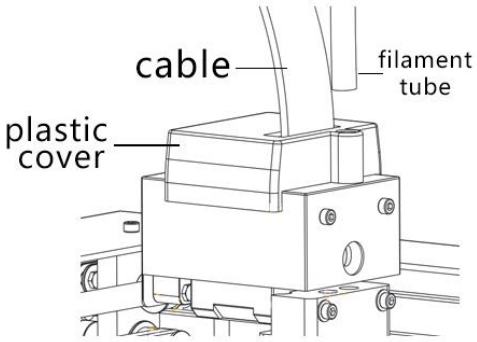
# 3D Printer
# User's Manual

# contents

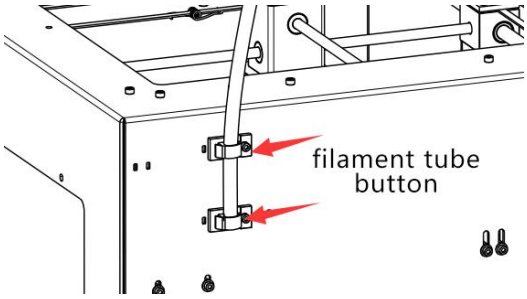
# 1. To assemble the machine

## 1.1 Installing the extruder

- Cover the fan box, insert the cable into the pinboard on th extruder and insert the feeding tube into the fan box.

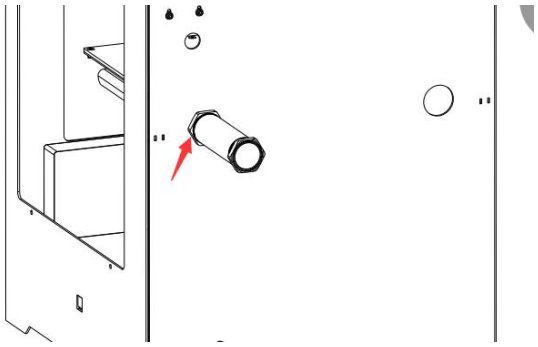



- Install the feeding tube,fix the tube with fixed buttonon the back of machine

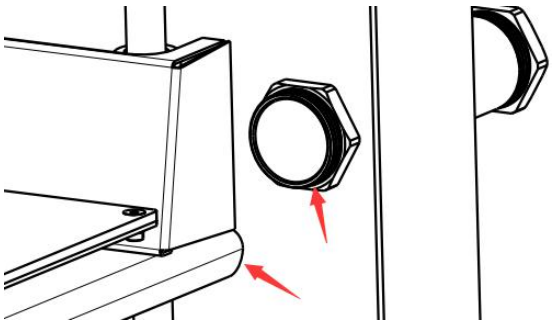



## 1.2 Install the Spool holder

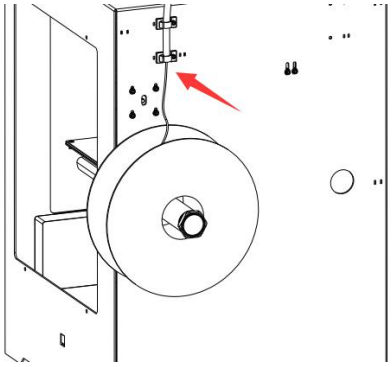
- Fix the spool holder:Tighten the 2 screw capto fix the spool holder with machine



- Do not insert the spool holder into machine too much or the platform will crash the spool holder when peinting



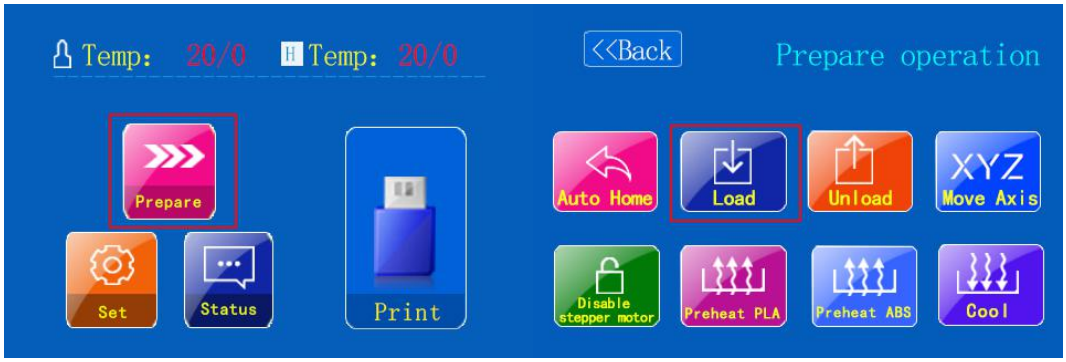
- Hang the filamenton the spool holder and insert the filament into the feeding tube till the filament through the gear of extrude.



