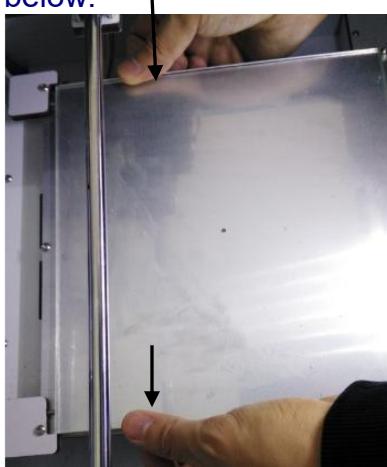
- 1.Put the glass plate on the aluminum heated bed (two ways).



Some printers don't have four clips sent with the printer, but with two clamps fixed on the backside of the heated bed. If so, the fixed way is below:



Push the glass plate to the bolts on the two rear clamps 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 also check the glass bed whether can be easily moved in horizontal direction.

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

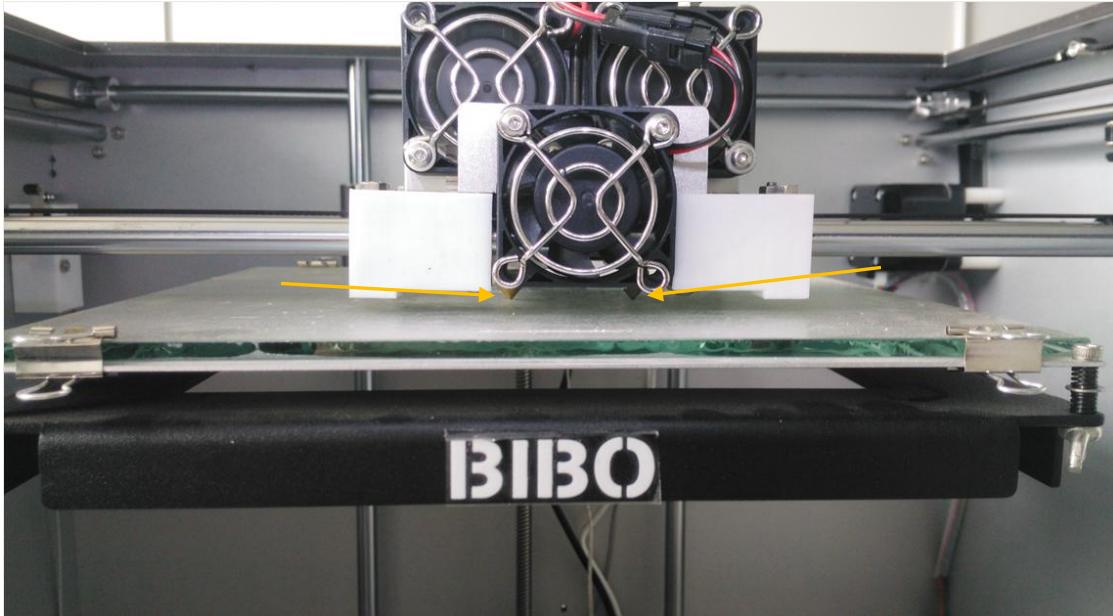


3. Now **move the extruders to the right side first**, and use your fingernail to touch the “Z” button to run the Z axis homing. If touch “all”, the extruders will be locked and you can't move it, and the following leveling step can't be done. You can restart the printer and touch “Z” again to home Z axis.



**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 second step to load the filament first.

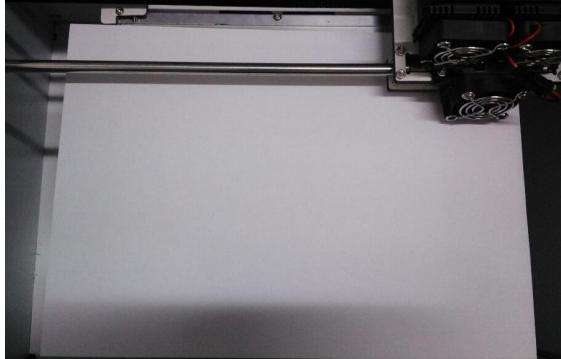


If nozzles can't be moved on heated bed as they are below heated bed glass a lot, the video instruction is here:

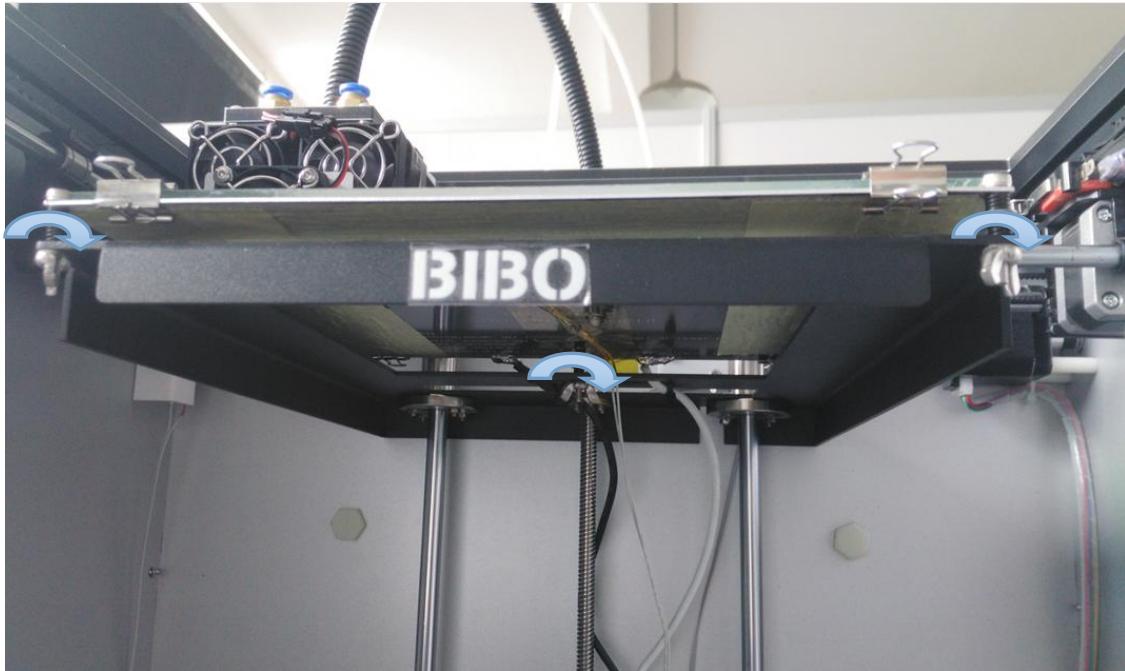
[https://drive.google.com/file/d/12BrsKYa6\\_Yj4E5fzE3rCPnxrbGgElkqr/view?usp=sharing](https://drive.google.com/file/d/12BrsKYa6_Yj4E5fzE3rCPnxrbGgElkqr/view?usp=sharing)

You should do the following steps to level the bed as below:

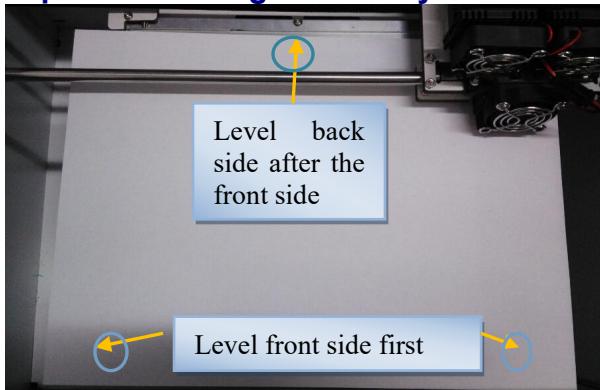
Put a paper such as a A4 paper on the glass plate,



Rotate three wing nuts Clock-wise to let the glass bed down and nozzles can move on the paper. If the nozzles still can't move above the glass bed, please check the file named "leveling problem Bend and move the Z axis limit switch" in the troubleshooting file in the SD card sent with the printer.



Now move the extruders between the printer nozzles and heated bed to let a A4 paper be a little 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 screws and fix in place, then level the back side with one screw. Just level in this order to level the bed again and again. Please note that the leveling can't be perfect, as the bed can't be perfectly flat. 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. After leveling, please make sure three springs under the bed should be tight, or the leveling calibration can't be kept well for long time and you should level it again.



You can also use leveling menu on the touch screen (not recommended), just click one by one with fingernail or a pen, checking the paper movement and adjust the wing nuts. **Without leveling the center point, the leveling can be easier. This leveling 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.**

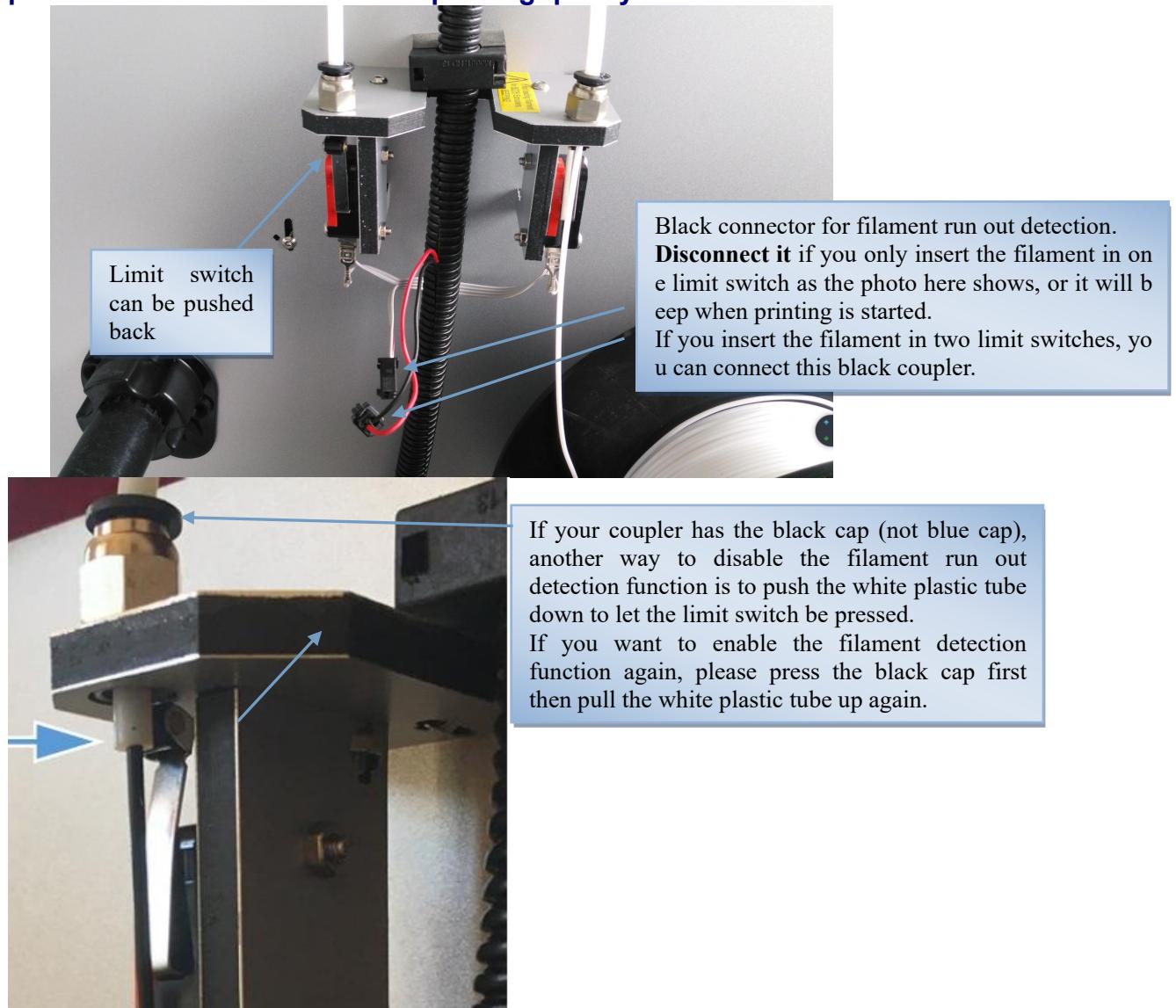
**The Second step is loading the filament for two extruders**

BIBO touch 3D printer has the function of filament run-out detection.

**Beeping problem:**

We should load filament for both extruders. Or we can't print or engrave and the printer will beep when you start the printing because of filament run out detection. We send one spool of PLA filament, and you can cut some from that spool to have a roll of filament. If the printer has the latest touch screen firmware, the touch screen will show "filament detection switch not pressed" when two filament detection switches are not both pressed because of filament run out detection.

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 the printer and you will see the main menu:



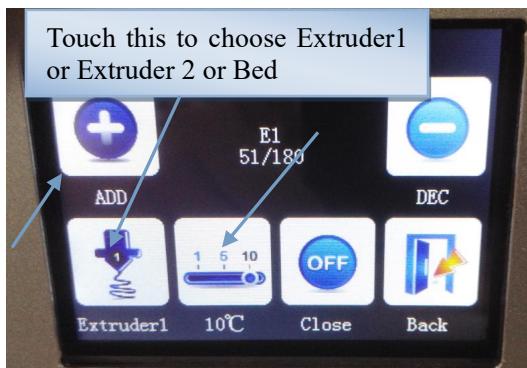
If your touch screen's main menu's **background is not in black** as the photo above,

You can update the latest firmware according to this link:

<https://docs.google.com/document/d/1dpXsl0ZKrTG7c7OuIY9BxZCUDk8tSWcX/edit?usp=sharing&ouid=103768291371300327502&rtpof=true&sd=true>

- 2) Now use your fingernail to touch the “Preheat” 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/file/d/0B76TjeOkslHINI9fZUI3U0xoMTg/view?usp=sharing&resourcekey=0-TuyWre7lijIVJTd3nALrSQ>



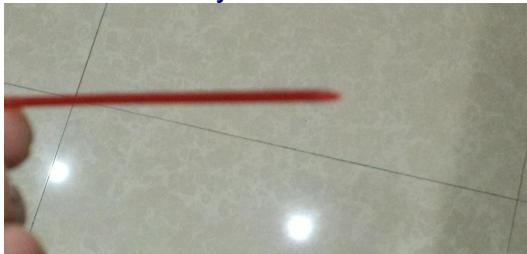
- 3) You can choose nozzle Extruder 1(right extruder) and nozzle2 Extruder 2(left extruder), then set the temperature at 220°C by “add” button as usual (this temperature is according to your material type and manufacture's instruction. For PLA usually 200°C, and for ABS usually 220-240°C. It also based on the former existing filament in the nozzle). Now you will see two extruder's temperature is rising.

- 4) Now we will do the **loading filament step for two extruders**. 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.



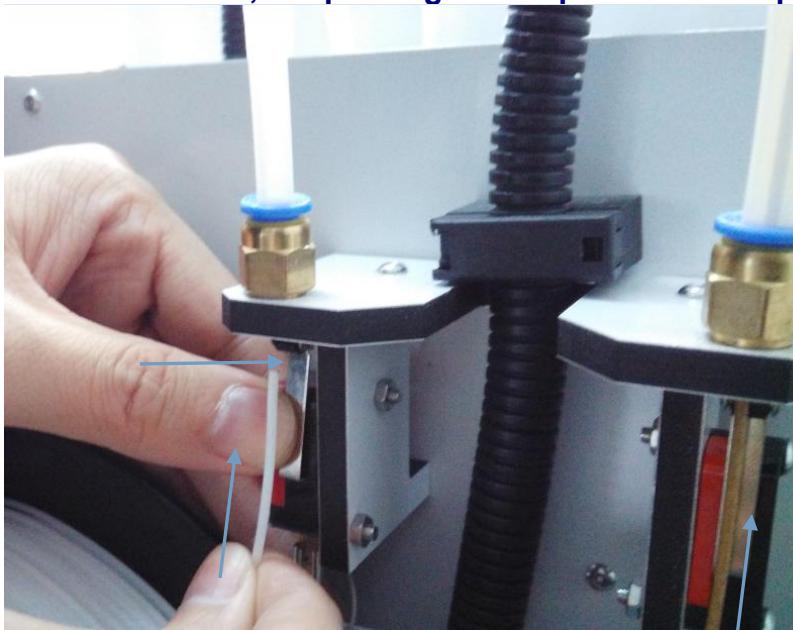
**Sharp the filament head** with scissor and **let the filament be as straight as you can** to ensure it come to the hole of extruder smoothly. If it is difficult to put the filament to the hole of nozzle on the way, please rotate the filament in different direction and try more times.



The loading filament video is here:

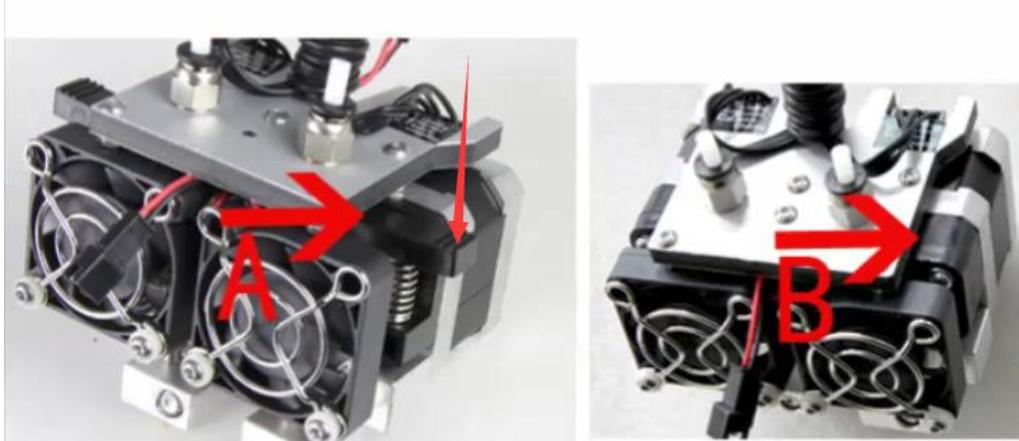
[https://drive.google.com/file/d/18hwVbFtGzk53abkJP-N7KkdXEnYZVg6\\_/view?usp=sharing](https://drive.google.com/file/d/18hwVbFtGzk53abkJP-N7KkdXEnYZVg6_/view?usp=sharing)

Load the filament to the filament guide tube for two extruders (**if only loading for one extruder, the printing will be paused or beeping**) as the photo below:



When the extruders reach 220°C, please push the filament down to the extruder about 90mm (3.54 inches ) deep by hand. **If the extruder has the lever, please**

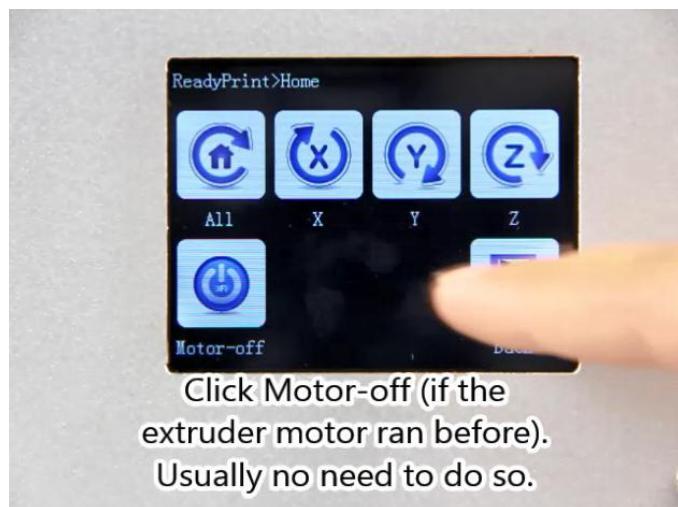
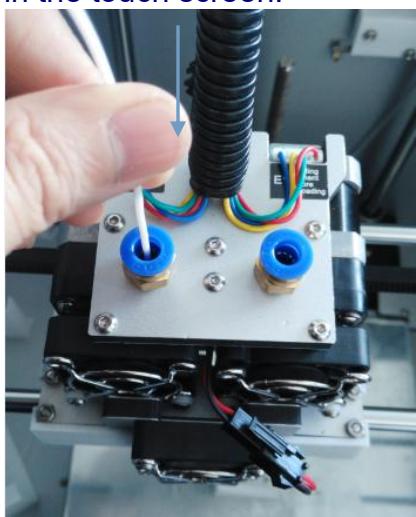
press the lever down a little before pushing the filament into the extruder.



**A. If the extruder has the lever, press the lever down a little first.**

**B. If the extruder has no lever, no pressing needed.**

If it is hard for you to push the filament down, please check the filament whether is **straight enough in 90 mm (3.54 inches)**, and the filament tip is pointed in "V" shape. You can also rotate the filament inside the extruder and try more times. Usually it will work. **Another cause is that the extruder motor is locked, you can restart the printer or click "Motor-off" button on the Home menu in the touch screen.**



This is for E2 (left extruder) in the photo above  
The filament will come out from the extruder's brass nozzle. Go back to the main menu to touch "Preheat" to go to the "preheat" interface again

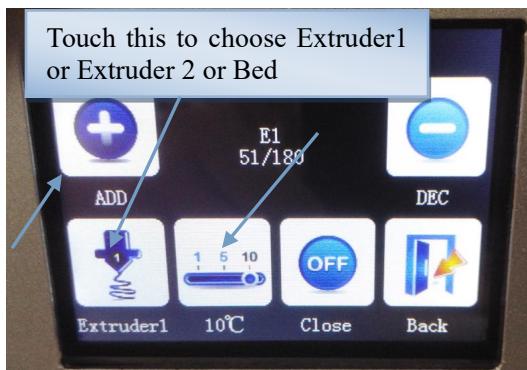


Touch "close" to let the extruder's temperature come down (if the filament stays in the hot nozzle for long time, the extruder will be clogged. We should cool down the extruder after loading filament).

Now loading filament for one extruder is finished.

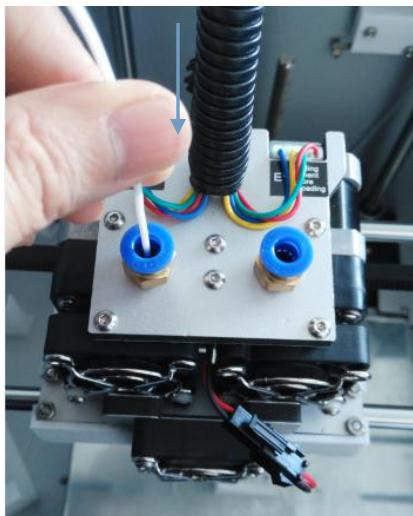
### **Another way of loading filament is below:**

Touch the "Preheat" button, and you will see



You can choose nozzle Extruder 1(right extruder) and nozzle2 Extruder 2(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 former existing filament in the nozzle). Now you will see two extruder's temperature is rising.

When the extruders reach 220°C, please push the filament down to the extruder about 40mm (1.57 inches ) deep by hand. If it is hard for you to push the filament down, please check the filament whether is **straight enough in 40 mm (1.57 inches)**, and the filament tip is pointed in "V" shape. You can also rotate the filament inside the extruder and try more times. Usually it will work.



This is for E2 (left extruder) in the photo above

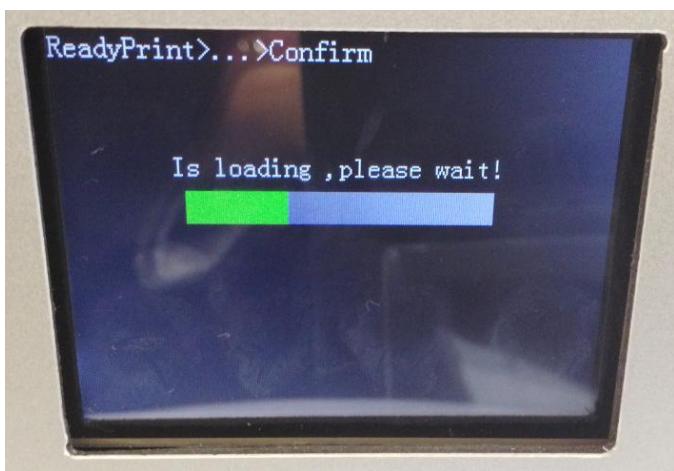
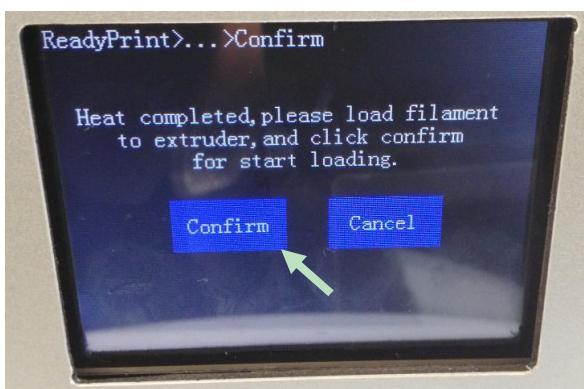
please touch "back" on touch screen to go to main menu, touch "Settings"

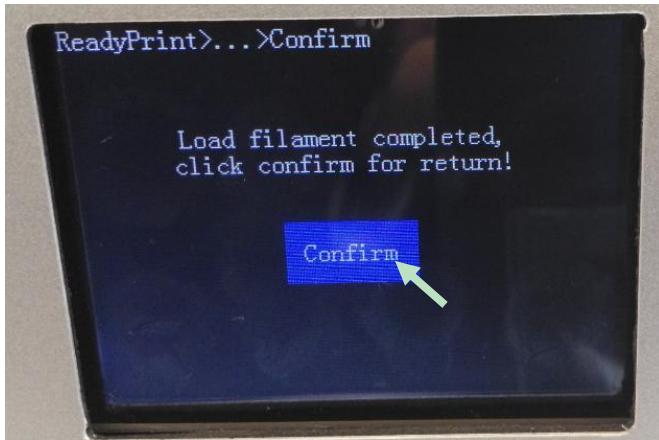


Touch "Filament" ( change)

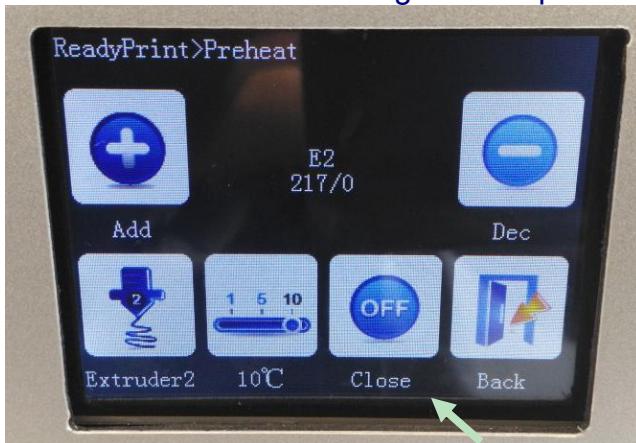


Touch "Load"





The filament will come out from the extruder's brass nozzle. Go back to the main menu to 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 loading filament.**)

If your printer has the latest touch screen firmware, the extruder will be cooled down automatically after loading or unloading, so you don't have to do the above step to close the heating.