## 1.3 Load filament

1.3.1 Plug in the power cable and trun power on ( I is for on ， o is for down)。


1.3.2 Operation on the screen：



3d printer will autohome and the plateform will go down for a while,and at the same time,the nozzle will heat,and when the temperature of nozzle is up to 230℃ ,the filament will come out from the nozzle。


## 1.4 Attentions

1.The temperature of Nozzle is up to 200℃ when printing and after a while,do not touch the nozzle.
2.This User manual is designed to help you start your experience with MakerPi .
It is essential that your read this manual carefully.
3.Wear gloves if you ned to remove the support,or you would get hurt by the support and tools.
4.Clean the linear shaft of acrossplate and driving and oil the driving linear shaft regular,do not drop the oil on the frame and belt.
5.Take pictures or videos when you have problems and provide the pictures,video and S/N(on the back of printer) to our aftersales.

## 1.5 Leveling the build plate

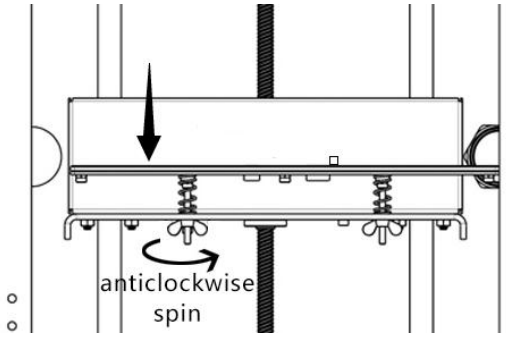
coarse tuning

- Touch Prepare and Autohom.
  the platform and nozzle will move to zero-point

- Touch disable stepper motor
  you can move the cross-plate by your hand.

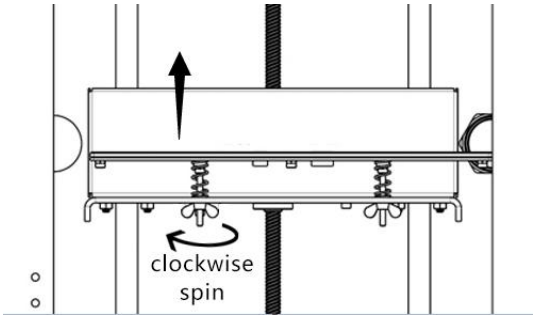
。


- move the nozzle to 4 coners of the build plate to adjust the distance between the nozzle and galss. Rotate the four buildplate screws to adjust the distance between the nozzle and platform




Rotating clockwise means closer to the nozzle.

Rotating anticlockwise means further to the nozzle.

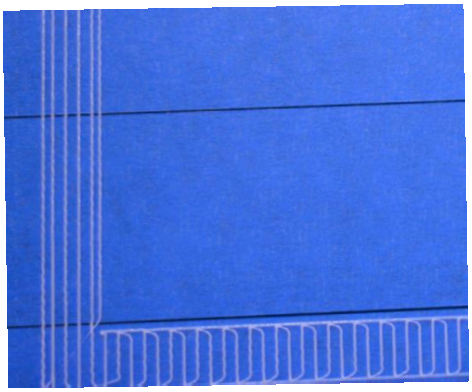



- When corse tuning,the distance between nozzle and galss is the thickness of name card,if the distance is too far, rotate the screw clockwise to make the distance right ，and if the distance is too close,rotate the screw anticlockwise to make the distance right.

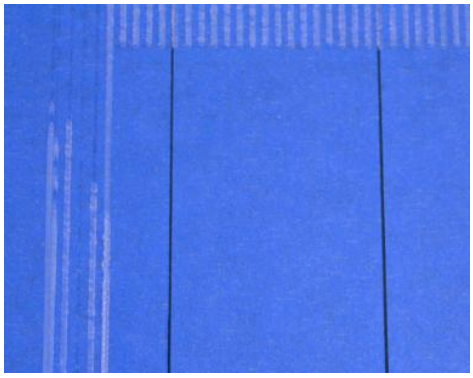
## 1.6 Fine tuning

Insert the U Flashcard card into MakerPi and choose **adjust.gcode** to printe,

If the filament come out like this,it mean the distance of nozzle and platform is a little far,and you need rotate the buildplat screws to tune the distance

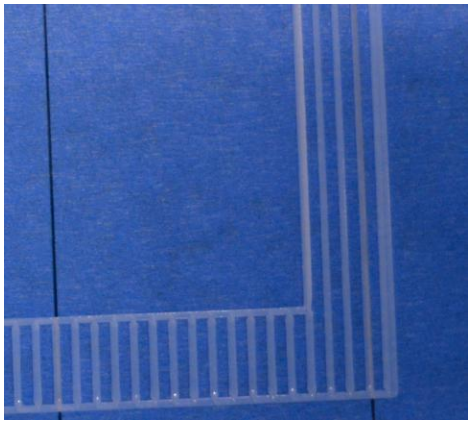


If filament in the platform like this,it means the distance is too close,you need rotate the buildplat screws to tune the diastance.



Filament come out from the nozzle should be like this.

。



Level the platform is very important,Most of the failure are caused by the the platform

# 2 Introduction of touch screen

## 2.1 Prepare

In the menu,there are some basic fuctions to choose.

。


### 2.1.1 Autohome

It means the nozzle move to (0,0,0)

Touch **Prepare---Autohome**
The nozzle will move and the buildplate will rise.

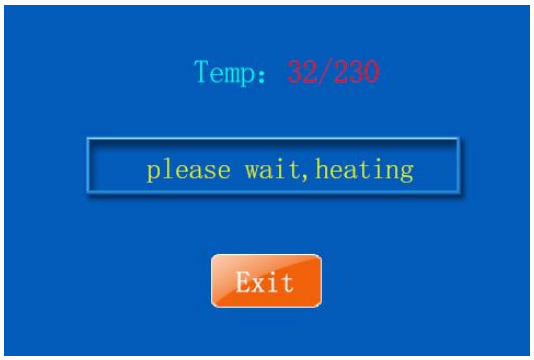

### 2.1.2 Disable stepper motor

it can make the stepper motor can move the crossplate by hand.
Touch Prepare--Diable stepper motor
Then you can move the crossplate by hand

### 2.1.3 Load

Before load,insert the filament into the extruder
Touch Prepare--Diable stepper motor

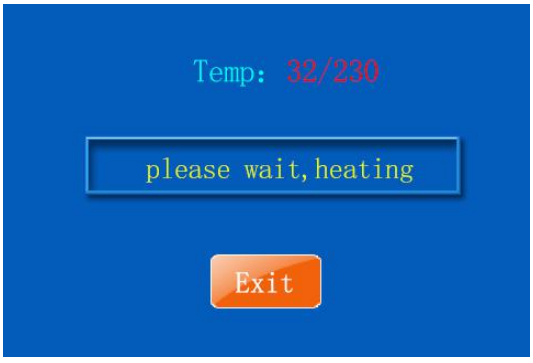

The machine will autohome and build plate will go down for temperature is up to 230℃，it will extrude the filament




### 2.1.4 Unload

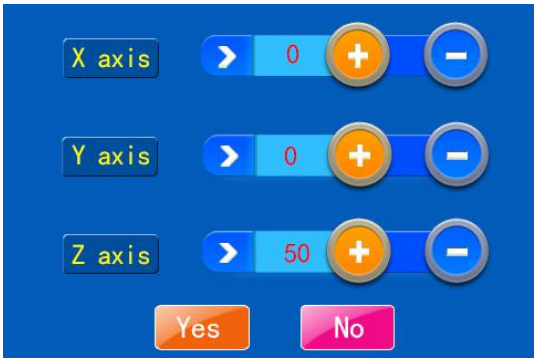
Unload the filament from extruder.
Touch Prepare--Unload


The nozzel will heat and when the temperatue is up to 230 ℃ ,the filament will unload



### 2.1.5 Move X,Y,Z Axis


Sometimes,we need to move the axis
Touch Prepare--Move Axis




then you can set the data to move axis, you can press the + and - to change the data.

Short press for 1 everytime,

Long press for 10 every time.

### 2.1.6 Preheat


- Before the printing, you can preheat the 3d printer,and you can start priting when you slice the 3d file.it can save your time.you can choose to preheat PLA or ABS.
- Touch Prepare--Move Axis

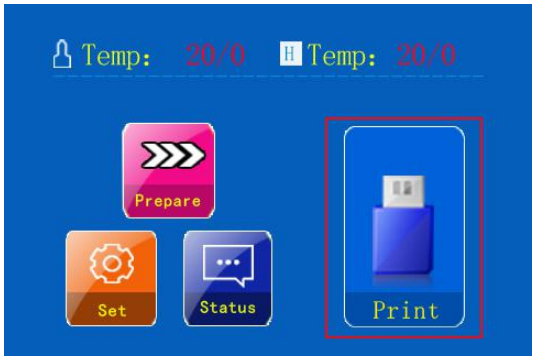

When you choose preheat PLA,the nozzle and the hotbed will be heated at the same time.The nozzle will be up to 180 and the hotbed will be up to 70.