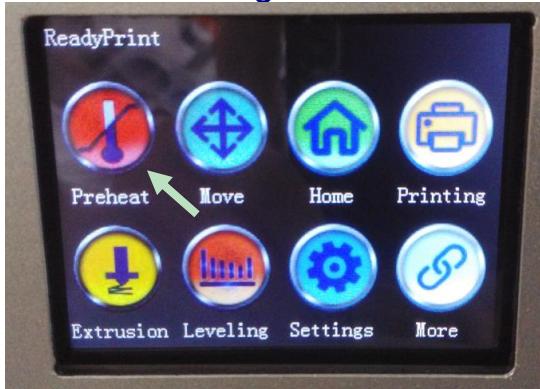
Now loading the filament to **the other extruder** in the same way.

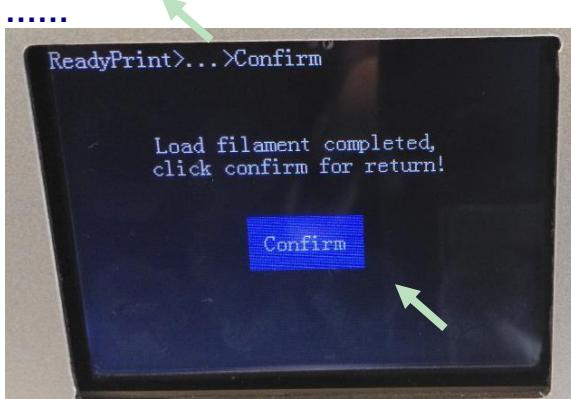
#### **Unloading filament:**

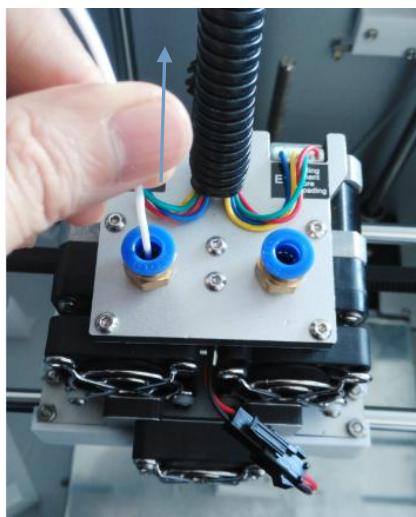
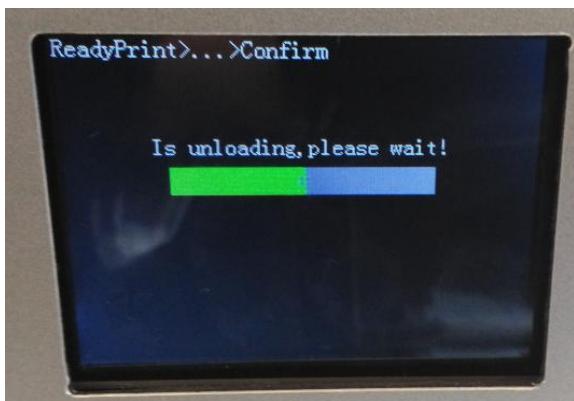
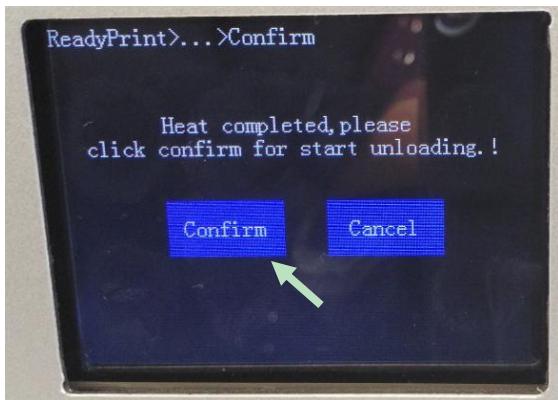
If you want to **unload the filament**, the operation is the same as loading filament, heating up the extruder to filament melting temperature such as 220°C in preheat menu on the touch screen. The difference is to **load the filament first (Load), then IMMEDIATELY unload the filament (Unload) . This will reduce the extruder clog risk. The unloading filament video is here:[https://drive.google.com/file/d/1vd5nefRwwZJMfgsBG\\_WOjXoQWKL-SlsQ/view?usp=sharing](https://drive.google.com/file/d/1vd5nefRwwZJMfgsBG_WOjXoQWKL-SlsQ/view?usp=sharing)**

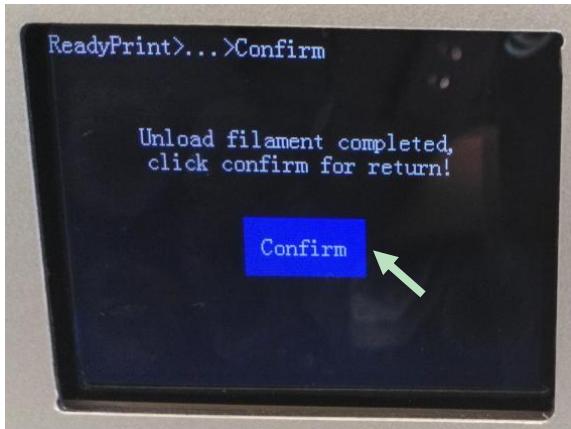
If your printer has the latest touch screen firmware, when you click unload button under settings---filament menu, it will load the filament first automatically then unload at once to reduce the extruder clog risk. so you don't have to click load button first then click unload button at once.

During unloading the filament, please hold the filament above the extruder by hand. Finally pull out the filament by hand and close the extruder heating on touch screen preheat menu.

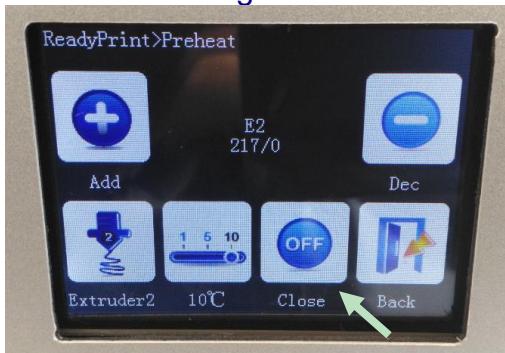








Close the heating.



If your printer has the latest touch screen firmware, the extruder will be cooled down automatically after loading or unloading, so you don't have to do the above step to close the heating.

**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:**

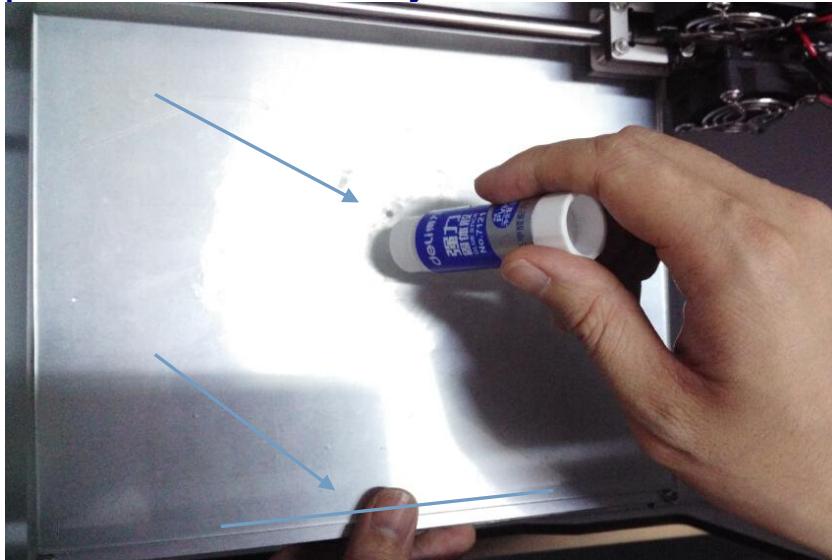


**We have calibrated the tension for extruding system. If you disassemble them, the calibration will be broken.**

**The 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. Clean your glass

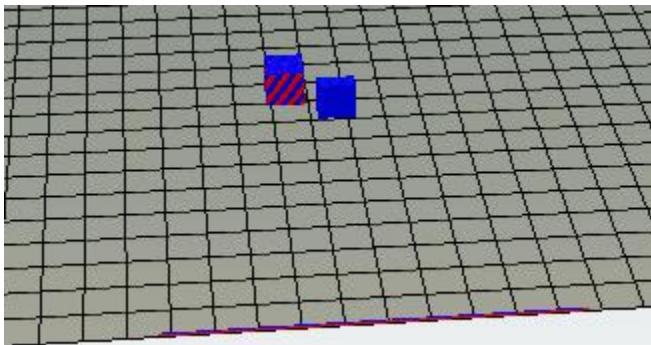
plate and then spread the glue onto it (for PLA, about 3 layers. For ABS, about 5-6 layers). With a damp piece of paper or cloth spread the glue out over the bed and **let the glue dry** (starting a print that uses the heated bed will make it dry quickly). After the water evaporates you will be left with thin layers 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 heat the bed to 60 degree and then the print can be removed easily.**



We will give you some test gcode files in SD card's test gcode file sent with the machine, two files are for testing the single extruder printing, and another one 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 printer to choose them to test the machine with our filament sent to you.

HOUSE left extruder PLA 180 degree 80 minute...	2017/11/14 13:46	GCODE 文件	5,807 KB
y186 dual extruder cuboid PLA 180 degree 15 ...	2017/11/14 13:47	GCODE 文件	311 KB
y186 right extruder cuboid PLA 180 degree 13 ...	2017/11/14 13:49	GCODE 文件	104 KB

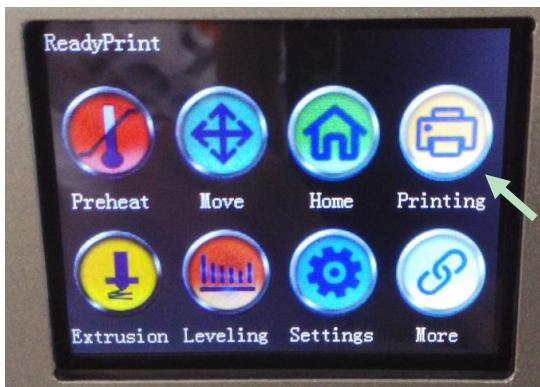
PS. For dual extruder printing test, the blue one in the photo below is the only object you want to print, the other one in red and blue is wipe & prime tower for better dual printing, so its printing quality is not concerned.



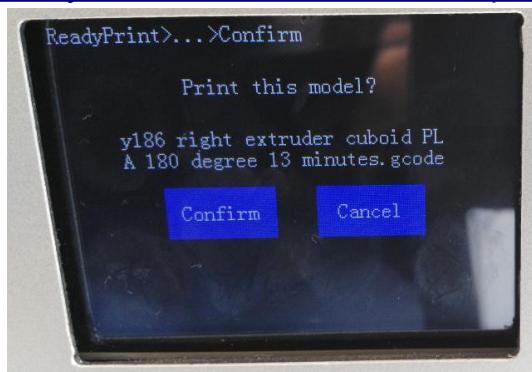
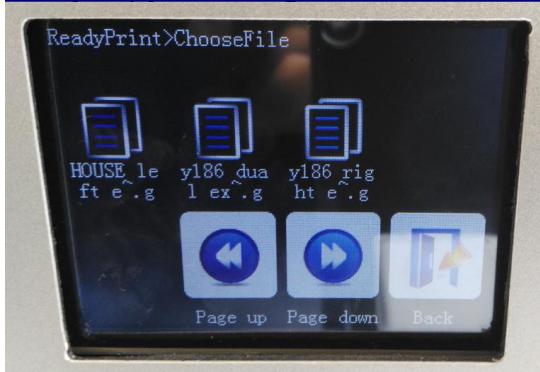
Before the first print, there are something **IMPORTANT** to read first:

1. 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) for long time. 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

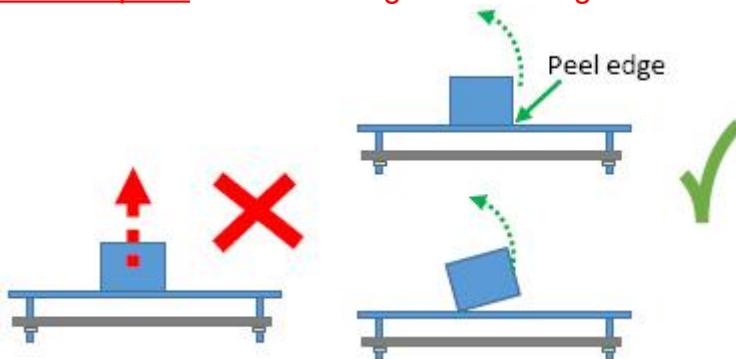
1. Some filament were not good winded on rolls in order by filament manufacturer or the user, so it will be tied a knot. See the photo as below:

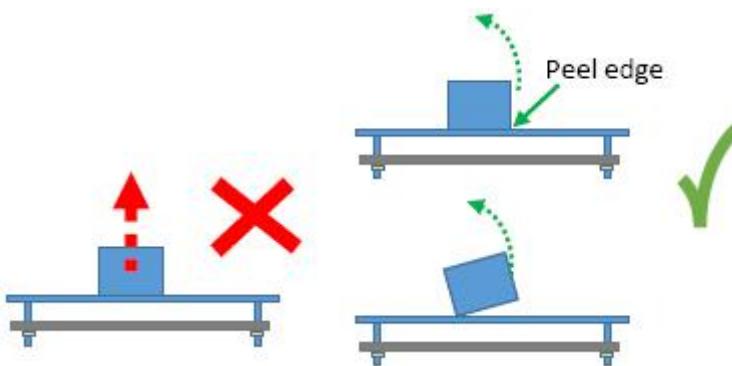


So please keep an eye on the filament roll and loose the filament if there is a knot. If not, you will see the base of printed object is ok, but the top part is bad in printing.

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 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 60 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 may 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 for both printing speed and retraction speed to be 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. Please also don't let the printer be connected with other devices by usb cable, or the touch screen may be not responsive.

- 
11. 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.
  12. 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.
  13. When you pause the printing and want to change the filament for both extruders, please change filament for the other extruder first, then change filament for the former printing extruder when paused, or the printing will switch both extruders as changing (extruding) filament for the other extruder before resuming the printing will activate this extruder and let this extruder print when you resume the printing.
  14. Please note that our heated bed Max. temperature is 100 degree Celsius when the printer is in the enclosed room (room temperature 30 degree celsius), and the printer should be enclosed too. If you set the bed temperature above 100 degrees, the printing will be failed.

## 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.

**Simplify 3D settings** and instruction are also in the folder named "BIBO simplify 3d setting" in the SD card sent with the printer.

**Please kindly use the newest slicing software settings here (have solved some bugs, better than the settings in the sd card):**

<https://drive.google.com/file/d/0B76TjeOksIHIZ2VBLXBSdzJYdms/view?usp=sharing&resourcekey=0-boJmBv7vQyKVocbp8qtJRw>

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:

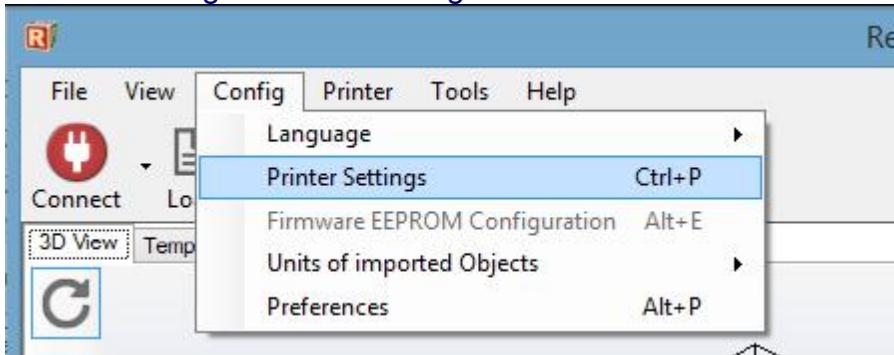
<http://www.repetier.com/documentation/repetier-host/>

For Mac users, please kindly check the link below:

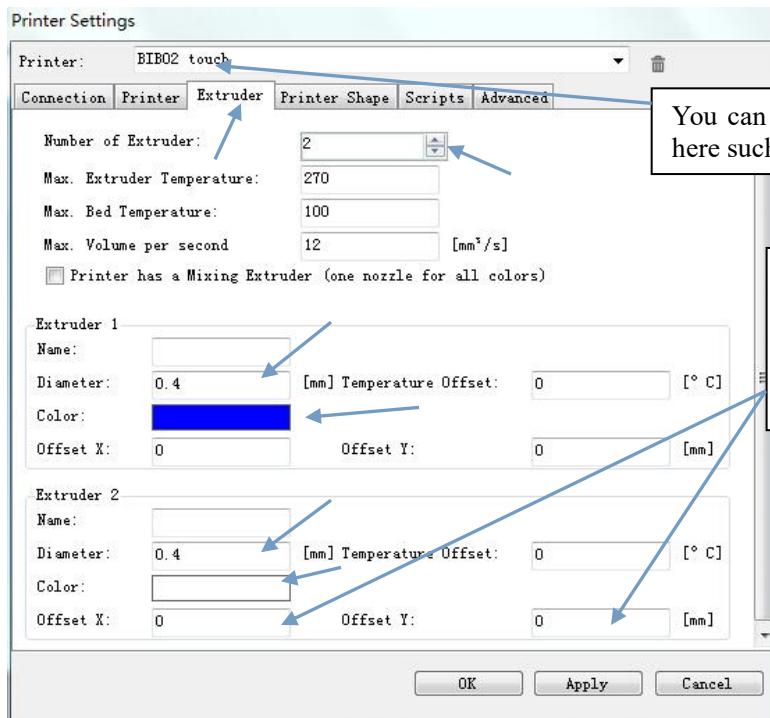
<http://www.repetier.com/documentation/repetier-host-mac/>

3. Start Repetier-Host.

4. Go to Config -> Printer settings.



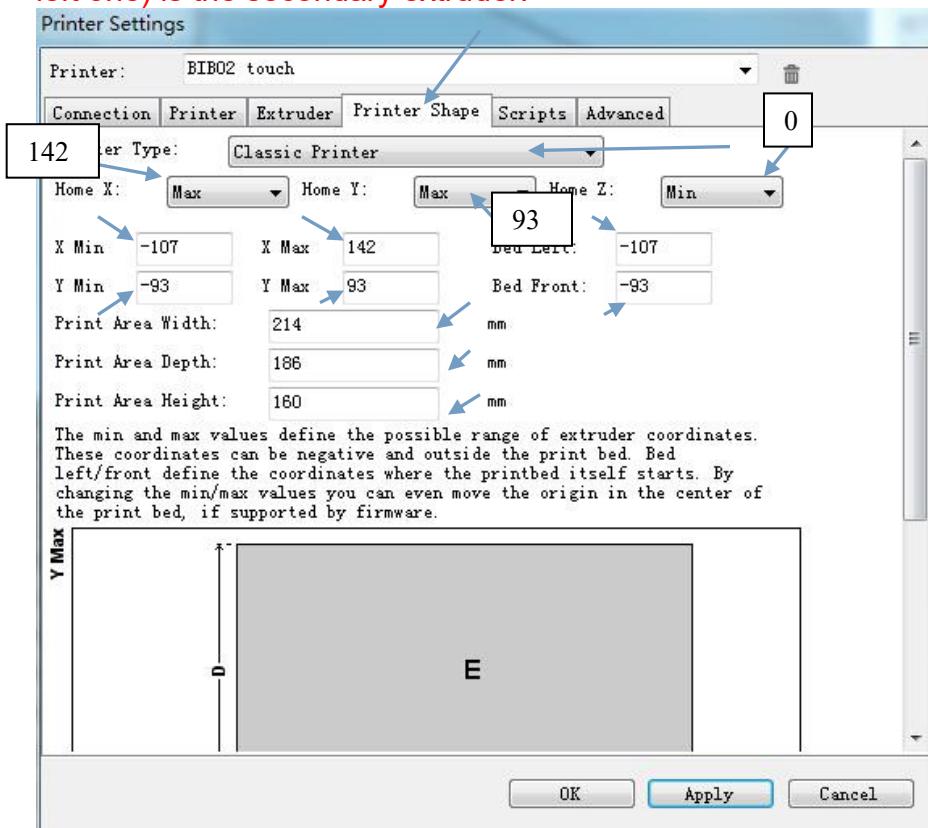
5. Once you see the printer settings window, please fill out the parameter as photo below:



You can also edit your printer name here such as BIBO2 touch

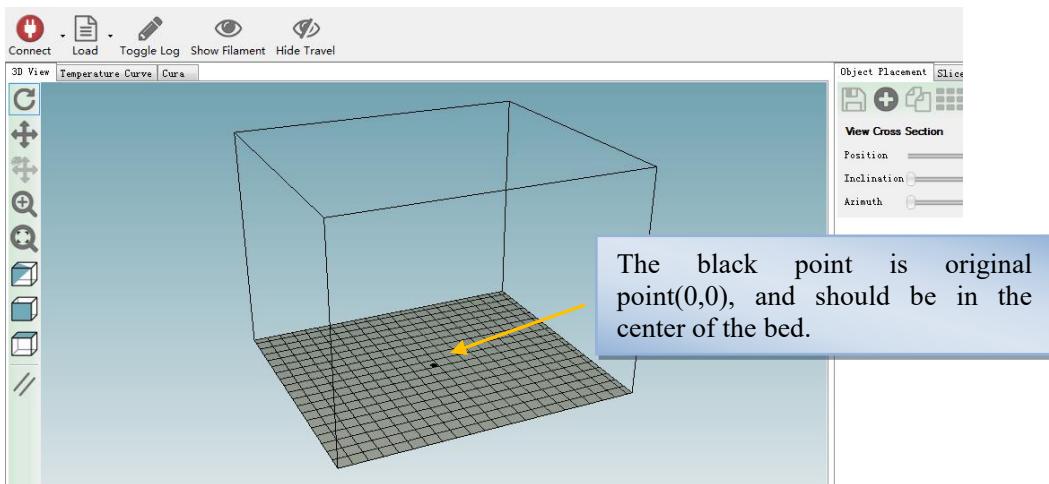
These two value of extruder2 will be showed in white name plate on the backside or two sides of the printer. Please fill out these two matching value.

PS. Extruder 1(the right one near x axis motor) is the first extruder. Extruder 2(the left one) is the secondary extruder.



**Please fill the value as the photo above, or your printing will be not in the bed center.**

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.

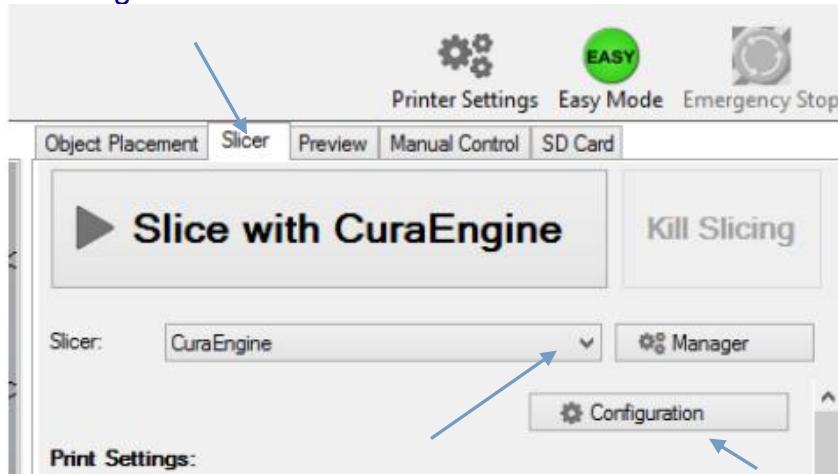


## 7. Slicing for A Gcode File

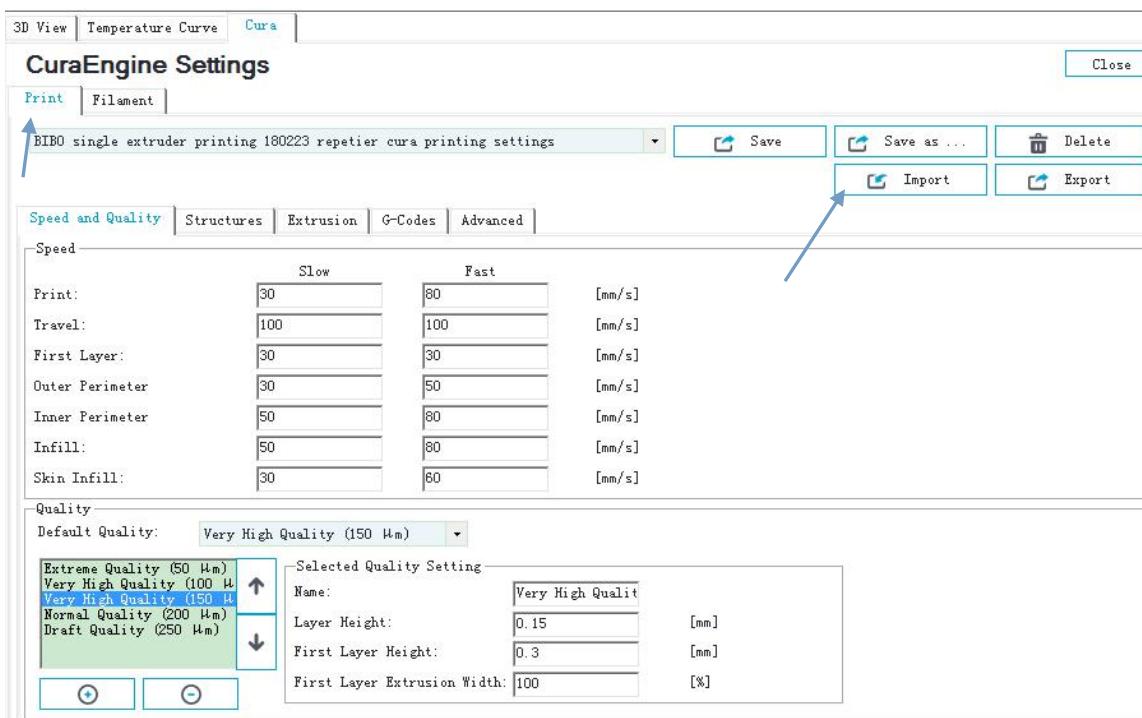
Repetier-host is an open-source software and is supplied software with our printers. We will give you repetier-cura settings, and you can import these settings to repetier-host to slice the printable object.

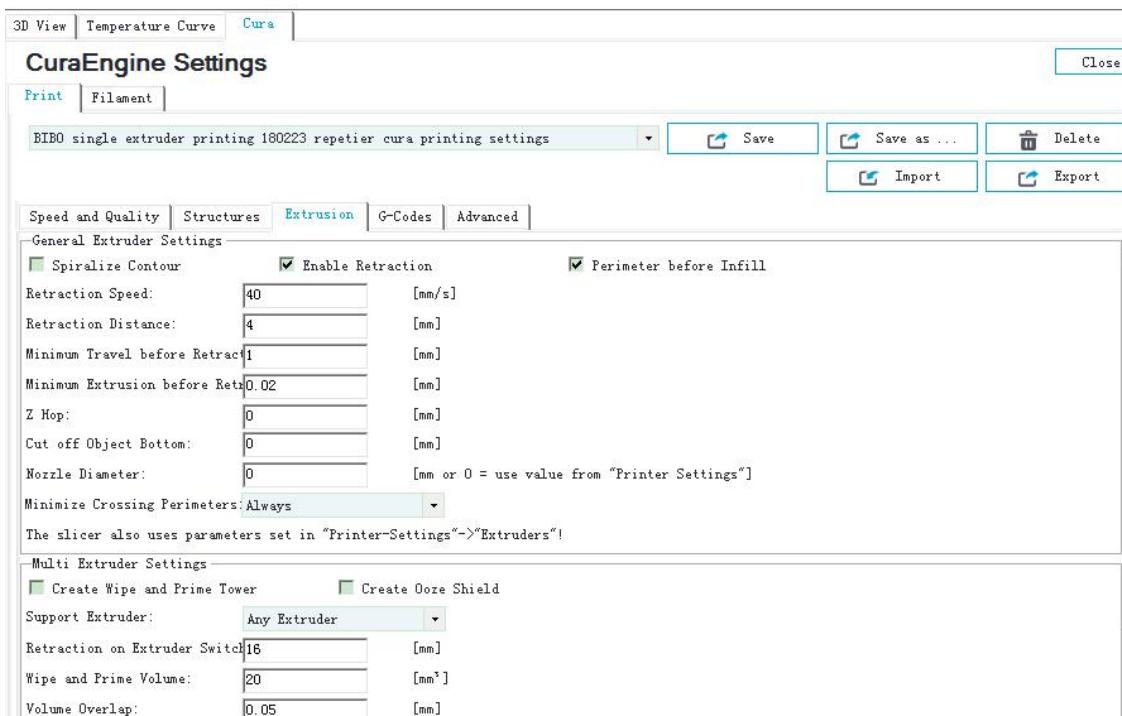
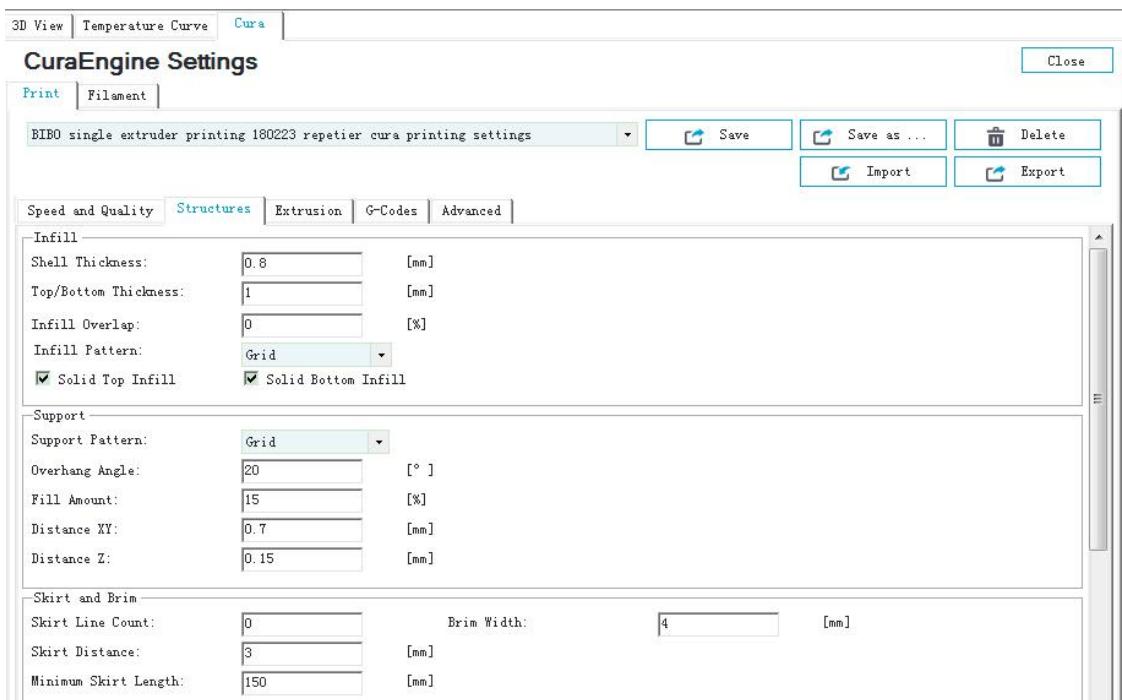
### Single extruder printing:

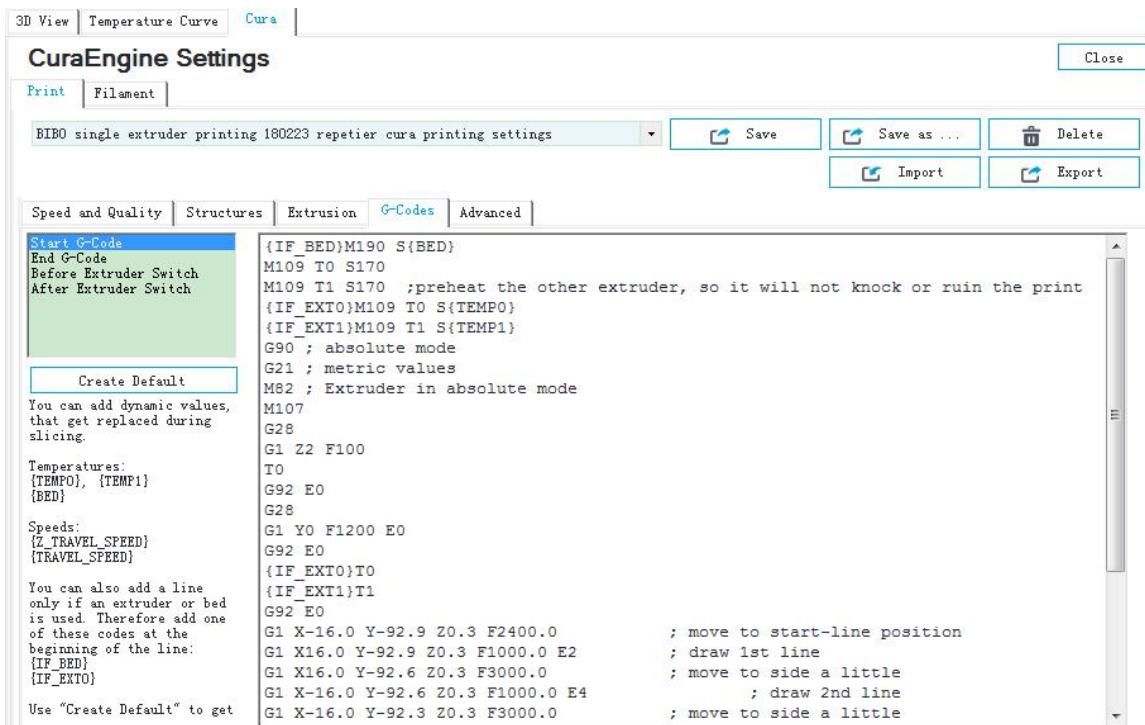
Now start the Repetier-host, click the Slicer tab, and choose the slicer to be CuraEngine.



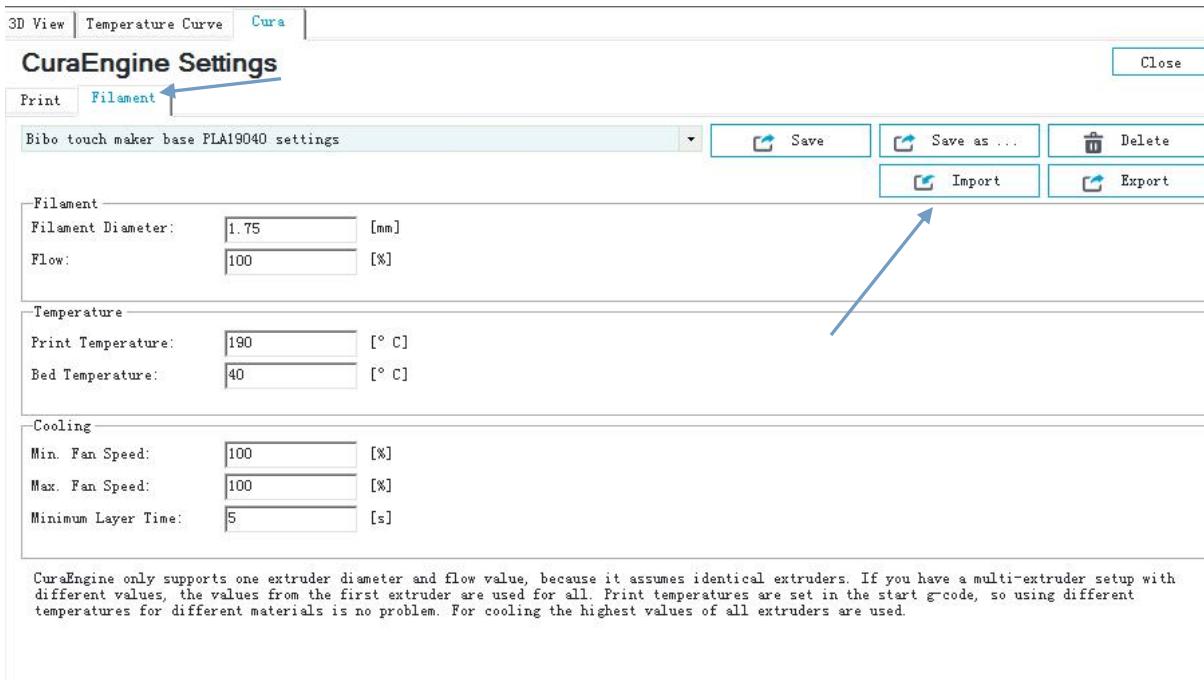
Then click Configuration, choose the Print tab. Click Import button, then import "BIBO single extruder printing 180223 repetier cura printing settings.rcp" from SD card's "1 BIBO touch maker repetier host settings" folder. You will see our printing settings as the photo below:

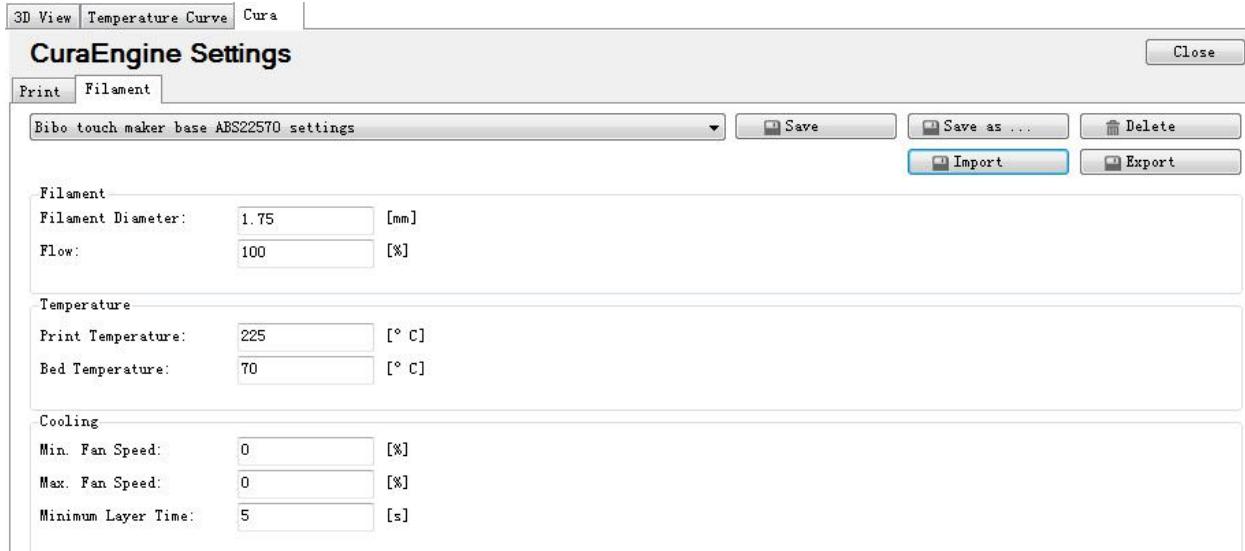






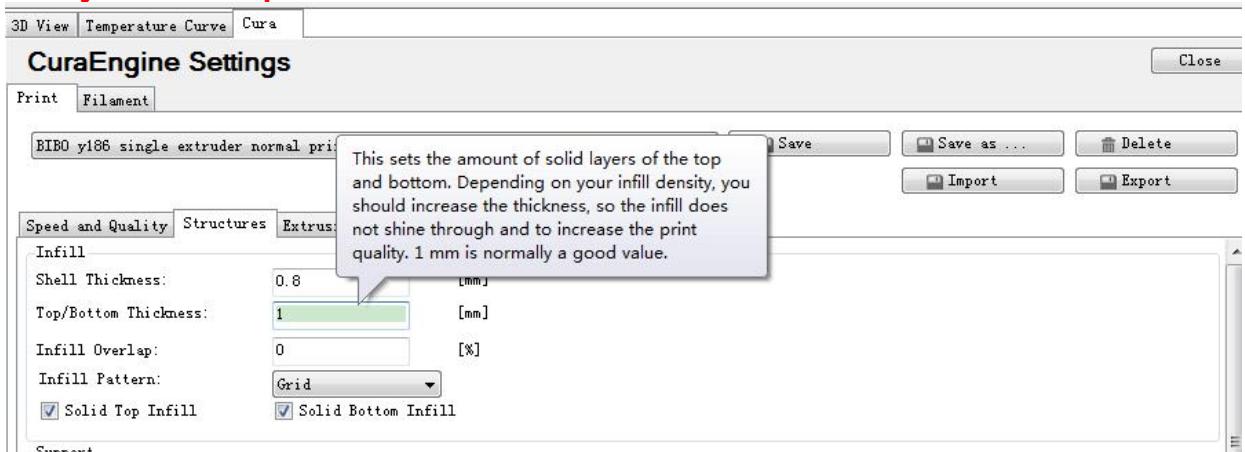
Meanwhile, choose the Filament tab. Click Import button, then import the “Bibo touch maker base PLA19040 (maybe 20040, depends on filament) settings.rcf”. from SD card’s “1 BIBO touch maker repetier host settings” folder. You will see our printing settings as the photo below:





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 host 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 degrees 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 degrees, 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.

But please don't print in too low speed for long time. Because of too low printing speed, the stepper motor works in a sympathetic vibration mode and makes the noise. This is not good for motor and its driver. Long time printing in very low speed (below 20mm/s) will cause the printing randomly stopped .

Our printer has two extruders, which may cause the other nozzle knock the print during printing especially for big or tall object. You can preheat the other extruder which is not for printing in about 170 degrees, so the other nozzle will not ruin your print.

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:

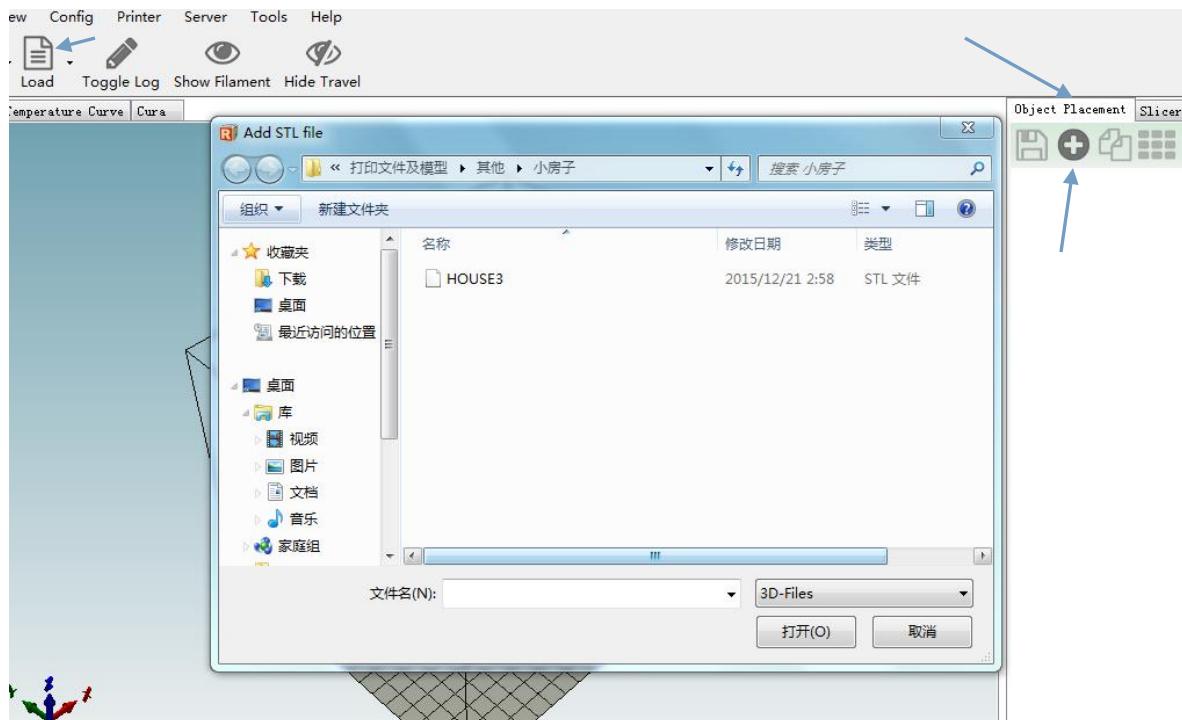
<http://support.3dverkstan.se/article/23-a-visual-ultimaker-troubleshooting-guide>  
<http://support.3dverkstan.se/article/30-getting-better-prints>

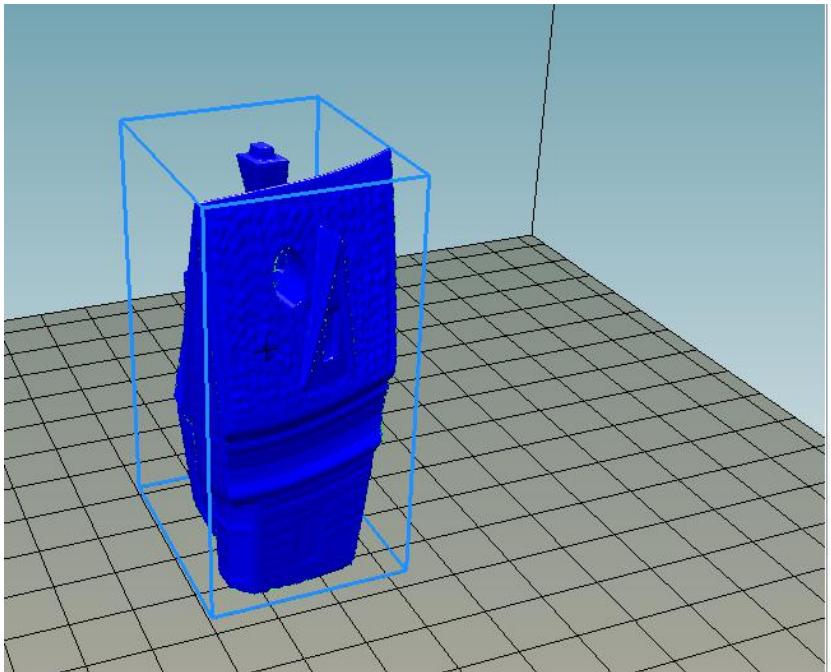
**If the layer is thinner, the printing temperature should be lower;**  
**If the printing speed is faster, the printing temperature should be higher and the layer should be thinner;**  
**If there is stringing and overhang, the printing temperature should be lower;**

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.

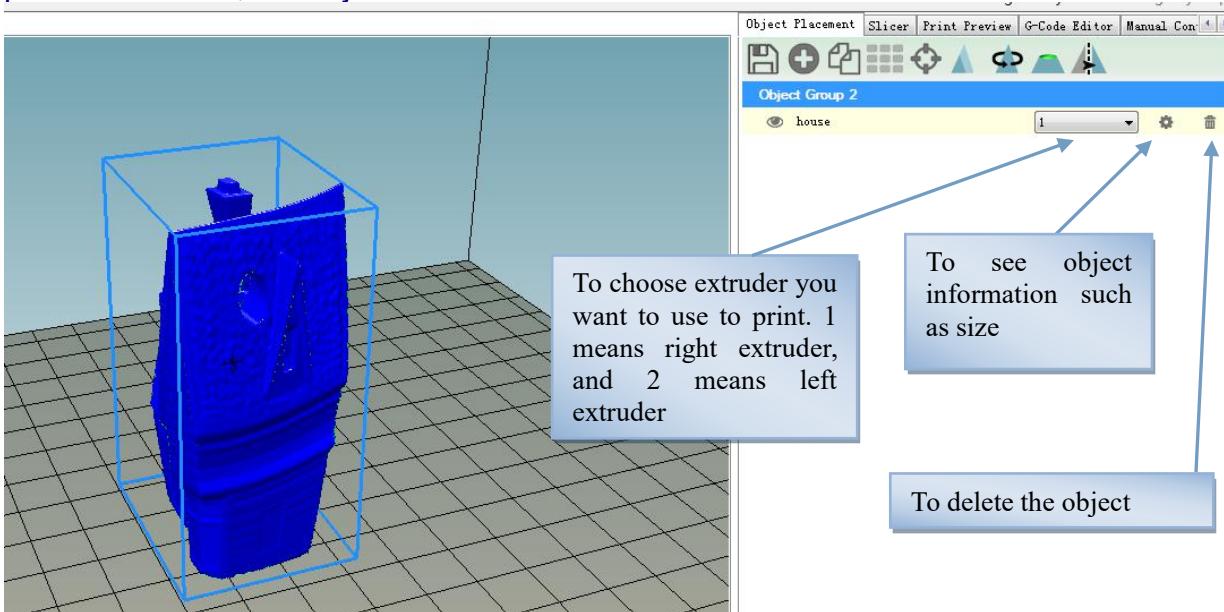
Now you can load the object first. click the “Object Placement” tab, and click “Load” in the left or “+” in the right. Then choose the object you want to print.