### 2.1.7 Cool

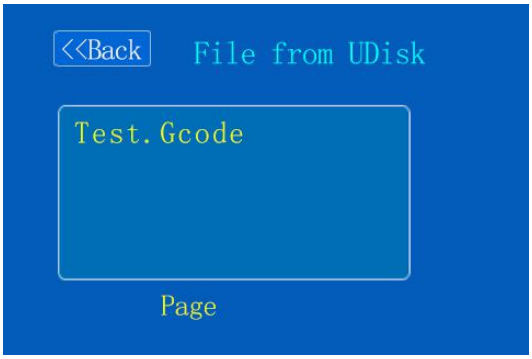
- After you preheat the 3d printer,and you dont printe in the end,you can use this function to cool the printer.
  Touch Prepare--Move Axis


## 2.2 Printe

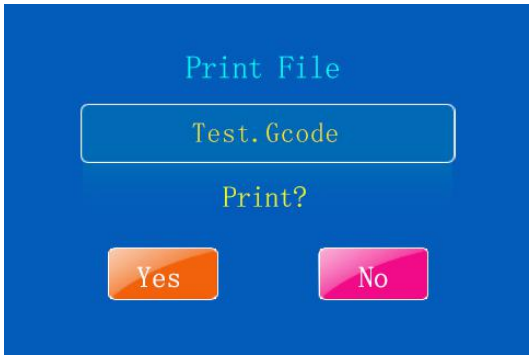
- Before printing,you need level the building palteformand load the filament.
- Touch printe




Choose the file you printe.



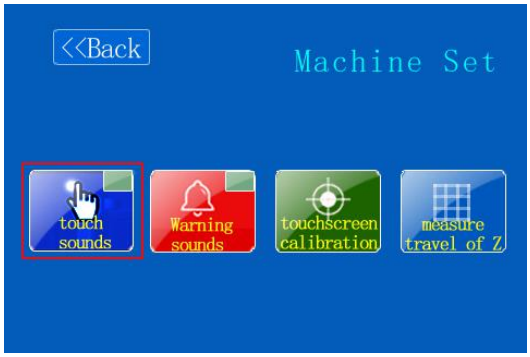

Touch yes to comfirm the file you choose.



## 2.3 Set

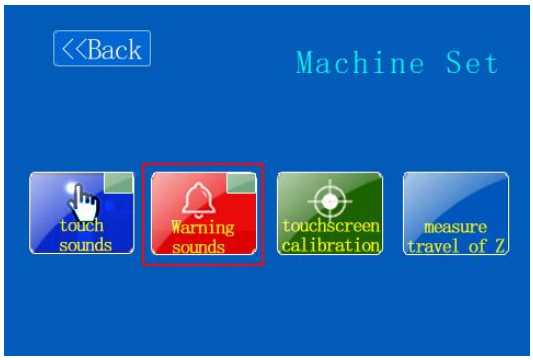
In this section,you can set the touch sounds,warning sound,calibtare the touch screen and measure the Z travel lengt.


### 2.3.1 Touch sounds



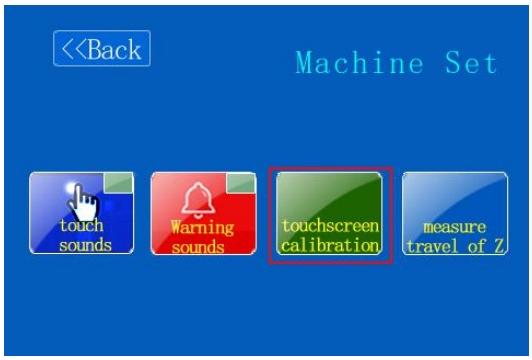

- Touch touch sounds,you can cancel the sounds and you can touch again to resume the sound.

### 2.3.2 Warning Sounds




- Touchwarning sounds,you can cancel the it and you can touch again to resumethe warning sounds.we suggest to keep the warning sounds.

### 2.3.3 Touchscreen calibration



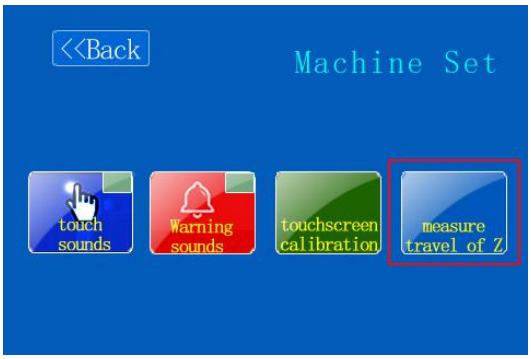

- When the screen seems imprecise or unresponsiv,you need calibrate the touch screen.

  Touch touchscreen calibration

- Now begin calibrationg,touch the red center,when it finish,it will inofrm you calibration sucess or fail.if failed,you need calibrate it again.


### 2.3.4 Measure Z travel length

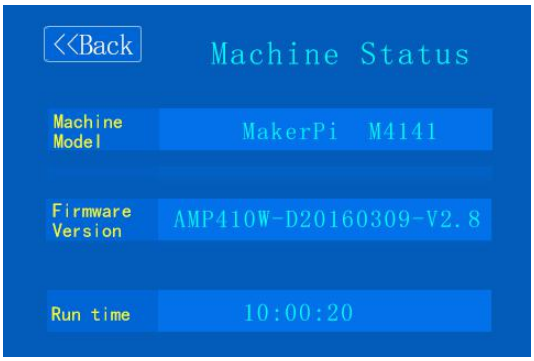
- The Z length maybe change a liitle when using,we can measure it.




## 2.4 Check the statues

- Touch the sattues on the menu,you can check the infomation of the printer


You can see 3d printer infomation,firmware version,and run time.



## 2.5 Other functions

### 2.5.1 Basic function

- **Pause：pause printing**


- **Stop：Stop printing.**

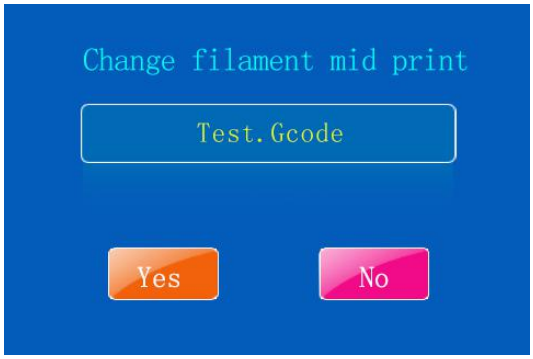
If printing fails or other problems that you want to stop,you can touch stop.


- **Change filament：**

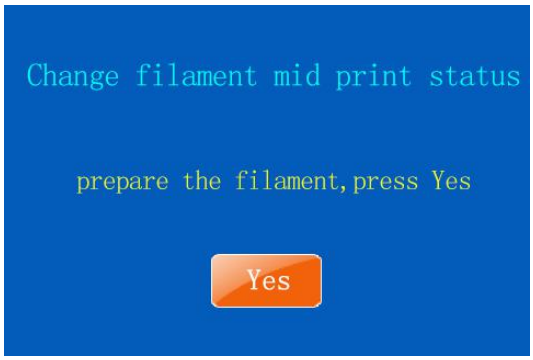
When the filament is not enough or you want to change the filament when printing,you can touch Change filament when printing。


### 2.5.2 Change Filament mid printe

- When you printe big models,your filement maybe not enough,so you need to add new filament to continue printing.

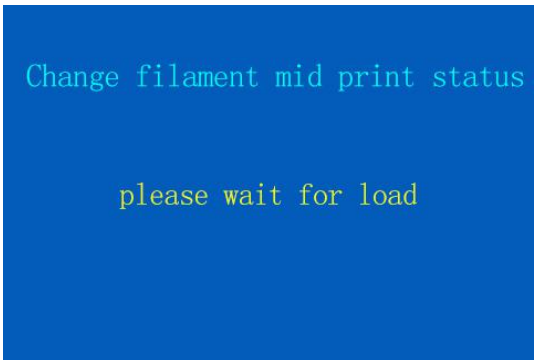


- Touch Yes,it will unload the filament.




- When unload finish,we should load new filament,after we insert the new filament into the extruder,touch yes to load.

When load finish,it will continue printing.



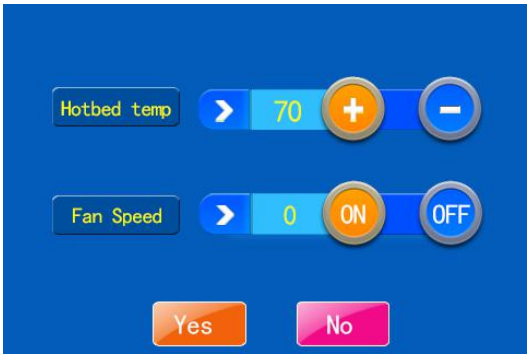

### 2.5.3 tune temperature,printe speed and fan speed

- After the slicing,you find some parameters ,such as tempetature,printe speed,fan speed, are not so suitable,you dont need to slicing again,you can change them on the screen.


Touch 

you can change the nozzle temperature and printe speed.

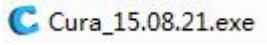

Touch you can change the nozzle temperature and printe speed



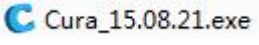
# 3 Software Installation and Operate

## 3.1 Installation of Slicing Software

Slicing software program CURA is in the WIN (file name) of SD card software (Please choose mac - file name for Apple computer)




(Software version will be updated continuously, and we will provide the best version before machine delivery)

Double click and run CURA 


software installation program, and operate by software reminding. Please select the relative components if OBJ and AMF file will be opened during operating CURA, please select the relative components, which is showed in the following image.