For object you want to print can be downloaded free from  
<http://www.thingiverse.com/> or <https://www.youmagine.com/>



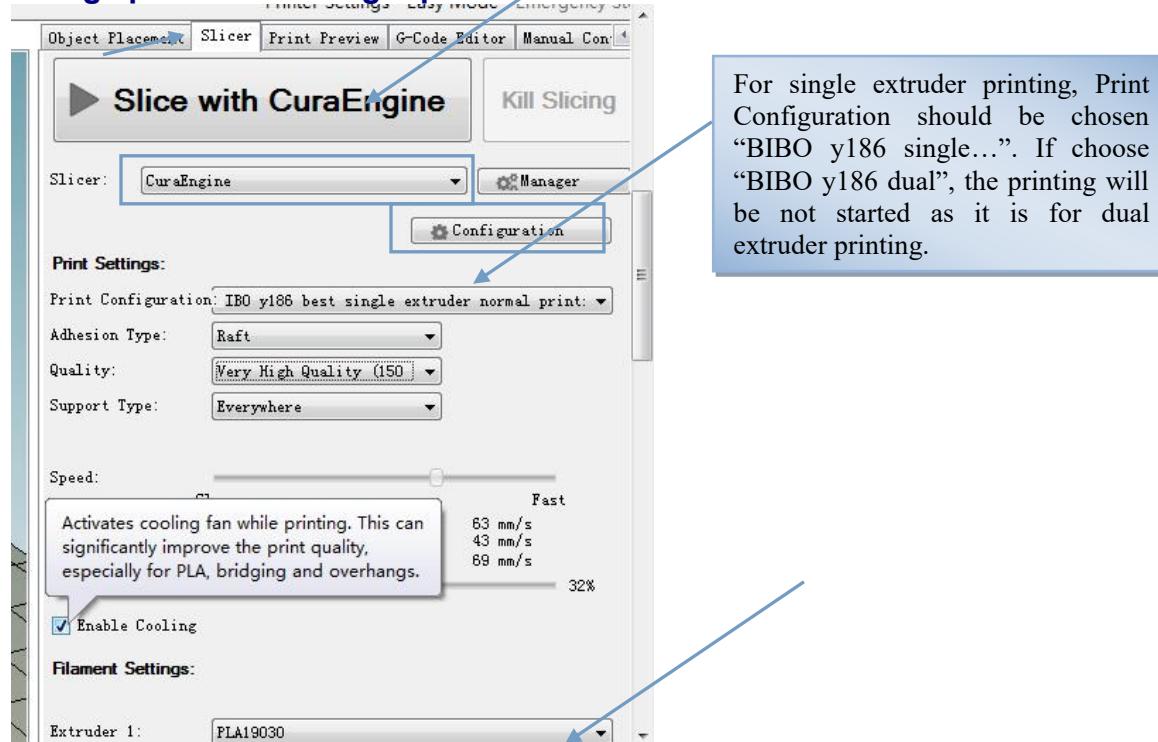


You can center the object on the bed by clicking "center" icon. You can also rotate the object by "rotate" icon. When you move the mouth arrow to every icon, repetier-host will show what this icon mean. What you want to do is let object more printable, such as center the object, less support to be printed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

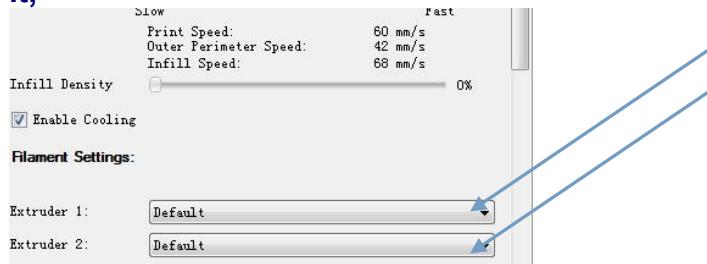


Then go to Slicer tab and choose the right settings. For every settings, you just move the mouth arrow above every settings, and the meaning will be showed. Such as you move mouth arrow above "Enable cooling", it shows the information as below. When printing ABS, please don't choose Enable cooling.

Please note that we should choose the settings at the slicer tab as the photo below. All settings revisions on the configuration (will be showed on the left part of the software) will be only valid when you choose the corresponding settings profile on the right part in slicer tab.



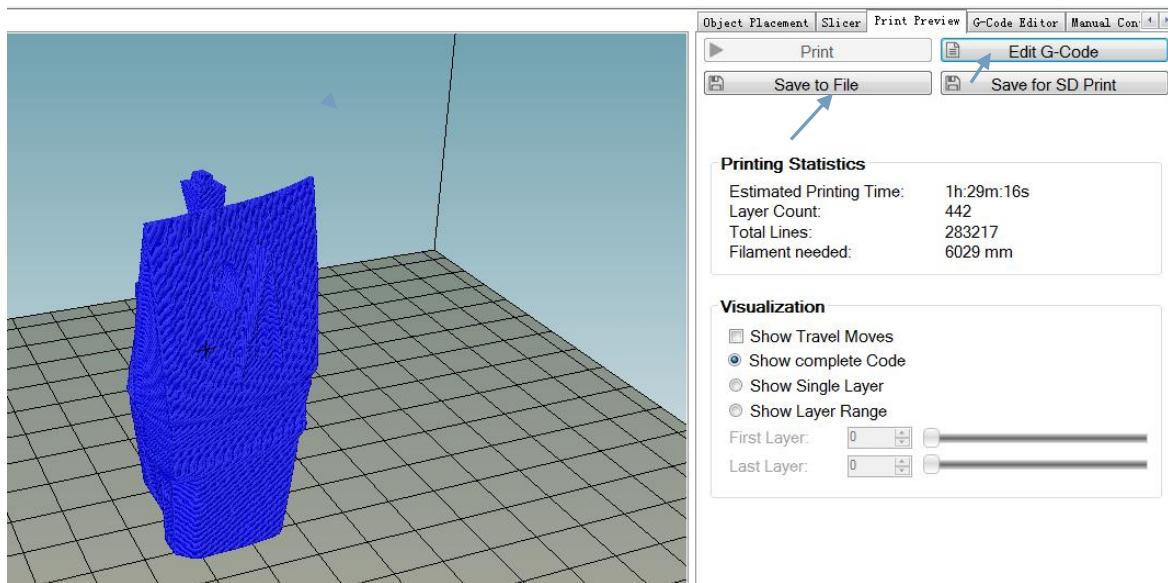
Sometimes you choose the printing temperature in repetier host, but it has a different printing temperature actually during printing. This problem is caused by repetier host's bug. You can choose the default filament setting first and slice it,



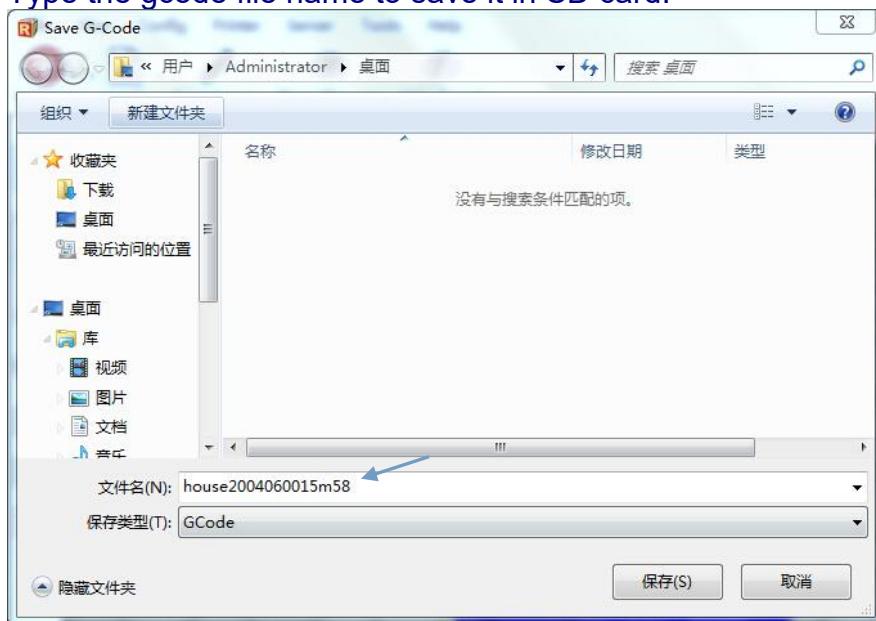
then choose the filament setting you want and slice it. Now your settings or choice will be valid for the slicing. If still not ok, just restart repetier host and it will be ok.

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

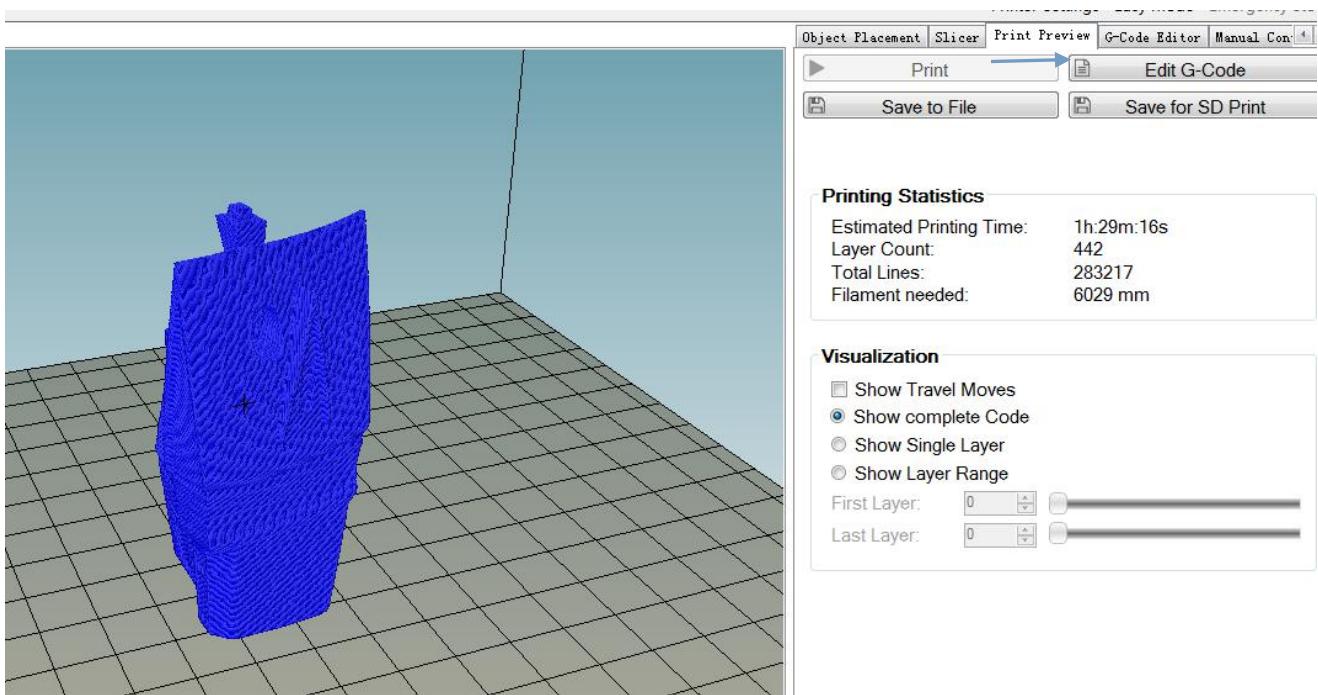
Then go to Print Preview tab, click “Save to File” and you can see as the photo below:



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

```

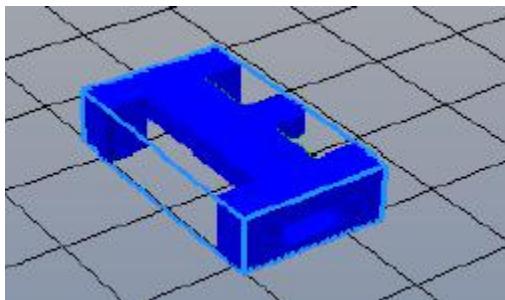
1 ;Generated with Cura_SteamEngine 14.12
2 ; Default start code
3 G28 ; Home extruder
4 G1 Z15 F100
5 M107 ; Turn off fan
6 G90 ; Absolute positioning
7 M82 ; Extruder in absolute mode
8 M190 S30
9 ; Activate all used extruder
10 M104 T0 S180
11 G92 E0 ; Reset extruder position
12 ; Wait for all used 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 F1.17075

```

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

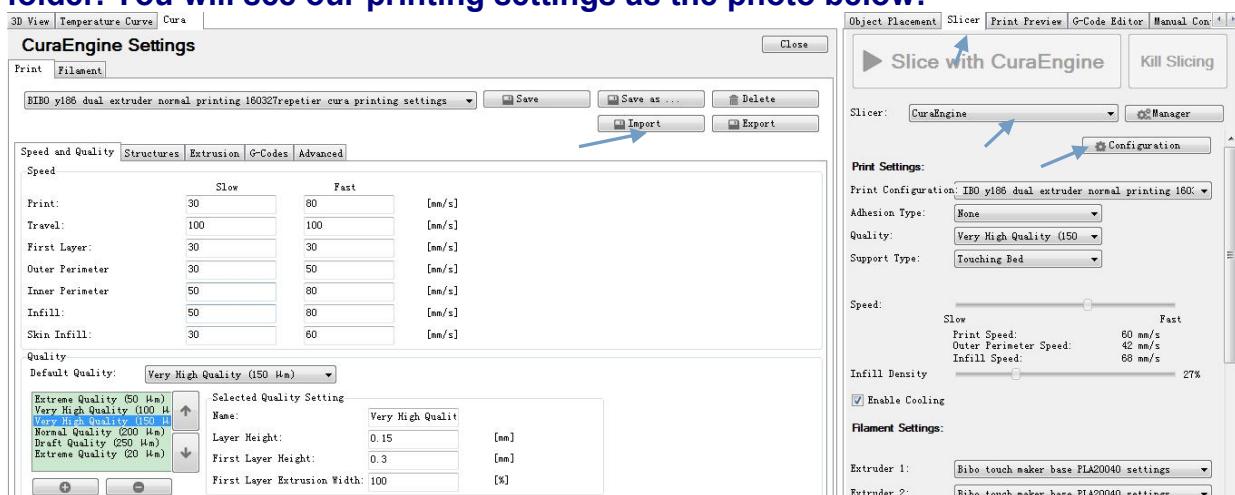
We use this object for reference. This can only be printed well with support material

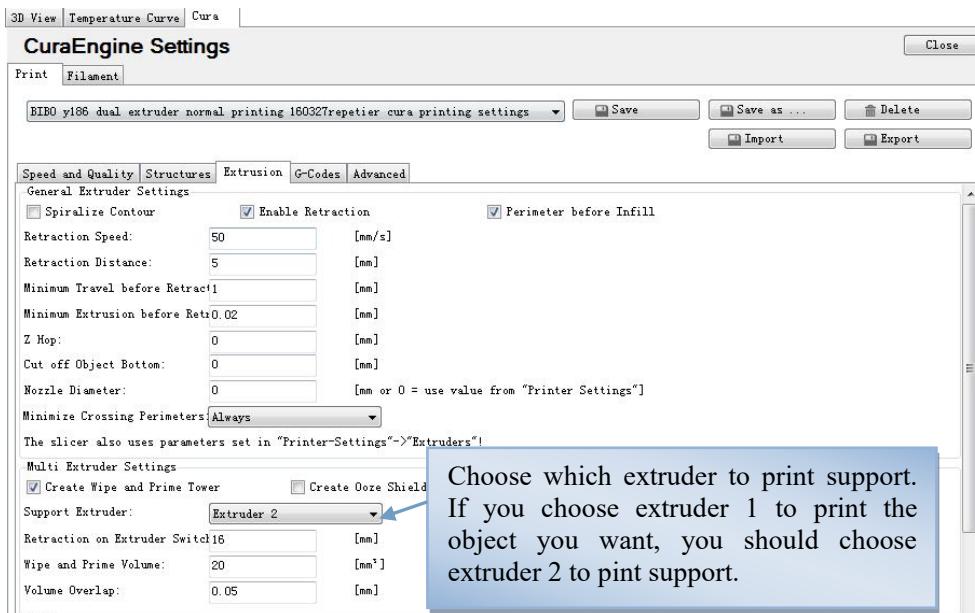


Go to the slicer tab and choose the following:

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:





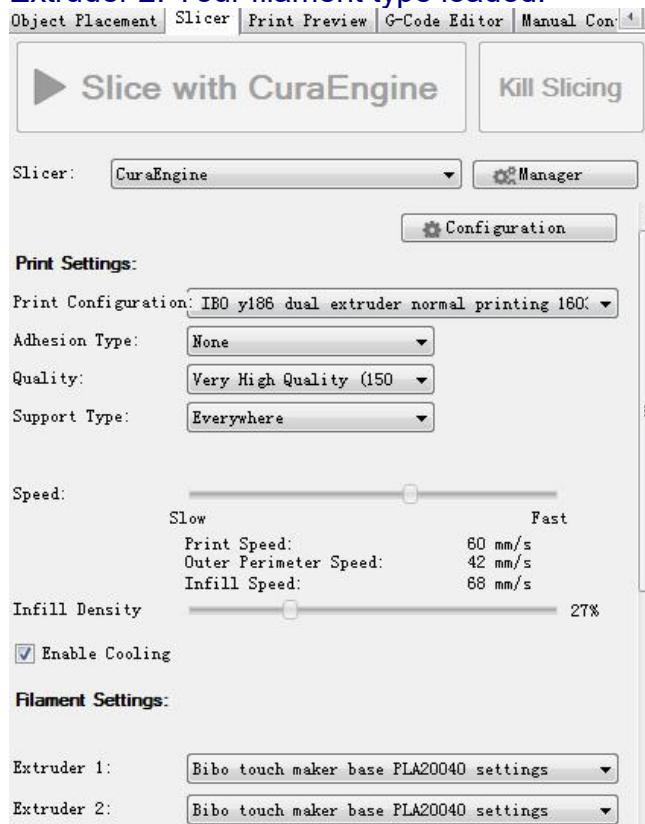
Adhesion Type: ...

Quality: ...

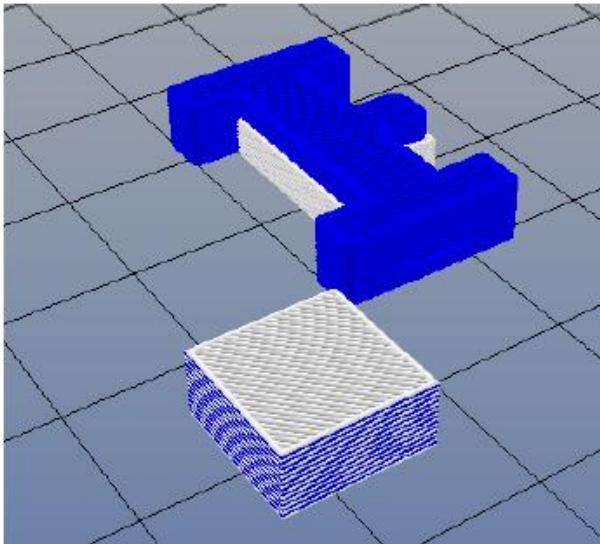
Support type: Everywhere

Extruder 1: Your filament type loaded.

Extruder 2: Your filament type loaded.



Press slice, this should be the result.

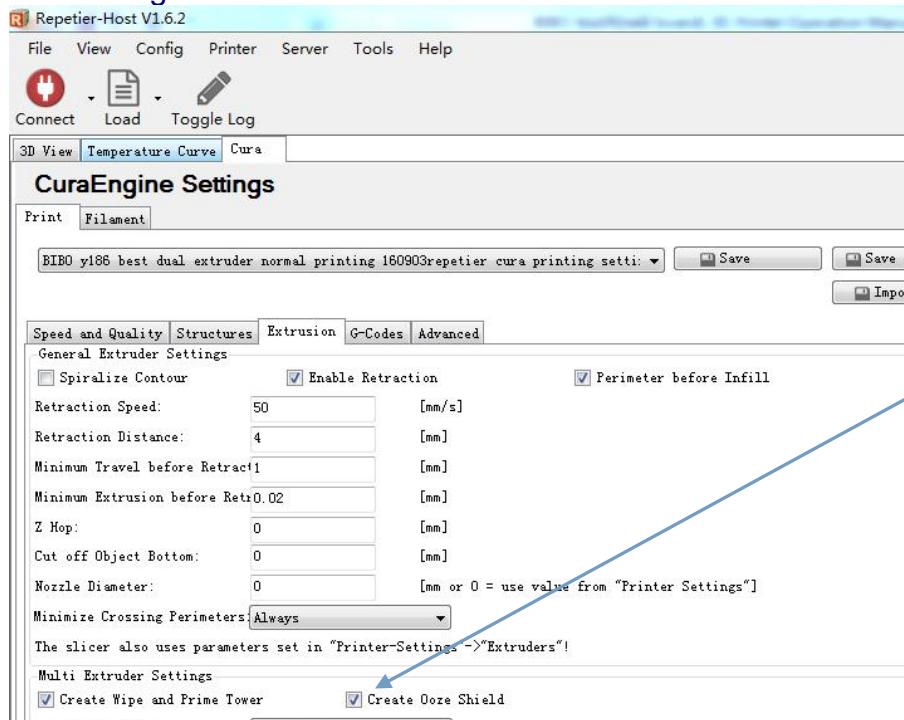


The white colored lines are the support material and the blue lines are the object itself.

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:

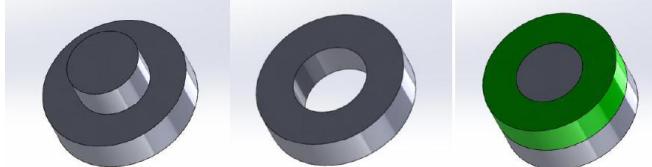


### Assign an extruder to an object (two colors printing)

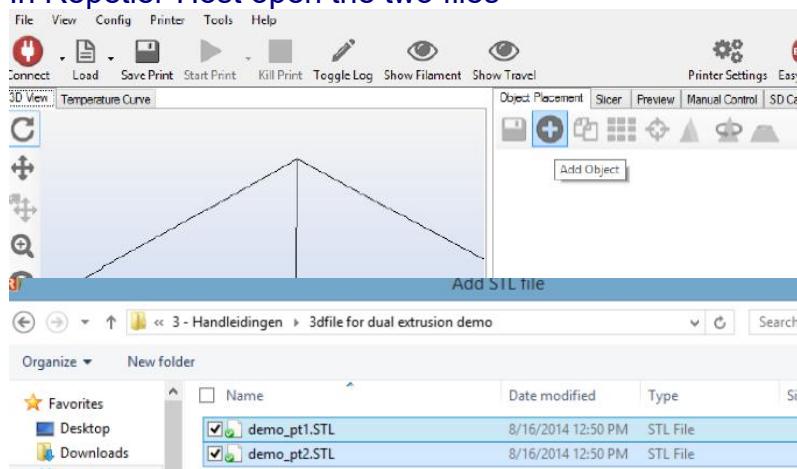
This short tutorial will show you how to **print two objects in a single print with different extruders**.

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.

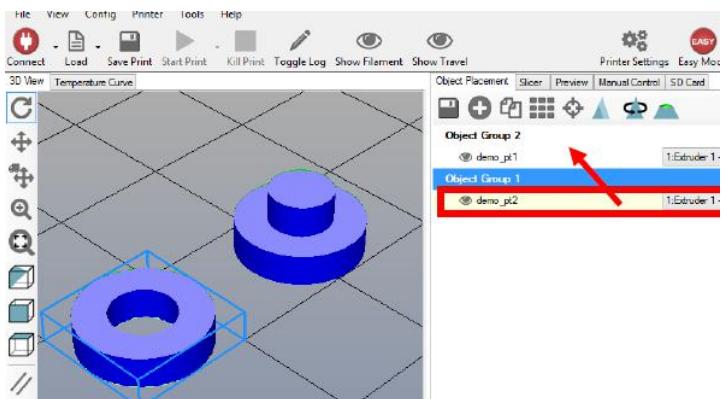


In Repetier-Host open the two files

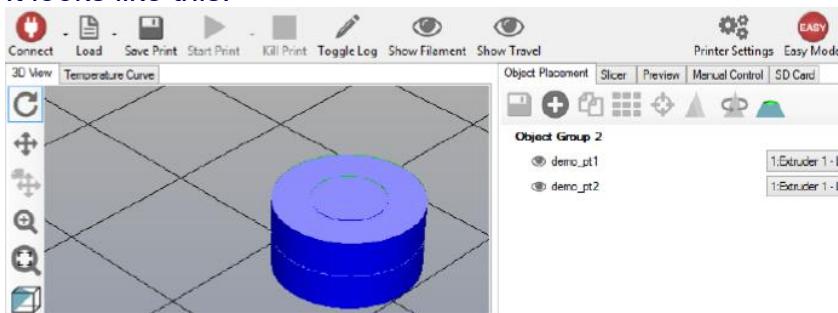


It should now look like this:

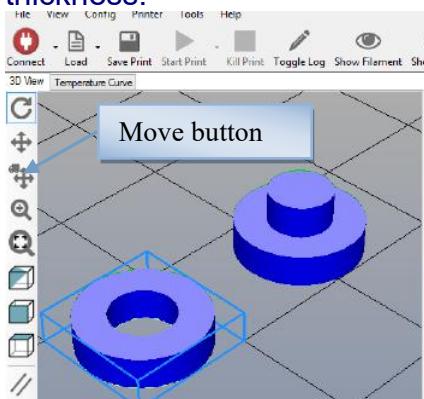
Select demo\_pt2 and drag it into object group 2



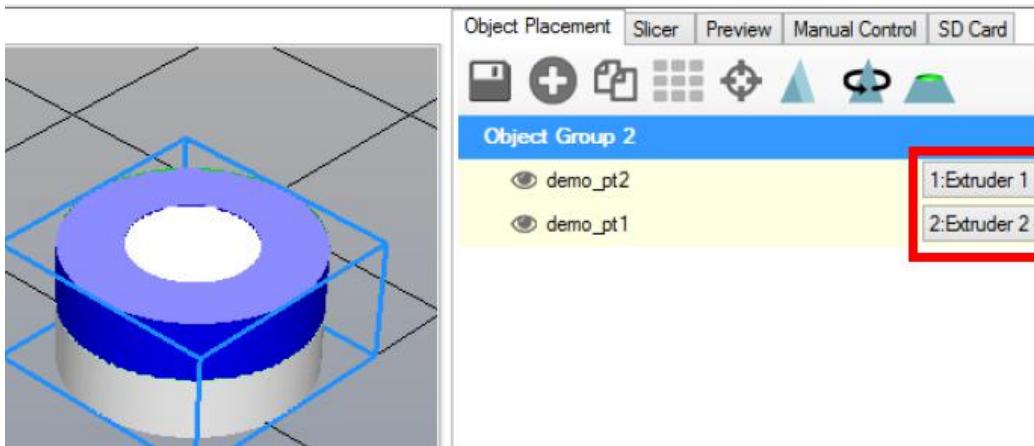
It looks like this:



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.