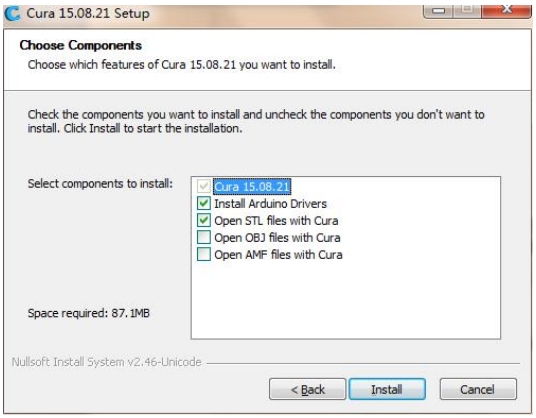
- Select by printing requirement,
  And click install for further installation

**1. Install Arduino Drivers**

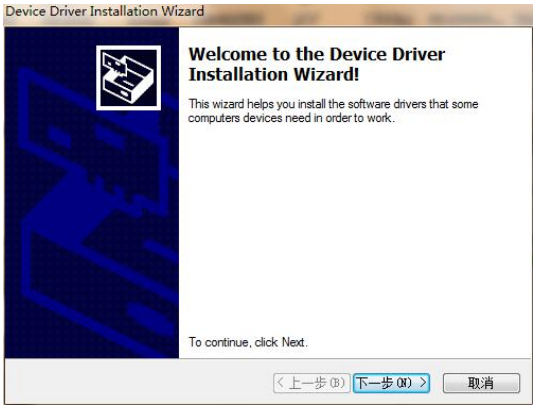
**2. Open STL files with Cura**

**3. Open OBJ files with Cura**

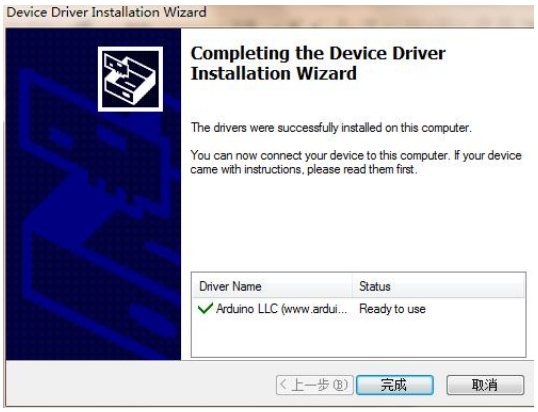
**4. Open AMF files with Cura**



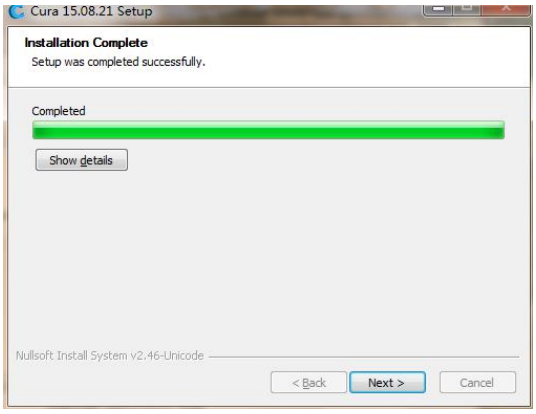

- It needs to install software driver if the menu (in the right image) pops up. Please click 'next step' and select trust if anti-virus software warns.



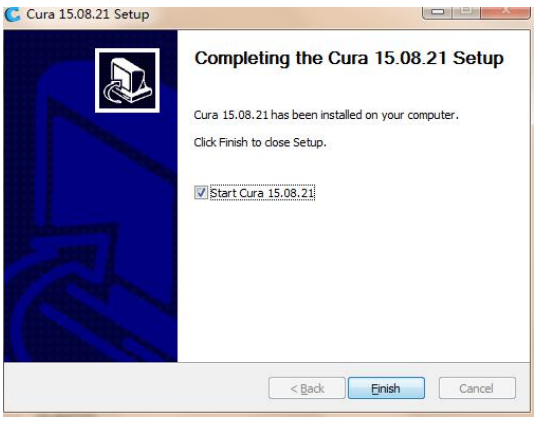
- Click finish button and finish driver installation.



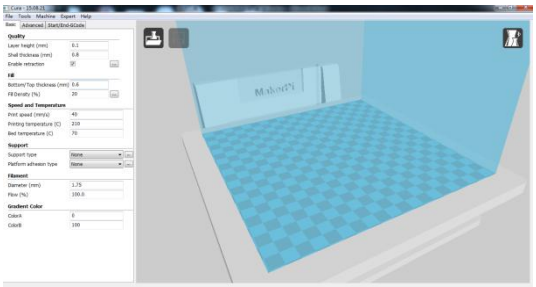

- Slicing software installation continues. Click 'Next' after green progress bar is full.




- In the end, click 'Finish' and finish installation of slicing software.




- After installation, the software will automatically open and menu (in the right image) will appear.