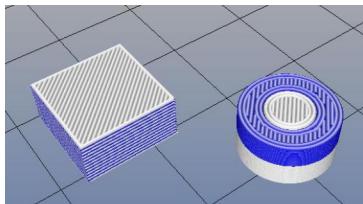


Goto the Slicer tab and select at least the following:

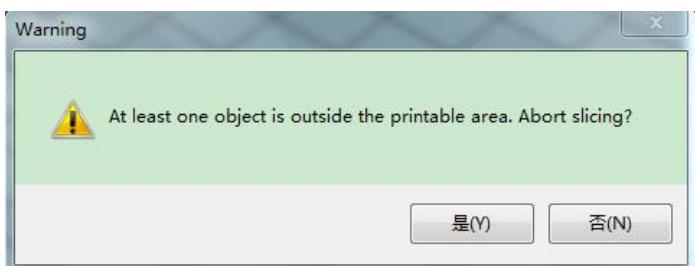
Print Configuration: BIBO dual extruder repetier cura printing settings

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:



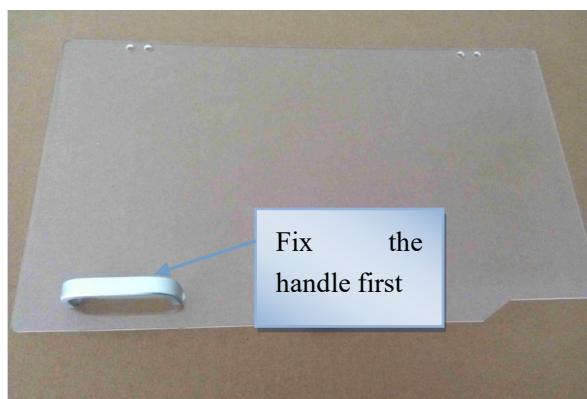
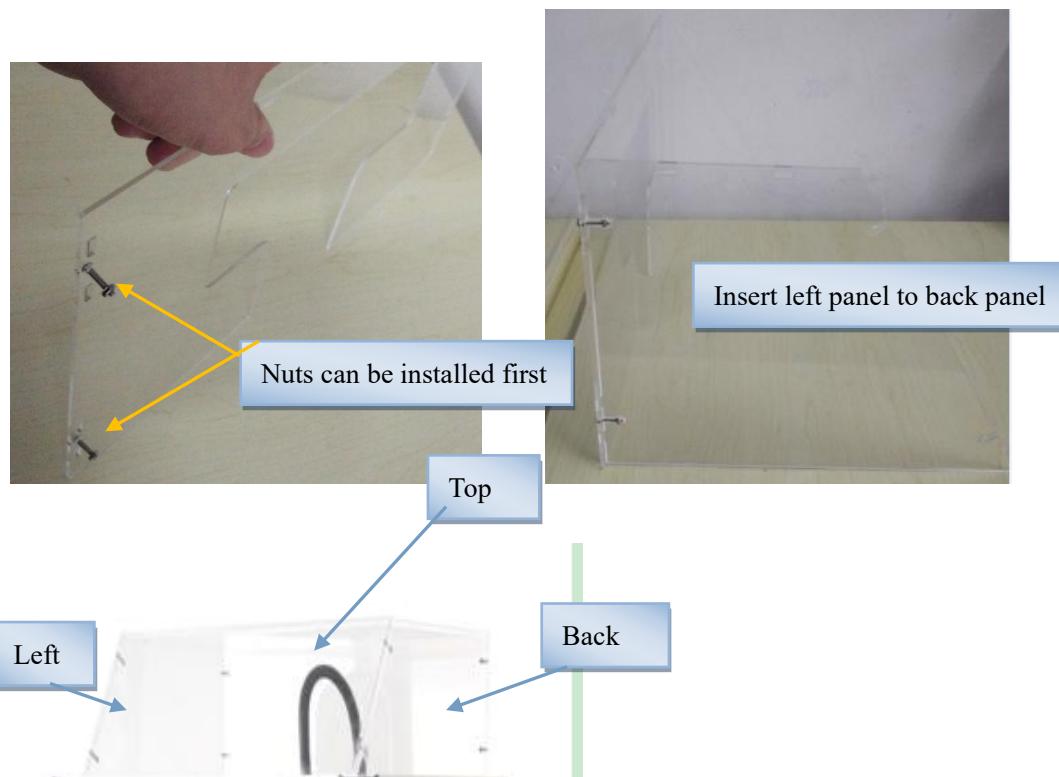
You choose now will be ok.

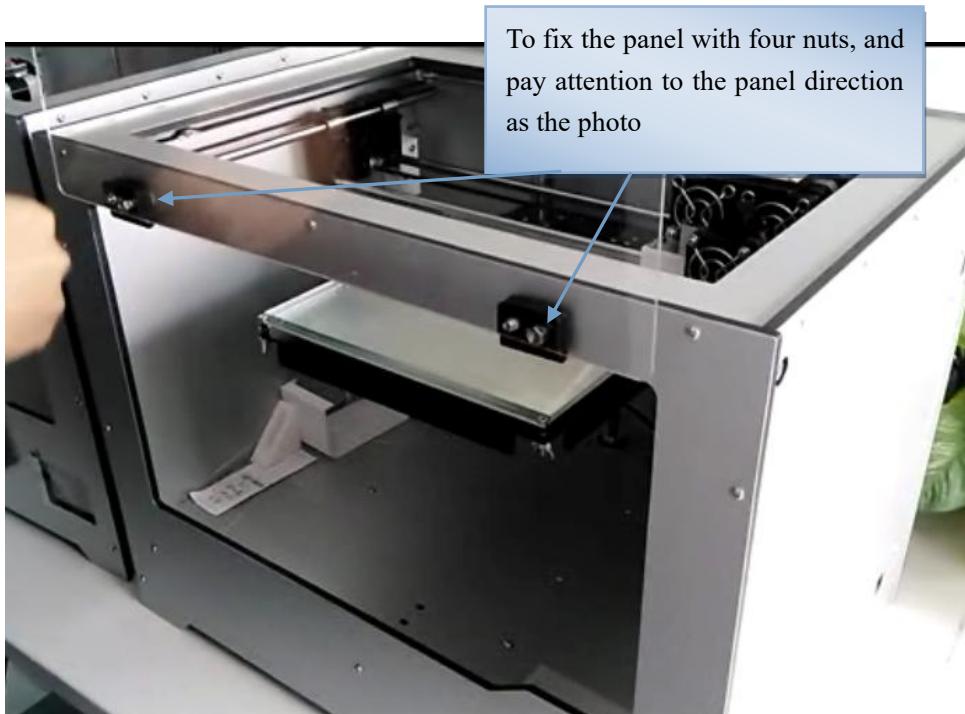
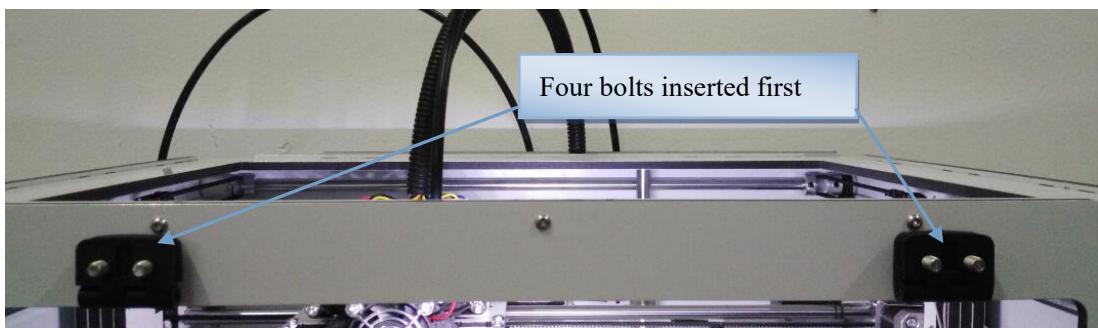
## 8. Acrylic Cover Installation

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. Acrylic panels usually are **red**, and some are **transparent**, and may be packed with brown protective paper. Acrylic clear door may be packed with white protective film.

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.





## 9. Function Introduction(including copy printing)

**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 (text editor).

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 there is already M109 T1 S169 or M109 T1 S170 in the start gcode, please delete this line, or the printing temperature will be low as 169 degrees or 170 degrees. Please check the screenshot below:

```
M420 S1
M109 T1 S190
;Generated with Cura_SteamEngine 15.01
M190 S40
M109 T0 S170
M109 T1 S170 , preheat the other extruder, so it will not
knock or ruin the print
M109 T0 S190
G90 ; absolute mode
G21 ; metric values
M82 ; Extruder in absolute mode
M107
G28
```

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 file.

If there is already M109 T0 S169 or M109 T0 S170 in the start gcode, please delete this line, or the printing temperature will be low as 169 degrees or 170 degrees. Please check the screenshot below:

```

M420 S1
M109 TO S190
;Generated with Cura_SteamEngine 15.01
M190 S40
M109 TO S170
M109 T1 S170 ;preheat the other extruder, so it will not
knock or ruin the print
M109 T1 S190
G90 ; absolute mode
G21 ; metric values
M82 ; Extruder in absolute mode
M107
G28
G1 Z2 F100
T0

```

**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:

```

M420 S1
M109 TO 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/>

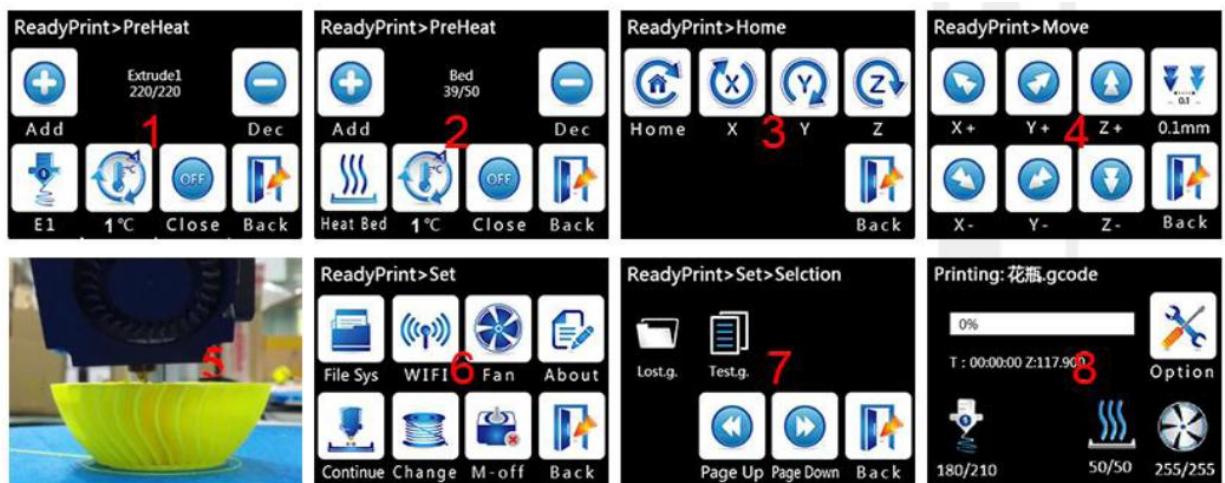
Cloud printing hasn't been opened yet. If you want to control the 3D printer from everywhere, you can also use software Teamviewer to control your 3D printer by connecting 3D printer to PC (details are in the next chapter). This

software can be downloaded on <https://www.teamviewer.com>

### Continue printing after stopping the single extruder printing by mistake

If you stop the single extruder printing by mistake, you can use this function to save the model already printed as the following steps with photoes:

1. Preheat the extruder to arrive the target temperature and extrude it for 1mm in “Extrusion” menu on the touch screen's main menu to **active this extruder**.
2. Preheat the bed to arrive the target temperature.
3. If we have powered off the printer after stopping the printing. Please home x, y, z axis first. **Please note that if the printed object will hit the X axis shaft after home Z axis (build plate rises), please don't do this step, or the printer can be damaged, and this model can't be saved.** If we haven't powered off the printer after stopping the printing, please **ignore this step**.
- 4-5. Move x , y, z axis to let the actived nozzle just above the printed object.
- 6.Click “settings” menu on the main screen to go to “continue” button
- 7-8. Choose the correct printing gcode file and wait the printer to continue the printing. If the object is very big and complicated, that means you have to wait for more time.



## 10. USB Driver Installation (not necessary)

Please make sure the plug socket in your room are well grounded, so the printer and the computer are both well grounded too. Please also make sure the USB port on the computer is ok and stable. If not, the USB port on the printer and computer and USB cable will be burnt. We recommend to power off the printer first, then plug the USB cable.

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, and **choose correct com port and baudrate (250000 or 115200)** in the slicing software.

When you connect the printer to the computer by usb cable, [please disable WiFi function first \(click Settings-WiFi-Disable on the touch screen\)](#), and [let the touch screen stays on the main screen after the printer is powered on, and the communication for USB port will be ok.](#)

[Please note that USB connection is not stable as SD card directly printing. The longer the usb cable is, the worse connection quality. It also needs stable USB port of the computer and the computer should be always on and no crash or sleep during printing. That's why we recommend the customer not to use USB printing in our operation manual.](#)

1. Power on the printer and disable WiFi function first (click Settings-WiFi-Disable on the touch screen)
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:

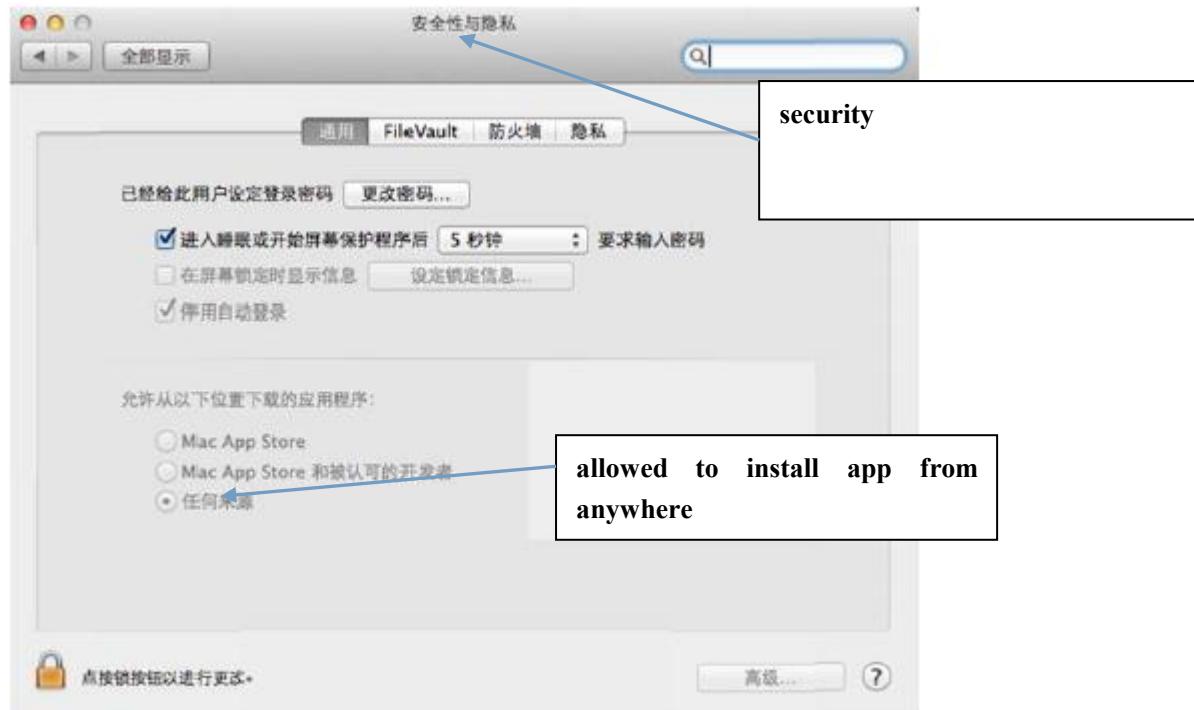
[http://wch.cn/download/CH341SER\\_EXE.html](http://wch.cn/download/CH341SER_EXE.html) (for windows)

[http://wch.cn/download/CH341SER\\_LINUX\\_ZIP.html](http://wch.cn/download/CH341SER_LINUX_ZIP.html) (for Linux)

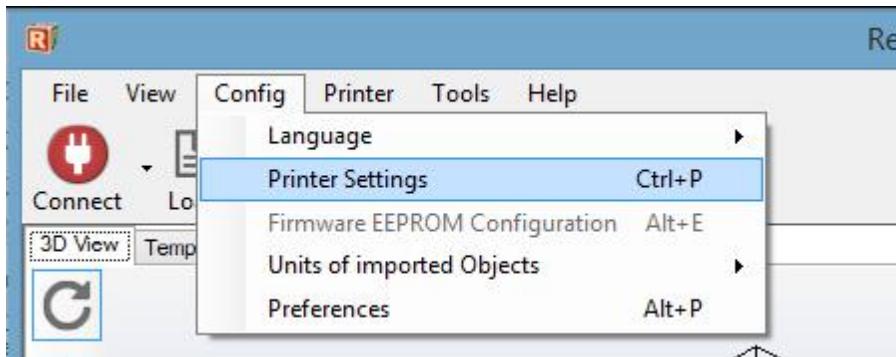
[http://wch.cn/download/CH341SER\\_MAC\\_ZIP.html](http://wch.cn/download/CH341SER_MAC_ZIP.html) (for Mac)

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.



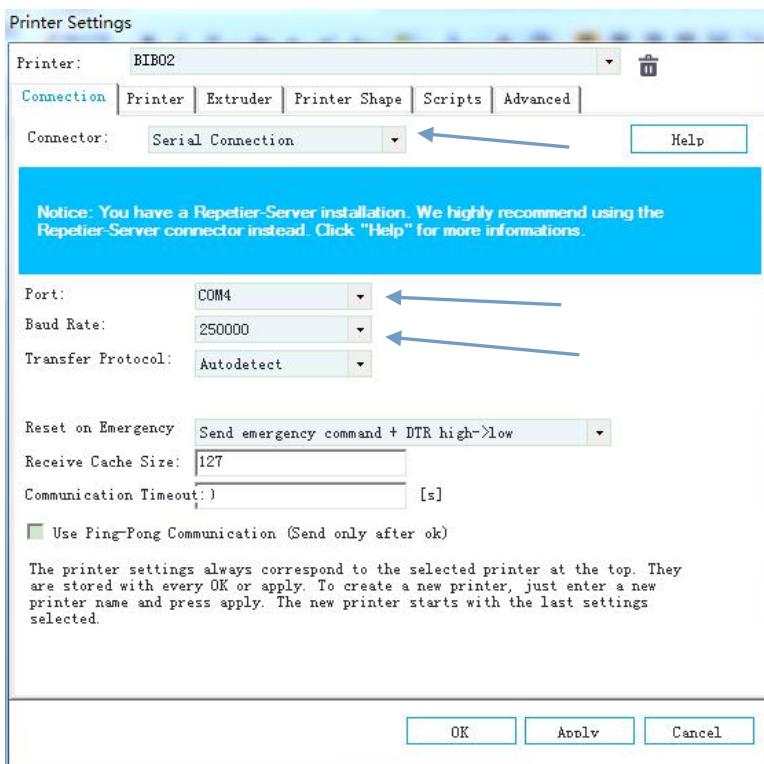
6. Once you see the printer settings window, please fill out the parameter as photo below:

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:

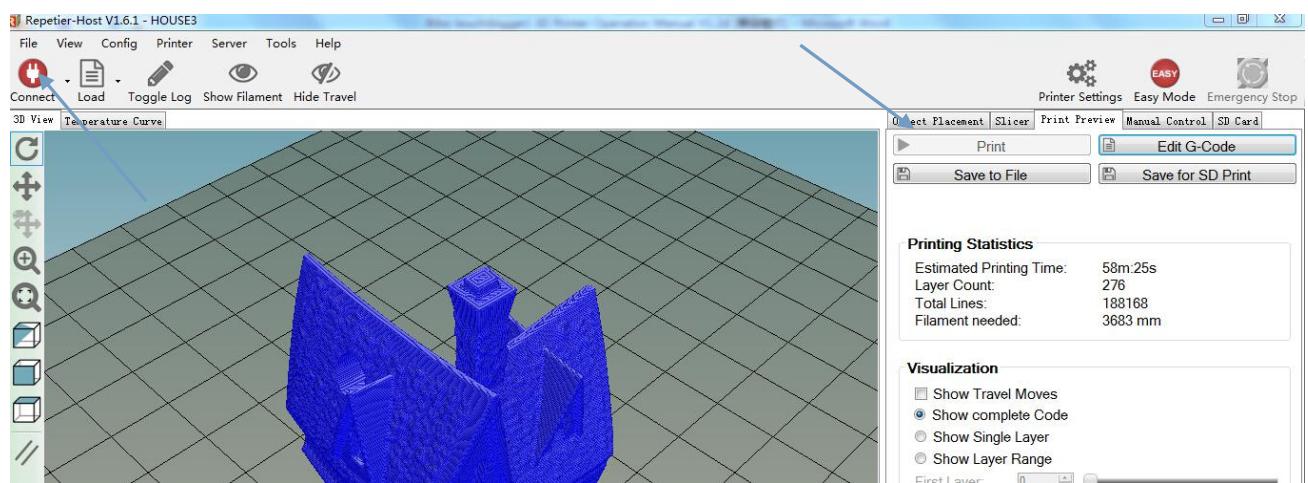
- your printer might not be plugged into the USB port or no power supplied.
- The driver is not installed, or currently installing by windows.

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



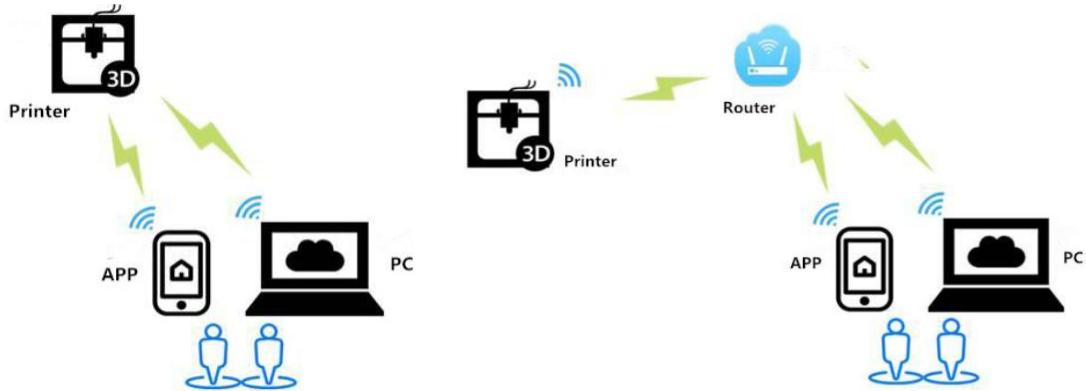
**NOTE:** what COM-port is present for your printer after installation. If there are more than one COM ports available, unplug the USB cable of the printer and then re-plug it in again. Check what port number is appearing and disappearing. This port number is the port present.

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.



**AP mode**

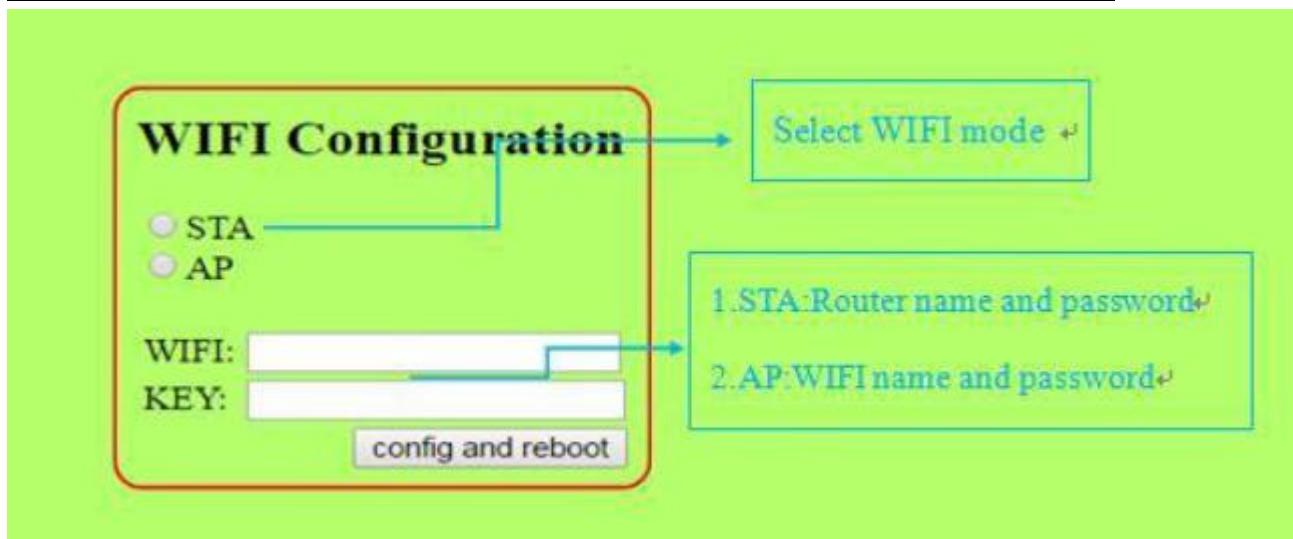
**Client (STA) mode**

Power on the printer, and go to the settings menu on the touch screen, then click wifi, and you will see the wifi name, password, and IP address in AP mode on the touch screen when the wifi status is connected.



Let your computer to connect the printer's wifi. For example the wifi information is the photo above, we should search the wifi named bibo, and then we can connect the printer's wifi on the computer.

Type the IP address showed on the touch screen to the computer's browser. For example the wifi information is the photo above, we should type 192.168.4.1, and the browser will show as below: (STA means client mode)



When you choose the wifi mode and type the name and password and click config and reboot.

Now press "WIFI" button on settings menu on 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).