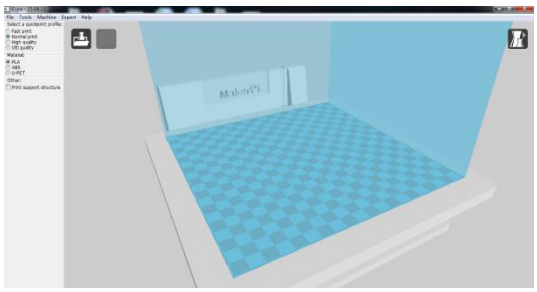


## 3.2 Operation of Slicing Software

### 3.2.1 Quickprint

Click 'Expert' - shift to 'quickprint'

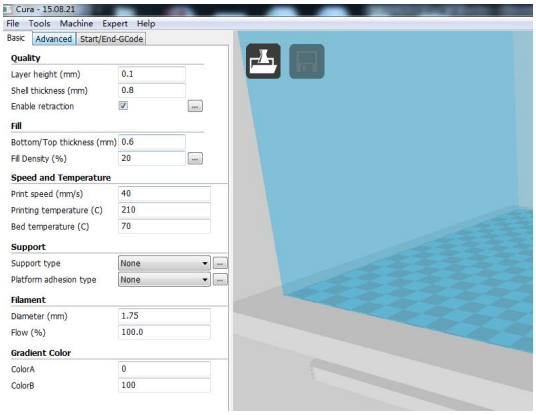
- Quickprint: It do not need user to set the parameters manually, and all parameters will be set as default value per requirement. The menu (in the right image) includes quickprint profile, material and support or no support



### 3.2.2 Full Settings Print

Click 'Expert' - shift to 'full setting' and then choose 'Y (Yes)'.

All parameters will appear in full setting mode. Put the mouse pointer on any parameter, and detailed introduction for this parameter will appear. Set the parameter per displayed information as reference.




### 3.2.3 Basic setting

1. Layer height: the thickness of every layer of the filament, support 0.05-0.3, value between 0.1-0.2 is recommended.
Effect: The thinner of the layer is, the finer the surface will be, the time of print will be longer too.

2. Shell thickness: the thickness of the outside wall of the model, 0.4 per layer of filament，value between 0.8-2.0 is recommended.
Effect:The thicker the wall is, the more intense it will be. The time of print will be longer too.

3. Enable retraction: retract the filament when print.
Effect: Without retraction, it will over pull the filament which will impact the overall printing.

4. Bottom/top thickness :the thickness of the bottom and the top.
Effect: If the printed model has damaged holes on the top,please adjust the thickness figure accordingly.

5. Fill rate: 0 for hollow, 100 for solid.
Effect: It can save print time by decreasing the filling, but will impact the print strength. Hollow might lead to failure of completing the print model because the wall is too thin. Therefore appropriate filling is necessary.

6. Print speed: recommend 40-60mm/s.
Effect: Slow down the speed appropriately, allow reasonable cooling time while printing, the model will be printed out better.

7. Printing temperature: the nozzle temperature while printing, ABS recommend 210-230, PLA recommend 190-220.
Effect: Too low temperature makes extrusion stopping, and it will block the nozzle to export filament.

8. Bed temperature: ABS recommend 90-100℃, PLA recommend 60-70℃.
Effect: If the temperature is too low and the filament is not adhesive enough, edge warp might happen.

9. Support type: because there will be overhang during printing, the filament might drop for gravity reason. Therefore, a support is needed. but not for all.
None: no support is needed.
Touching buildplate: support from outside, only add support for the overhang part outside of the model.
Everywhere: add to every part of the model as long as it is overhang, including inside of the model.
Effect: If the model is overhang, it needs support. If not, the filament will drop during printing.

10. Platform adhesion type: add a base and it can enhance the stickness of the print model. None: no platform is needed. Brim: extra thick base, and attached cohesive material around raft: net shape base.
Effect: Adding a base can enhance the thickness of the platform, Raft type base save more material.

11. Diameter: diameter of the filament.

12. Flow: the flow speed of filament while printing.
Effect: Diameter is used in conjunction with the rate of flow. The bigger the diameter is, the slower the filament come out. The bigger rate of flow is, the faster the filament come out.

### 3.2.4 Advanced setting

1. Nozzle Size: 0.4.currently this figure is fixed.

2. Retraction speed: Speed at which the filament is retracted.
Effect: Theoretically the faster the better, but it might possibly lead to unable to feed filament.

3. Retraction distance: the length of filament. These two parameter will only valid when choose retraction allow in the general setting.
Effect: If the length of the retractioned filament is too short, it will possibly lead to over-pulled, if it is too long, the filament might be able to be loaded.

4. Initial layer thickness: thickness of the first layer.
Effect: The thicker the first layer is, the more adhesive the model will be.

5. Cut off object bottom: some models have an uneven bottom or lack off contact surface, the bottom can be cut off partially.
Effect: For those models that doesn't matter much with the bottom or need to be printed separately, it can be cut off to print, the print outcome will turn out better.