Let your computer or smart phone connect the wifi name you just wrote for your printer. For example, i choose to connect the wifi "bibo" and type the wifi password , then my computer and the printer are in the same LAN.

#### Use Printron to control printer through WIFI

For Windows, Linux and Mac users, please kindly visit  
<http://reprap.org/wiki/Printron>

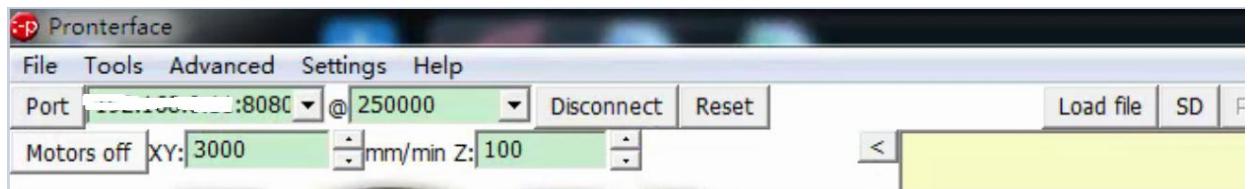
and download the printron: For windows:  
<https://drive.google.com/file/d/12FWbHQL1-WARaMIkSAJmNLaDz5ZcnO8u/view?usp=sharing>

For Mac:

[https://drive.google.com/file/d/1Mn31YhkLU90-a0pfACXG6LnfhKHz5glw/view?  
usp=sharing](https://drive.google.com/file/d/1Mn31YhkLU90-a0pfACXG6LnfhKHz5glw/view?usp=sharing)

Type wifi IP:8080 at port (for example, 192.168.4.1:8080)

Baudrate should be 250000 (some may be 115200), then connect



You can use Printron to print the files in the printer's SD card (click SD button on the Printron). If printing the files directly on the computer through Printron, that will be not stable.

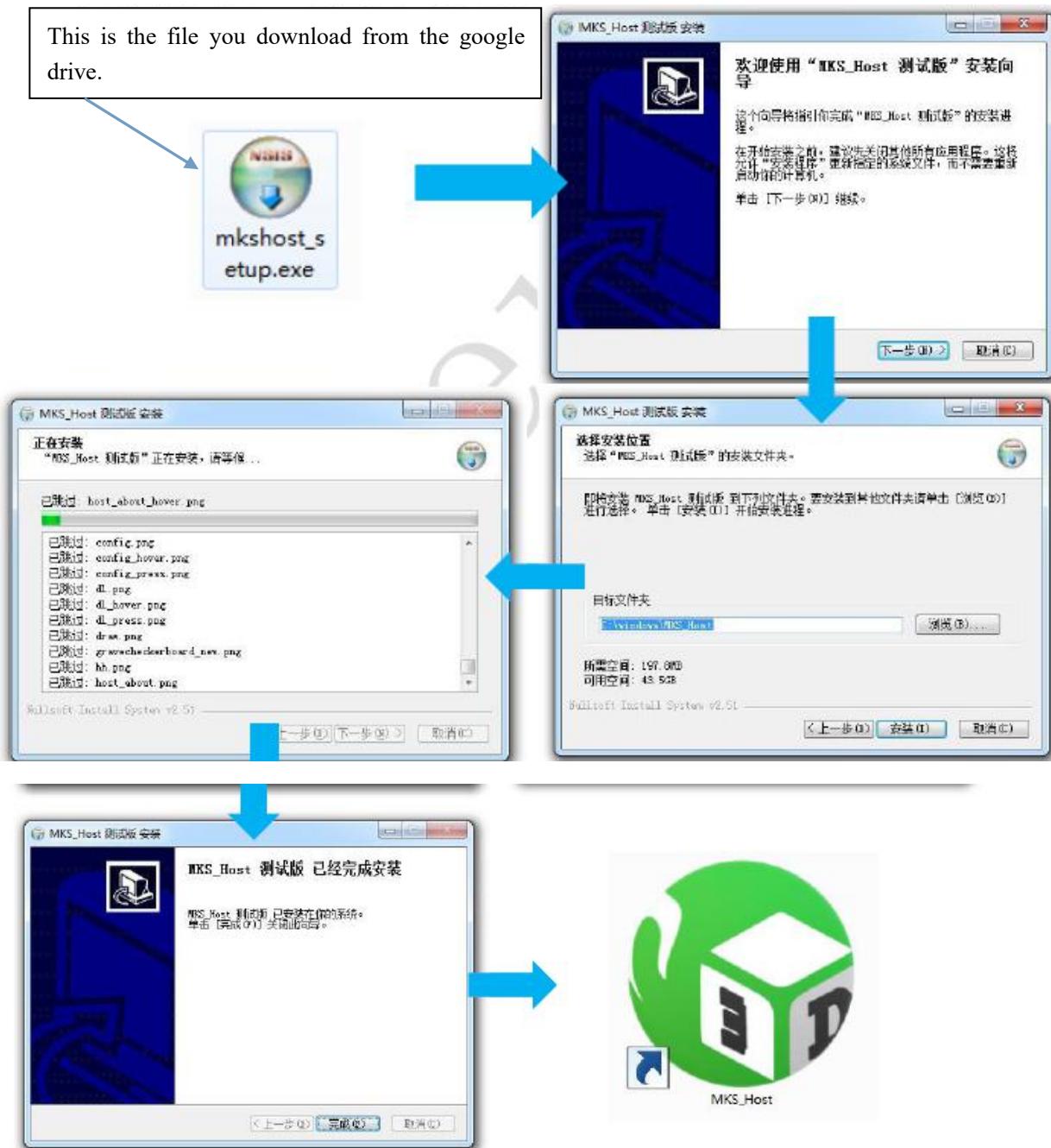
We can use **mks host** (only for windows) or **Cura** to transfer the gcode file to the SD card on the printer if you have the wifi version showed on the touch screen (click settings—about on the touch screen). That means we don't have to plug the SD card on the computer to copy the gcode file and then plug the SD card back to the printer. **Please kindly note that Wifi printing or USB printing are not stable as the SD card direct printing. We recommend SD card direct printing. You can try the both way (Mks host or Cura) to check which way is more stable for transferring gcode file through Wifi.**

#### A. Mks host part:

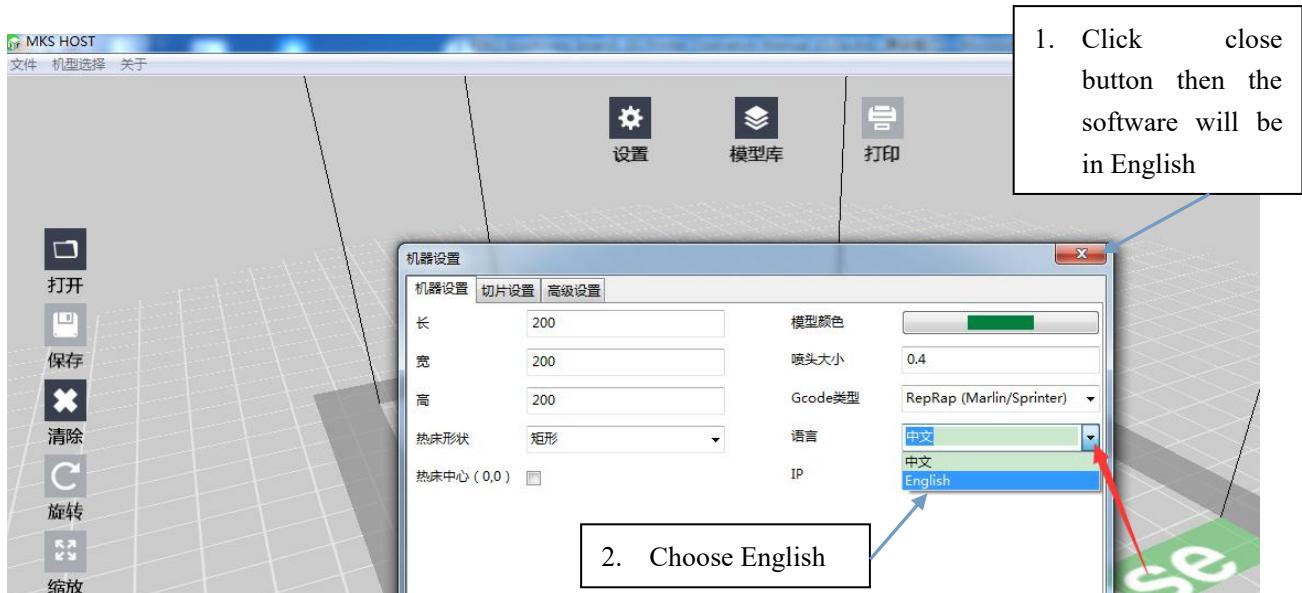
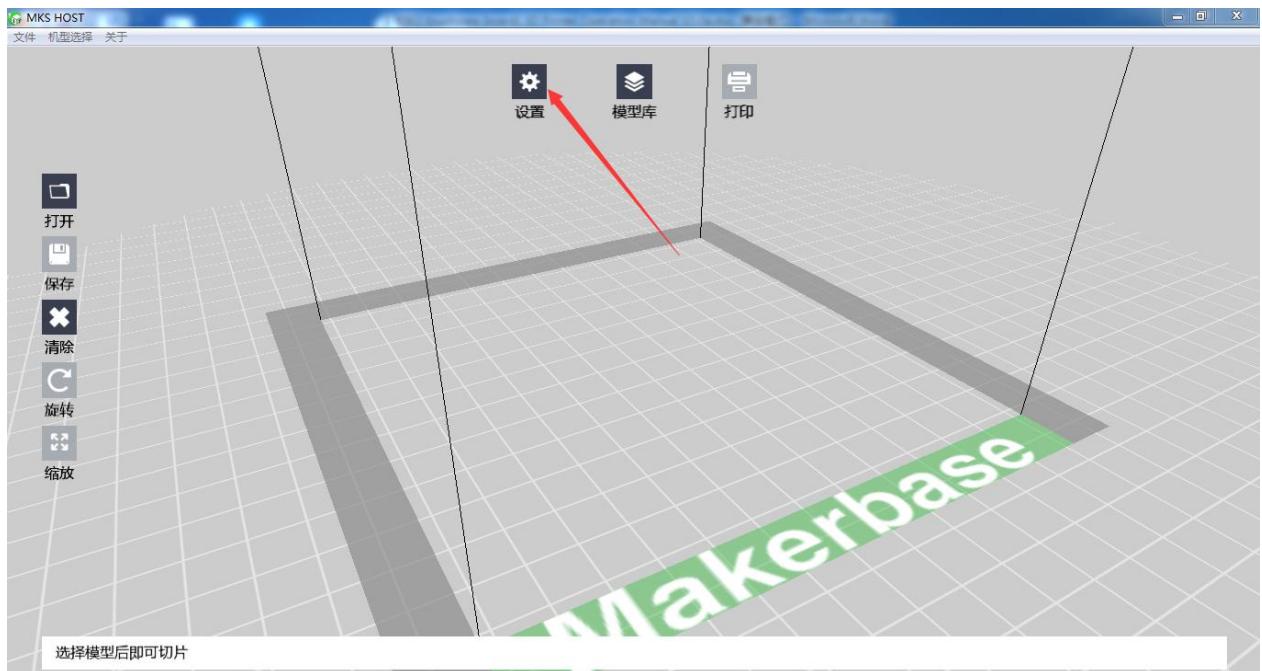
Mks host can be downloaded here for **windows** users:

[https://drive.google.com/file/d/1Z6FMNMeVI15\\_Er4Am-3nt-gFzDFT63TH/view?  
usp=sharing](https://drive.google.com/file/d/1Z6FMNMeVI15_Er4Am-3nt-gFzDFT63TH/view?usp=sharing)

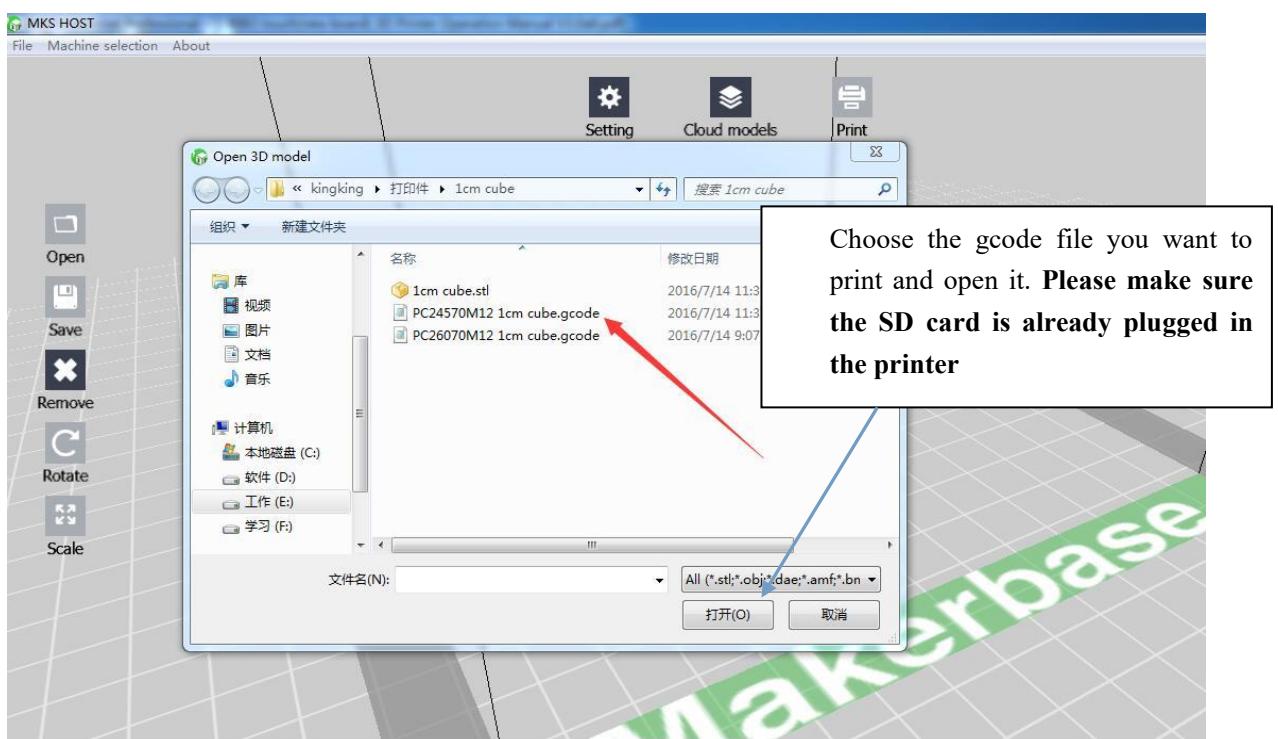
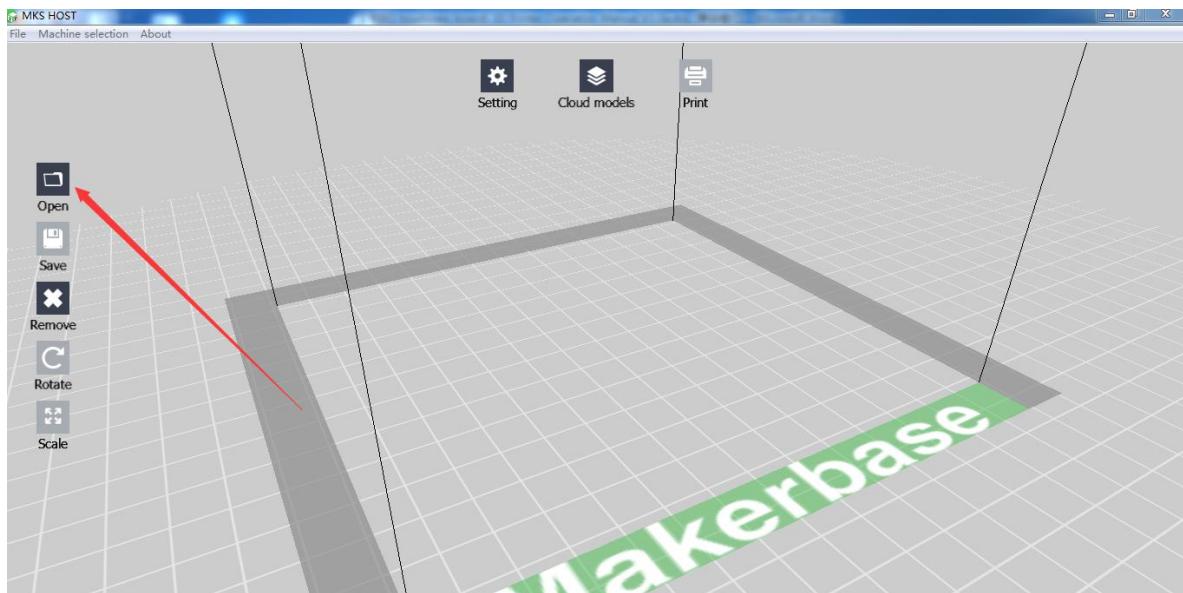
Mks host is not good at slicing now, so we just use its wifi transfer gcode file function. The installation steps are below(now are in chinese):

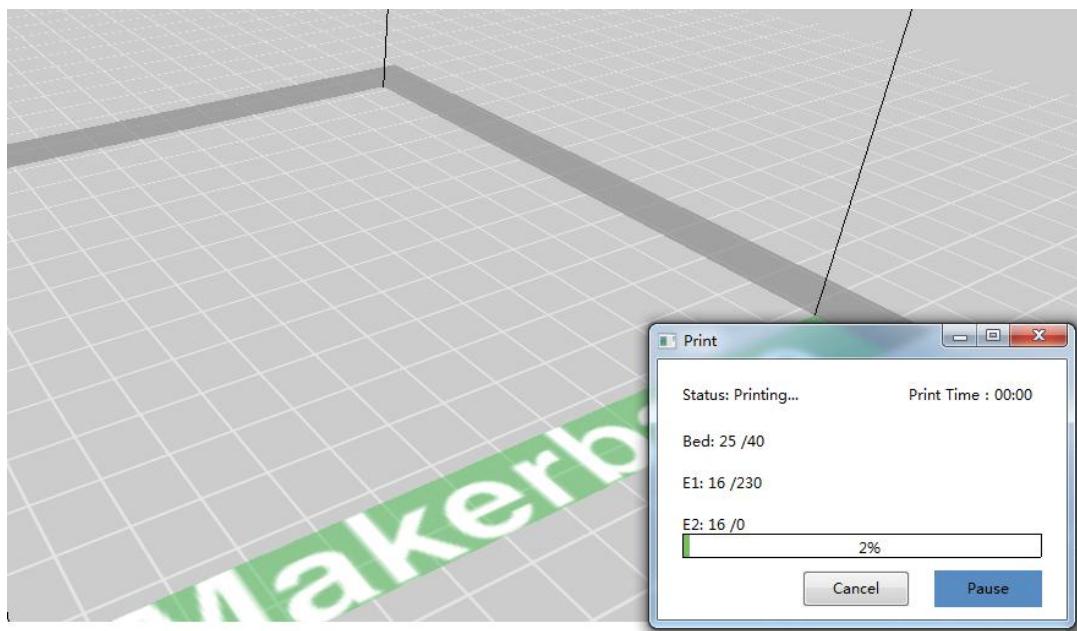
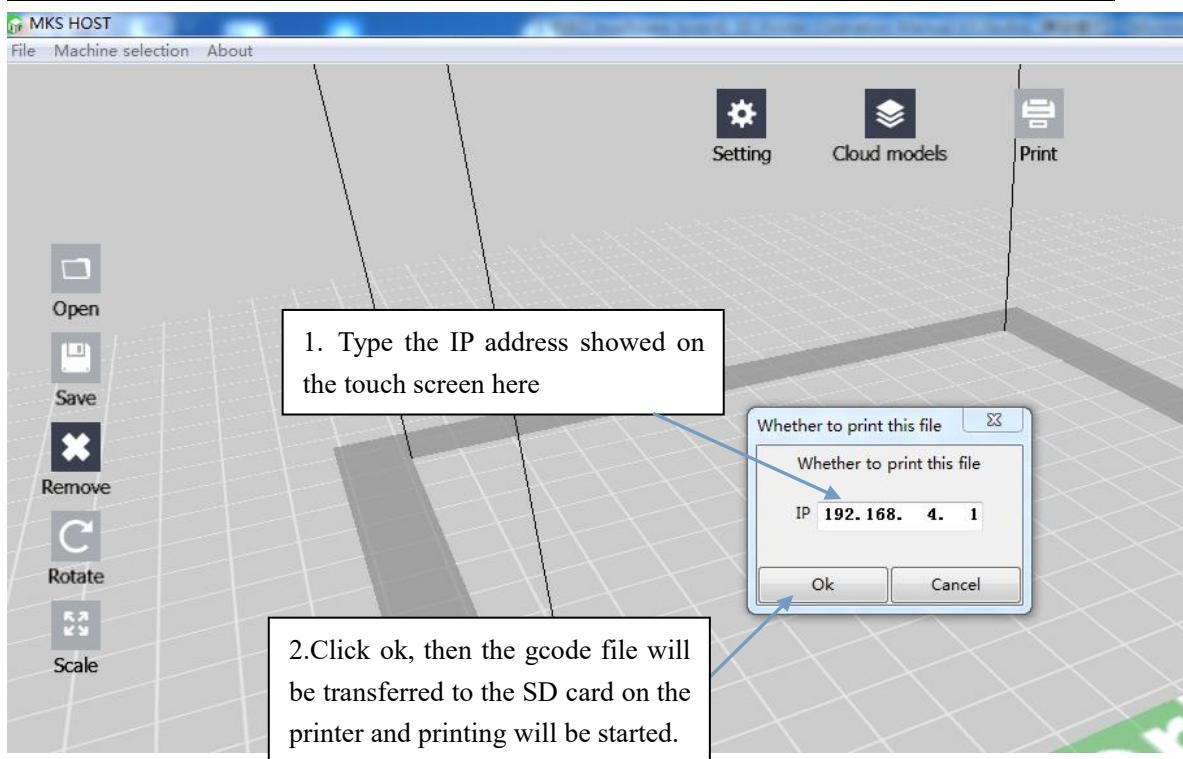


Then double click MKS\_Host on the desktop of the computer,



Click "open"button to load the gcode file you have sliced on your computer.





### B. Cura part:

Download the Cura (should be the version 3.5.1 or 3.6 or 4.0 or above 4.1) from here: <https://github.com/Ultimaker/Cura/tags>

If you just want to transfer the gcode file to the printer by cura and you are using other slicing software, please skip this step. If you want to use cura to slice the model then transfer the gcode file to the printer through wifi, please kindly check the configuration steps for BIBO 3D printer here:[https://drive.google.com/file/d/1xgZoQ2T6e91rUAsjxTB\\_gFdoacw8eSUZ](https://drive.google.com/file/d/1xgZoQ2T6e91rUAsjxTB_gFdoacw8eSUZ)

/view?usp=sharing

### There are two ways to install the plugin:

**A:** On your computer, go to the Cura folder.

Windows: Located in 'program files' folder.

Mac: Located in the applications folder.

Open the folder and find the following file path: Ultimaker Cura/plugins

The plugins folder can be found via Help -> Show Configuration Folder inside Cura.

Download the plugin here:

The plugin for Cura version above 3.6 :

[https://drive.google.com/file/d/1-iF7HX\\_x0fYkhYUGqQ3Ibcown\\_N2UEoM/view?usp=sharing](https://drive.google.com/file/d/1-iF7HX_x0fYkhYUGqQ3Ibcown_N2UEoM/view?usp=sharing)

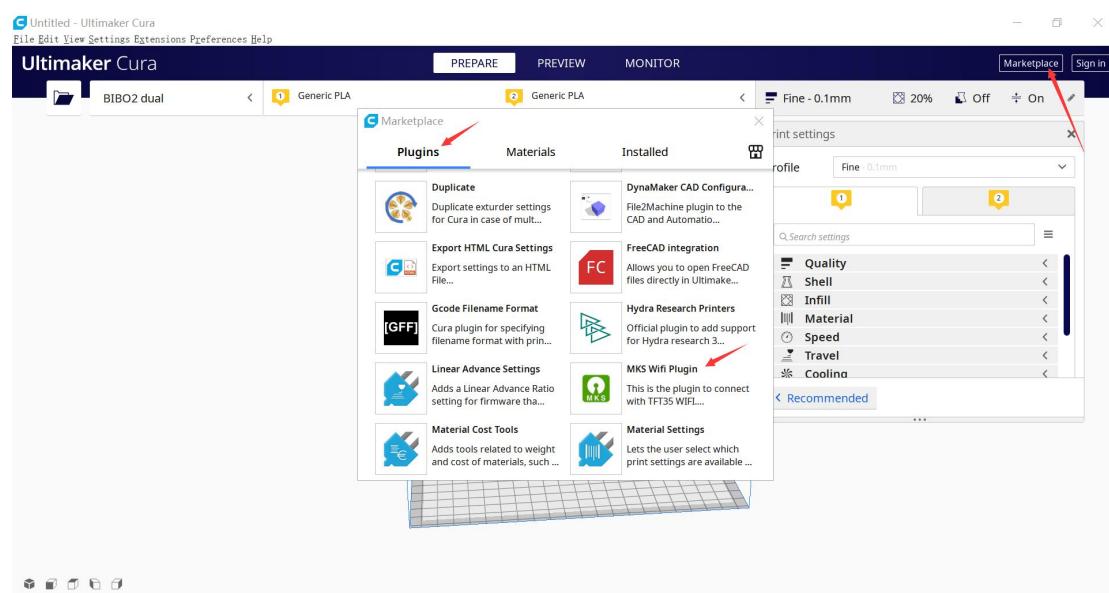
Unzip the plugin, and make sure the plugin file name is MKSWifiPlugin and the subfiles are directly under the file named MKSWifiPlugin .

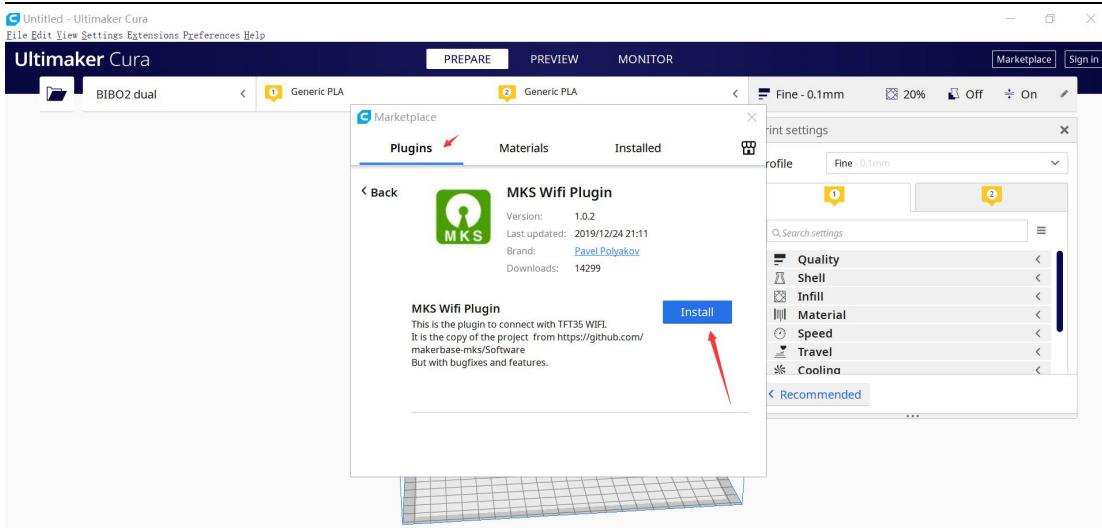
Move the entire folder which is named MKSWifiPlugin into the plugins folder.

(Re)start Cura

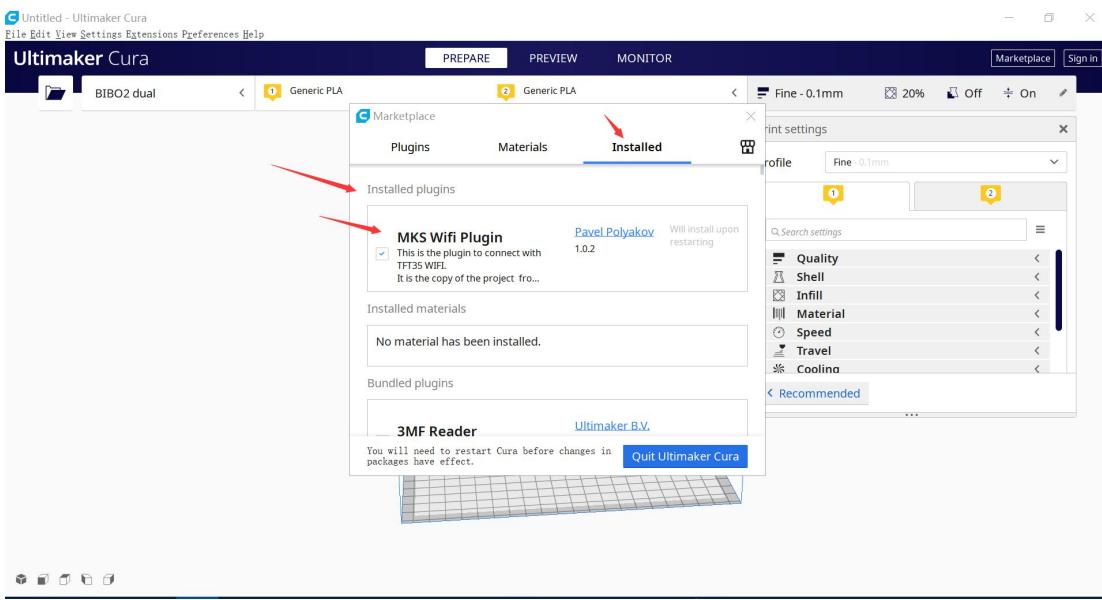
**B:** Another way is to go to cura's marketplace to choose MKS Wifi Plugin.

Usually you have to wait for cura to show these plugins.



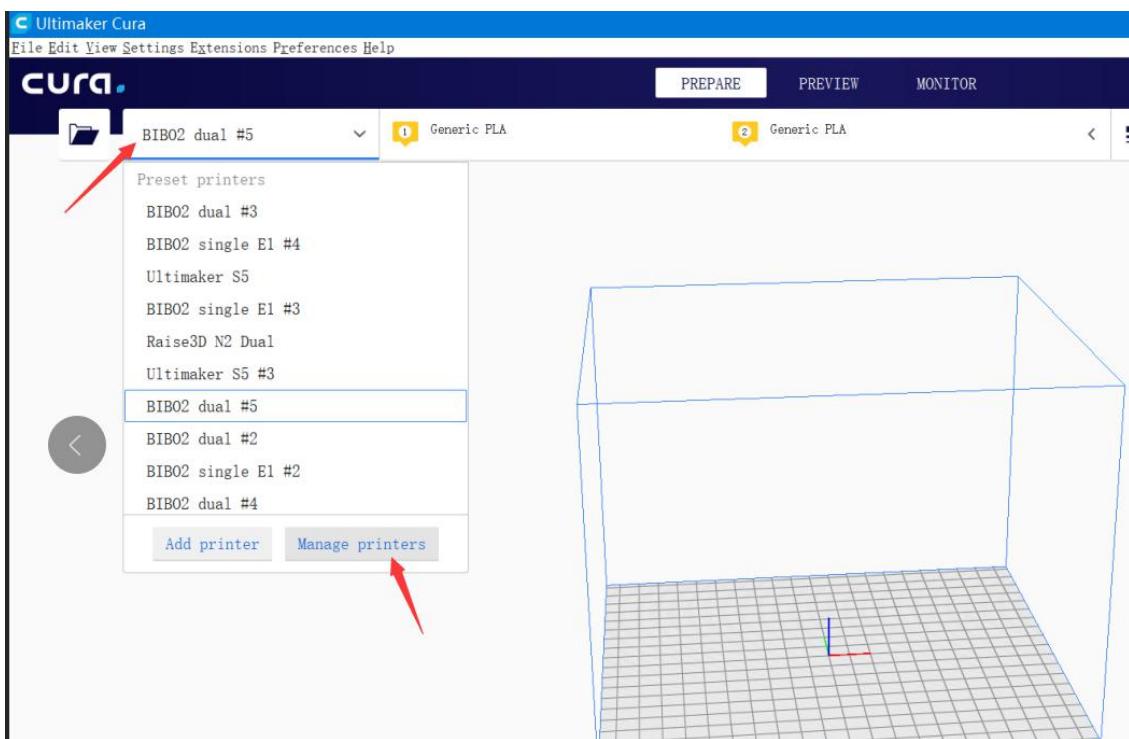


Sometimes it is hard to install the plugin, and please try more times to install it. Once it is installed, you can see it in the installed tab:

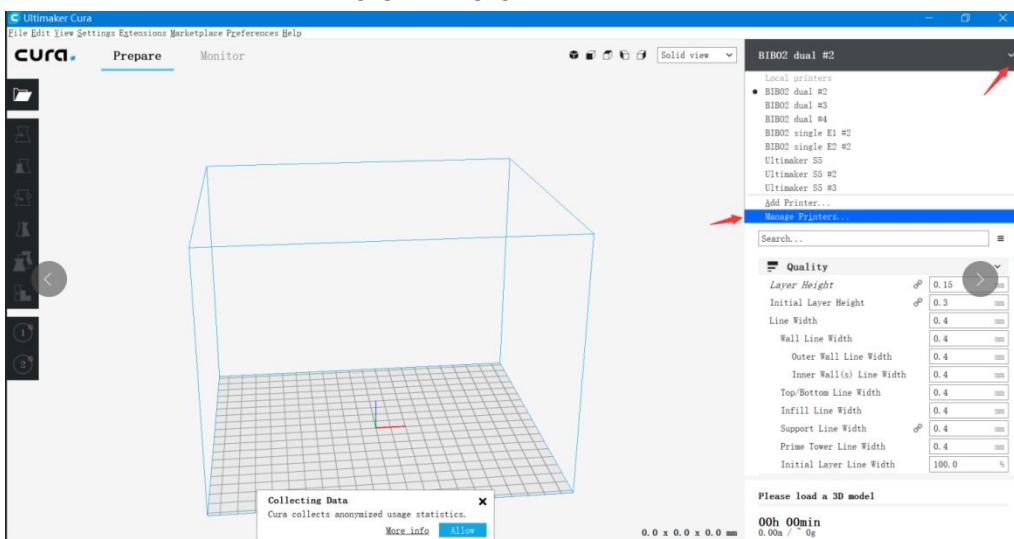


Finally restart the cura and go to the next step.

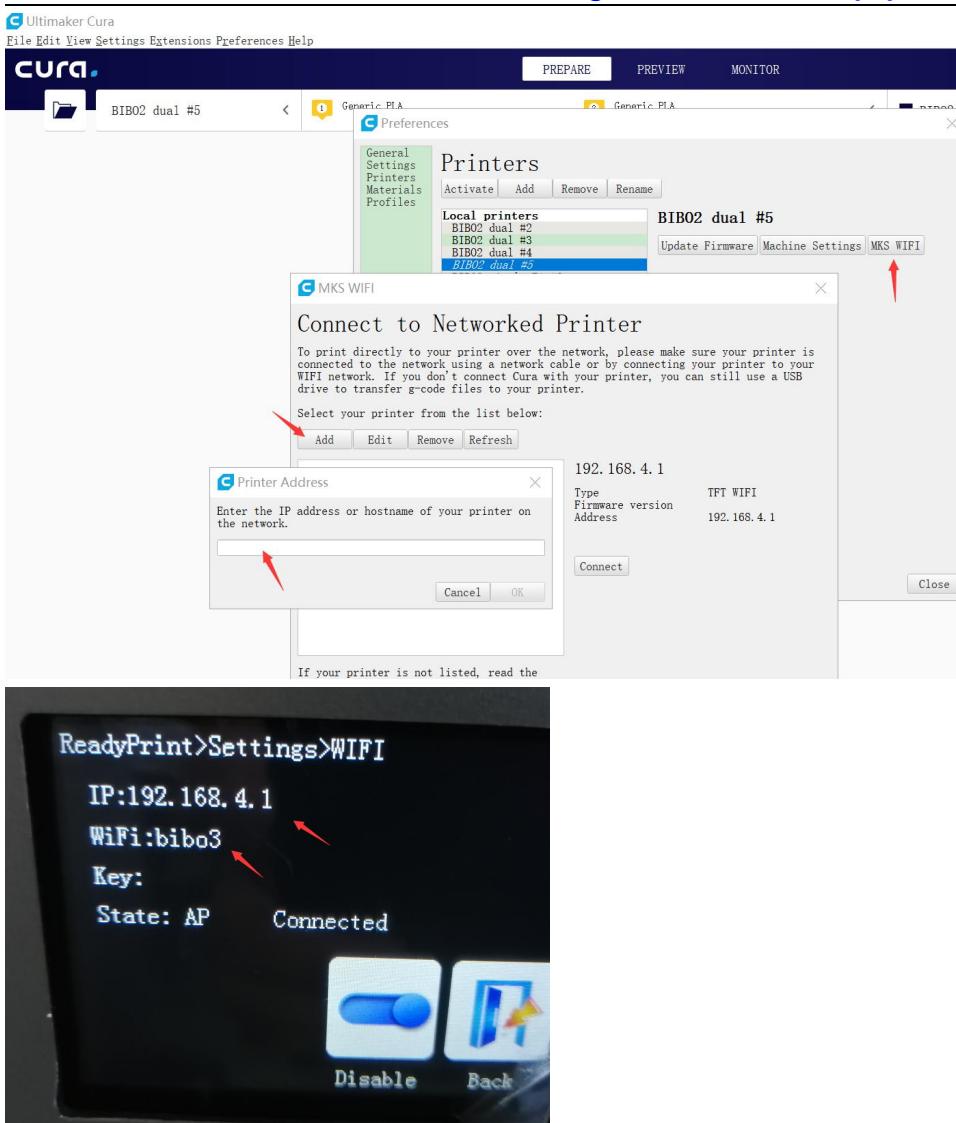
Once the plugin is installed, go to the Manage printers menu.  
The screenshot for cura 4.0 or above:



The screenshot for cura 3.5.1 or 3.6 .



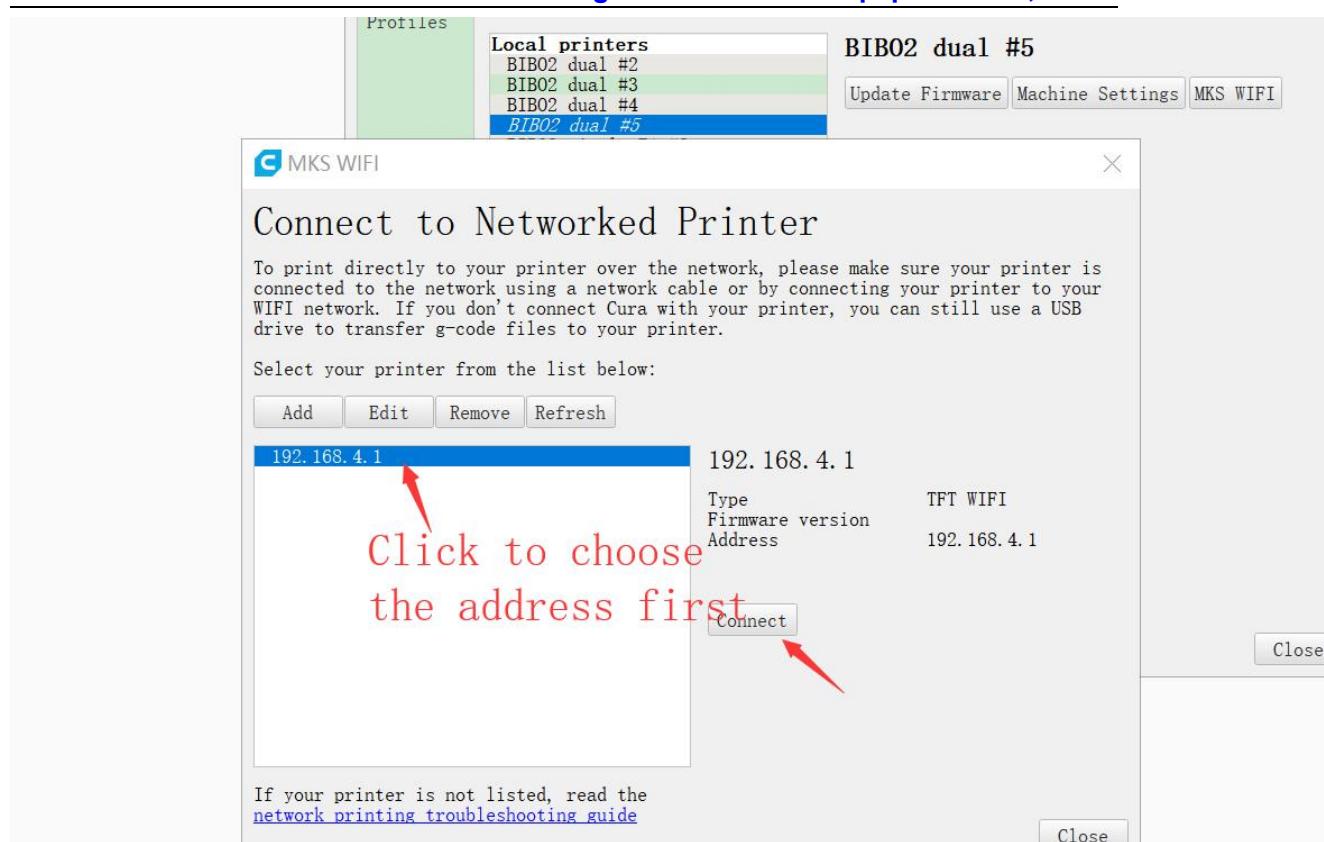
Click "MKSWIFI" tab, add the ip address showed on the touch screen.



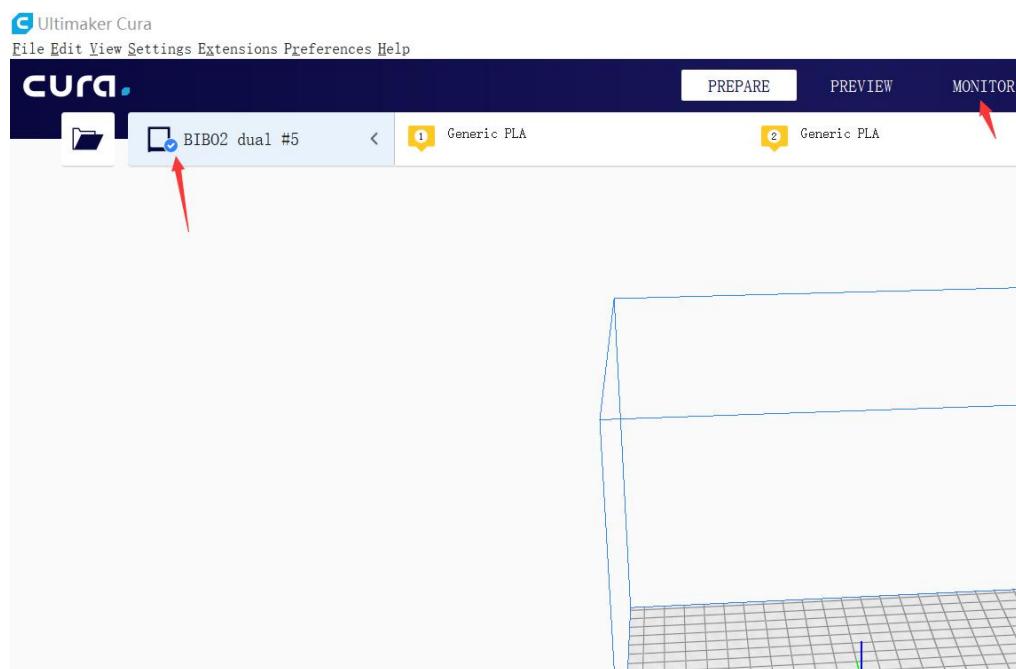
Let the computer connect the same wifi named showed on the touch screen.



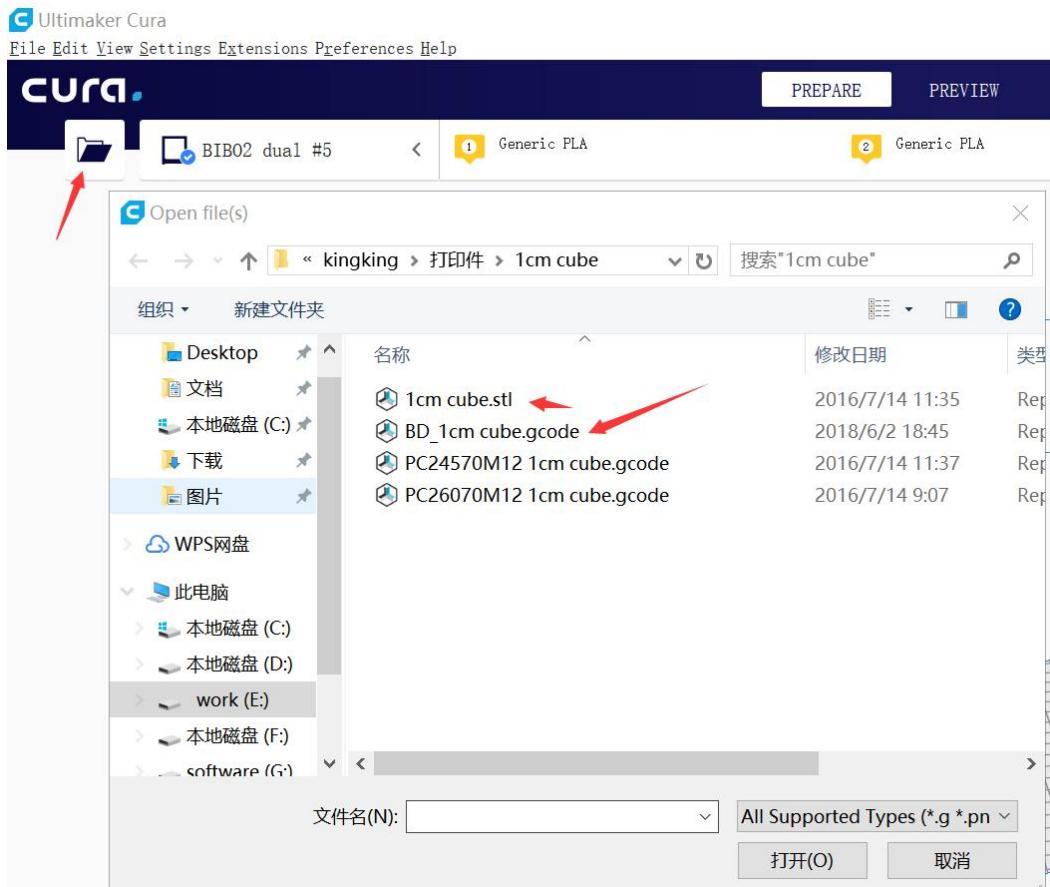
Click the IP address, then click “connect”



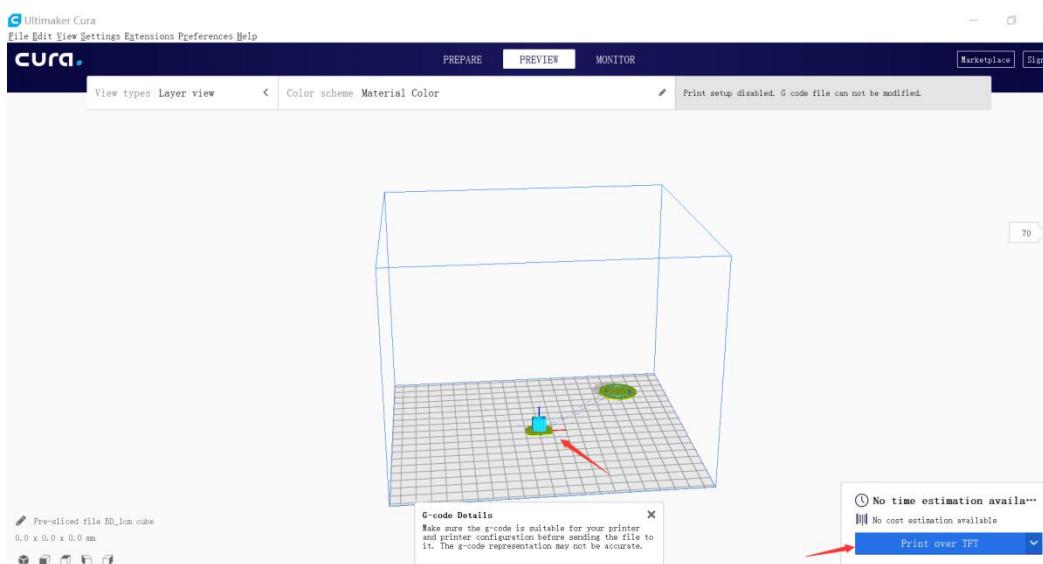
Restart the Cura, and you will see the blue mark before the machine name. This means the printer is connected with the computer through Wifi. Please kindly note that cura is not stable to control the printer through Wifi. We just use Cura to transfer the gcode files to the printer.



We can open the gcode file or stl file in cura.

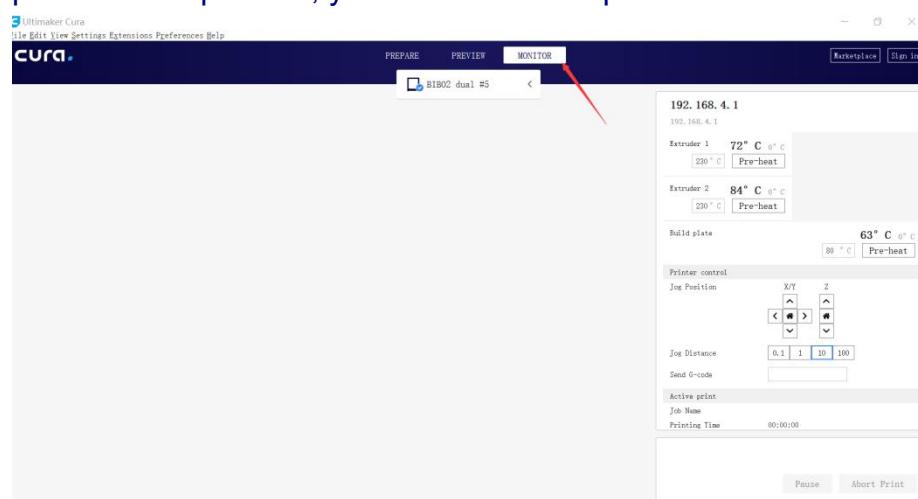


Once there is sliced gcode file in cura, we can click “Print over TFT” to transfer the gcode file to the printer, then the printer starts the printing. It is normal that sometimes the touch screen reboots before printing. Every gcode file transferred to the printer will be named “cura\_file.gcode”. So if you want to save the gcode file, please save it on the computer first. You can use the “Print over TFT” drop-down menu “Save as TFT file” or “Save to File” to save the gcode file. These two menus are the same for BIBO 3d printer.





The MONITOR menu in cura may show the status of the printing, but the information has the time delay. So usually we don't use this menu, and just look at the touch screen directly. When you stop (Abort) the printing, the printer may execute the last gcode command before stopping. So if the printer is heating the bed or extruder when you stop the printing, the heating will be still continued (seems not responding your printing stop command). If you want the printer be responsive, you can restart the printer.



### 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 two difference as below:



A. In mks\_config.txt file, revise the "cfg\_wifi\_mode" to be  
0 ,>cfg\_wifi\_ap\_name:bibo 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.

### \*\*\*Solution in Client (STA) mode\*\*\*

The WLAN access point / router cannot be connected.

Check the following points.

-Unplug the power to your wireless access point / router, wait for 10 seconds, then plug it back in.

-If your WLAN access point / router is using MAC address filtering, please disable the MAC address filtering first.

-Check how many devices are connected with your WLAN router. Usually one normal router can only be connected with 15 devices. If it is fully connected, no new devices can be connected.

-Printer's wifi module is not strong to receive the wifi signal as other normal wifi devices. Please ensure the wifi signal is good near the printer.

-This device does not support a 5GHz SSID / ESSID and you must select a 2.4 GHz SSID / ESSID. Make sure the access point / router is set to 2.4 GHz or 2.4GHz/5 GHz mixed mode.

-If you manually entered the SSID and security information (SSID / authentication method / encryption method / Network Key) the information may be incorrect.

-Reconfirm the SSID and security information and reenter the correct information as necessary.

-In SD card configuration way, we copy mks\_config.txt file directly (not folder) to the sd card and then insert the sd card to the printer which is powered off. Then power on the printer, and usually wait for about 2 or 3 minutes, the printer will connect your room's wifi .

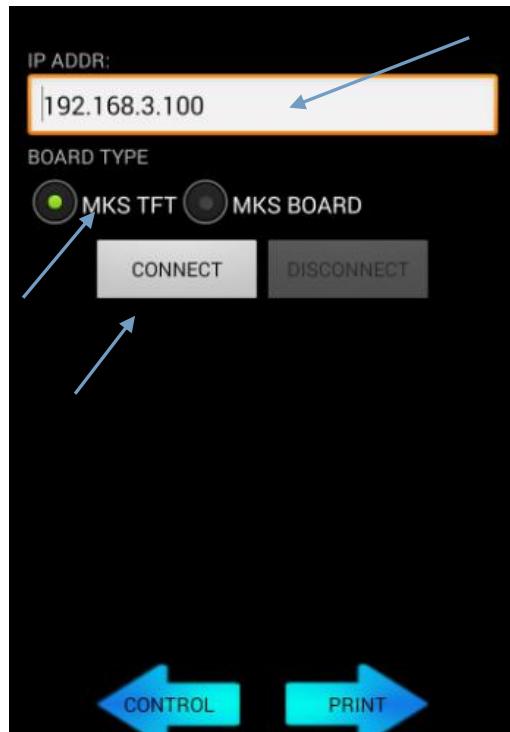
-Change another computer or mobile phone to connect with the printer by wifi. Some computers or mobile phones may have the problem to communicate with the printer by wifi.

-Change another router as some routers are not stable or compatible with the printer's wifi module.

### 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)

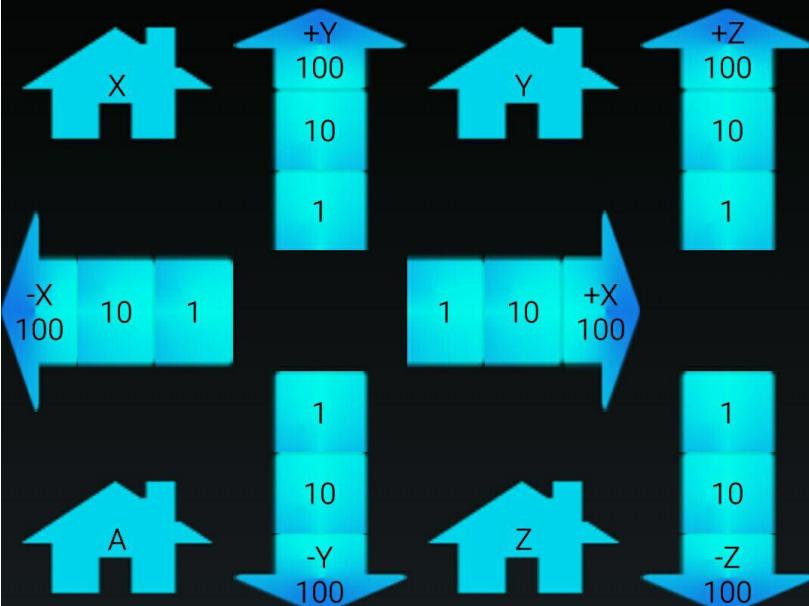
For control the printer, please choose left button "control" and you will see:

08:15

87

MKS\_3DCtr

OFF MOTOR XY: 3000 mm/min Z: 200



Extract: OFF 0 SET 1 2

Hotbed: OFF 0 SET

5 mm 100 mm/min IN OUT

GET TEMP

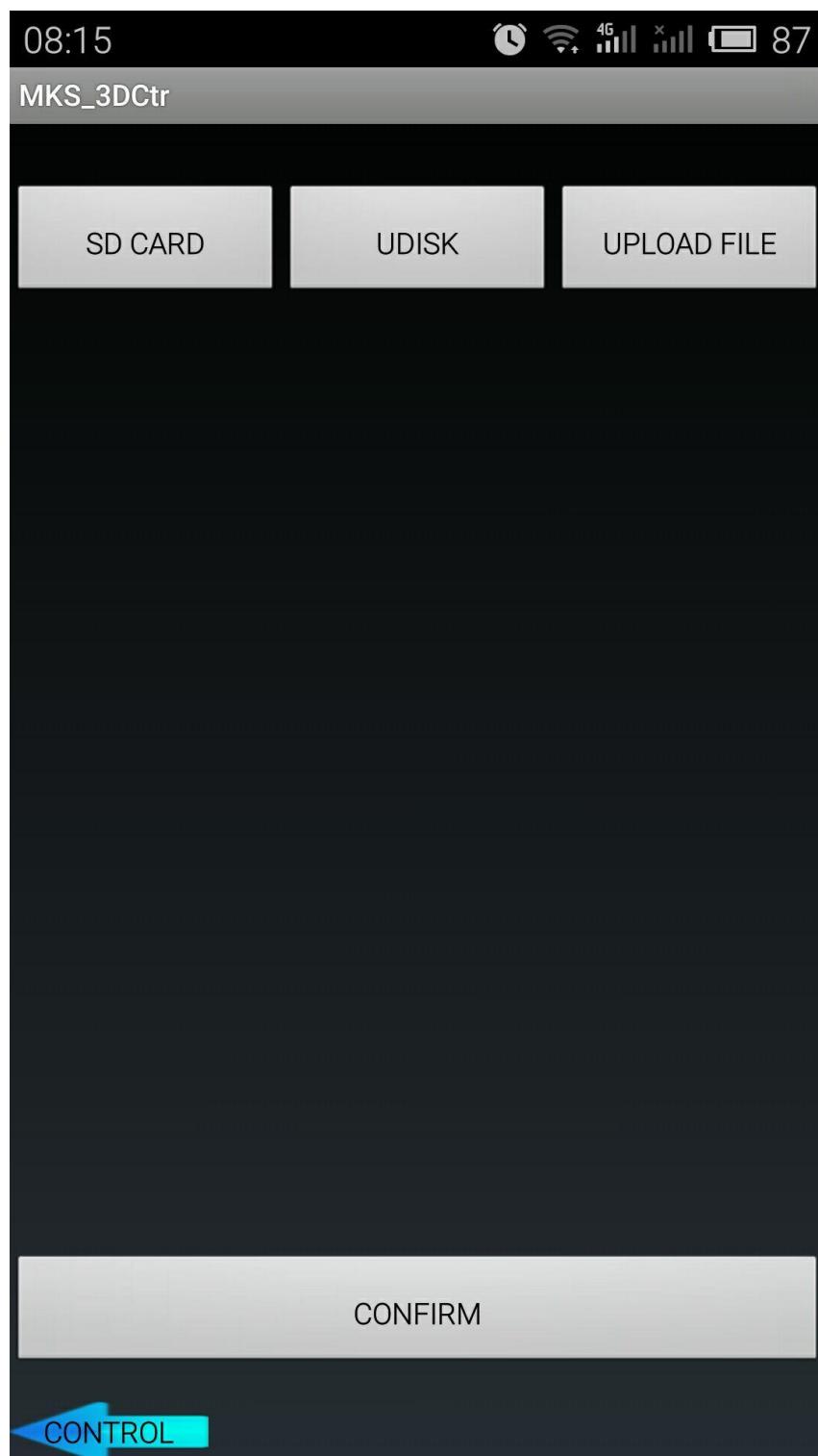
SEND

PRINT



Shaoxing Bibo Automatic Equipment Co., Ltd.

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



2. IOS (Android is ok too)

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

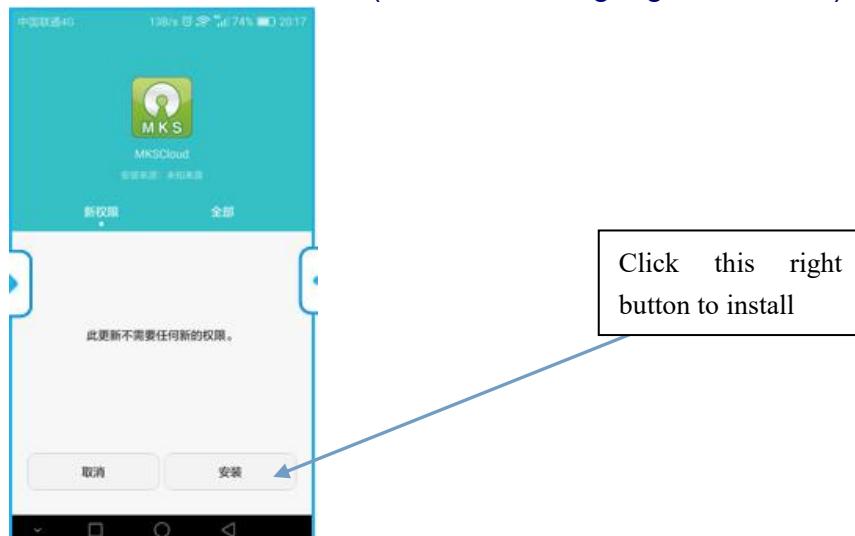
Tel:+86 575 88918283  
Url: [www.ourbibo.com](http://www.ourbibo.com)  
[bibo@ourbibo.com](mailto:bibo@ourbibo.com)

Mobile:+86 133 25759139 78  
E-Mail:



(Android4.0.3/IOS8 or newer)

Now install the mkscloud (The default language is chinese)

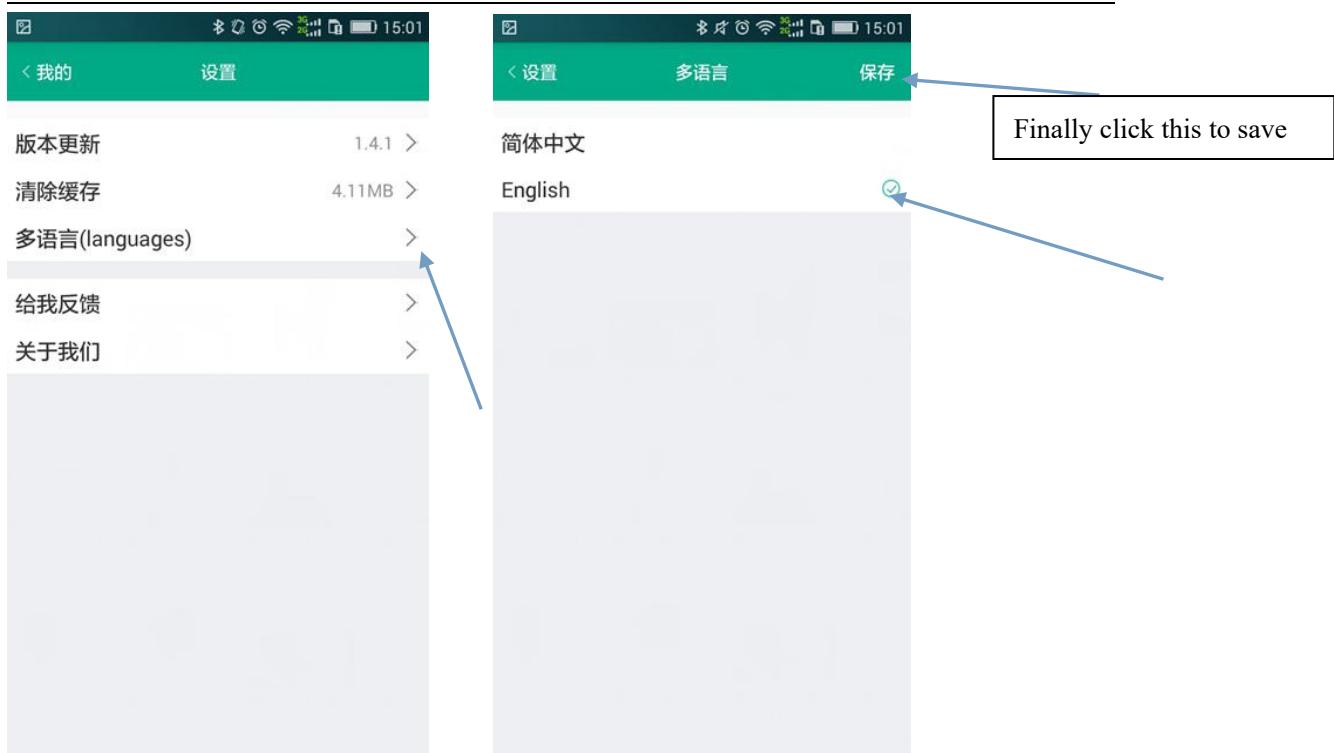


Set the language to be English (click the buttons as below)

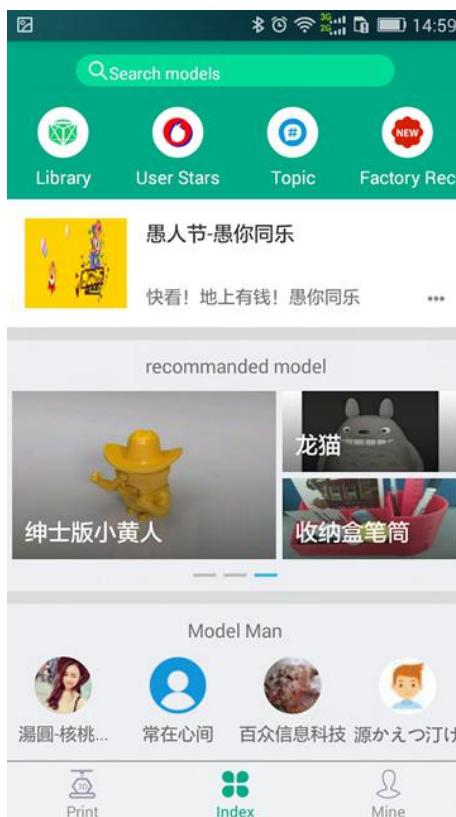




Shaoxing Bibo Automatic Equipment Co., Ltd.



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

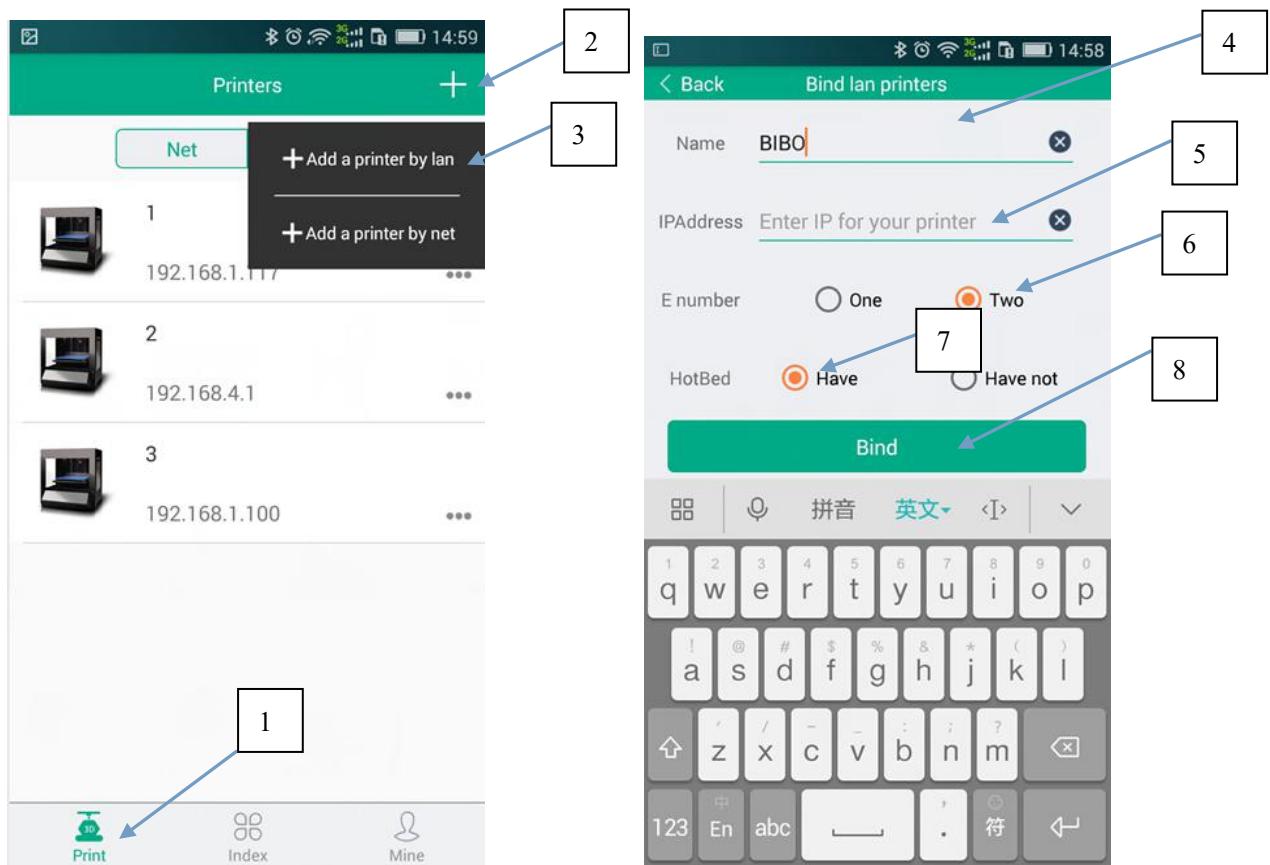


Bind the printer in LAN (Net is not open yet except in China)

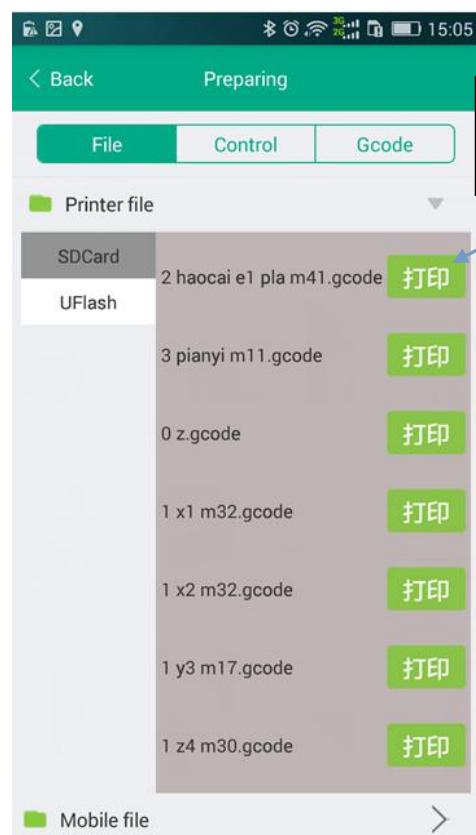
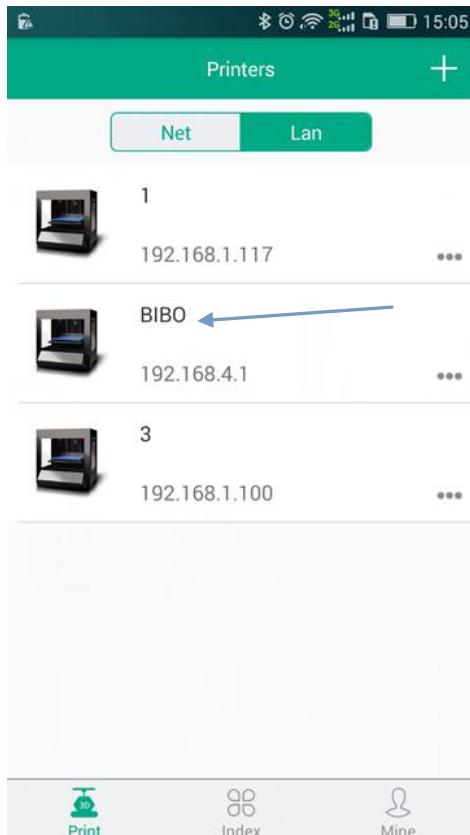
Tel:+86 575 88918283  
Url: [www.ourbibo.com](http://www.ourbibo.com)  
[bibo@ourbibo.com](mailto:bibo@ourbibo.com)

Mobile:+86 133 25759139 80  
E-Mail:

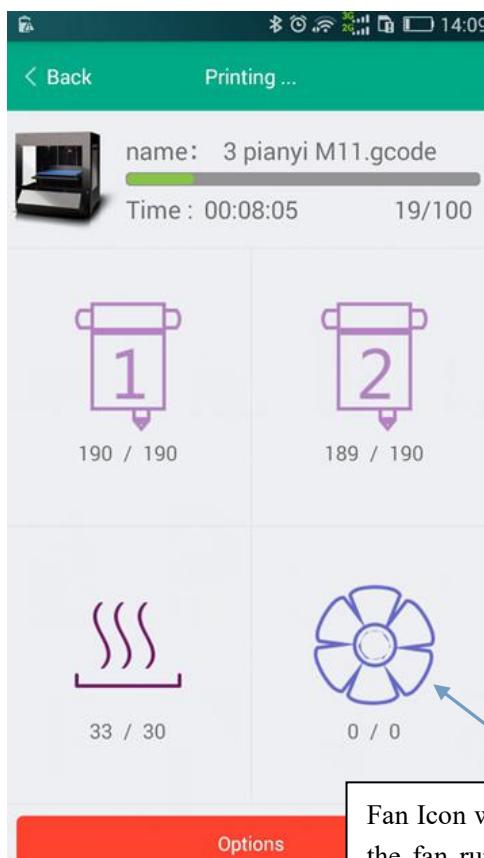
Type the **WIFI IP showed on your touch screen**.(If WIFI is in AP mode, please let your smart phone connect the BIBO 3D printer's wifi first, to ensure that your phone and the printer are in the same LAN)



Click the printer name



Click the green button to start the printing



Fan Icon will not show the fan running status, just an Icon now.



## **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 .

### **Nozzle repair or replacing**

- When the nozzle is blocked, we have to dredge it or replace it. Blocked nozzle may accumulate pressure inside which can cause a burst of molten filament. So our advice is to wear the goggle when repairing or replacing the nozzle.