6. Dual extrusion switch amount: double print matters when setting the double print, repeat the extrusion variable.
Effect: By setting a certain amount of repeat extrusion variable, two colors will be adhesive to each other better.

7. Travel speed:the speed of the machine moving.
Effect: The faster it moves, the shorter it costs for printing.

8.Bottom layer speed: the print speed of bottom layer, slow speed can make the print more adhesive.
Effect: Slow down the print speed of bottom layer appropriately enable the bottom adhesive better, in this way, the print effect will come out better.

9. Infill speed: the speed of print filling.
Effect: Speed up the filling enable print faster.

10. Outer shellspeed:the speed of print wall, slow print can make the wall print better.
Effect: Slow down the print speed of wall, the surface will be smoother.

11. Inner shellspeed: the speed of the print inwall. Higher speed can shorten the print time.
Effect: Speed up the inwall print can shorten the print speed.

12. Minimal layer time:the minimum print time of each layer, if print too fast,the machine will slow down according to the minimum print time of that layer to ensure enough cooling time.
Effect: Control the minimum print time of each layer to ensure enough cooling time.

13.Enable cooling fan: open the nozzle cooling fan to speed up the cooling.
Effect: It is used to speed up the cooling while printing, the print effect will come out better. Careful use for ABS, it is easy to rip off.

### 3.2.5 CURA Print Menu

Load model  Save model 

Save to SD card  USB network print

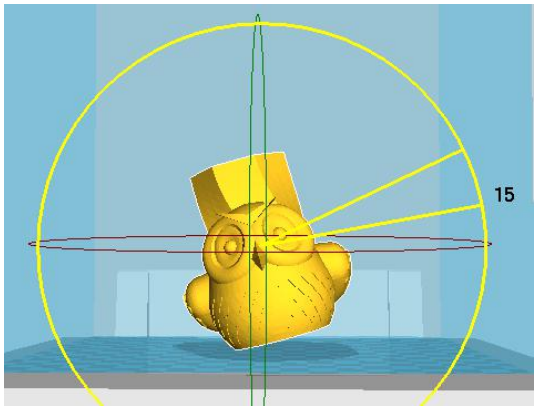
**Rotate:**

Rotate the model 

Automatically lay flat model 

Reset model

- Click rotate icon, and 3 circles in different color whose circle center is the same as model center. The model can be rotated by any angle around the center axis of vertical circle surface.
  Use mouse left button to choose the circle, and hold the click, move the mouse to rotate the model in any direction.




**Scale:**

Scale model 

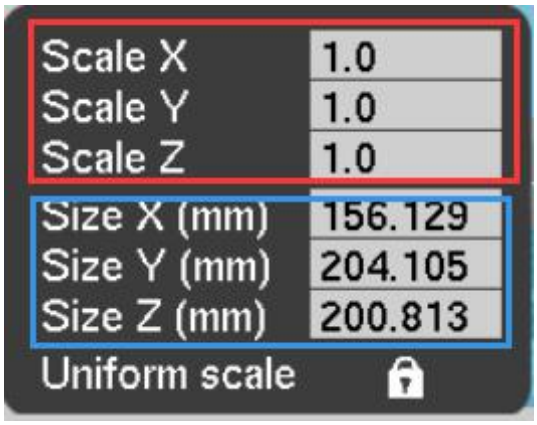
Reset model 

Maximize model

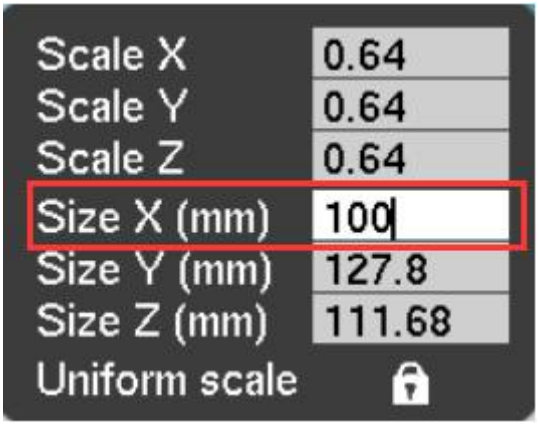
- **Scale the model**

Scale stands for dimension scale of the model, and model size can be scaled up/down by changing dimension scale.

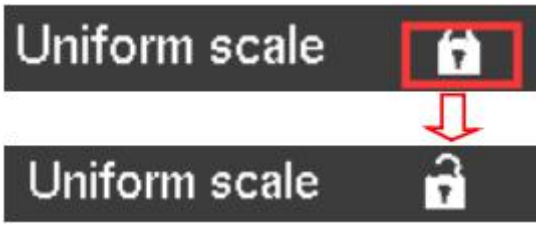
Size stands for dimension size of the model, and model size can be changed by changing value directly. Size change in any direction will cause size change of other dimensions at the certain ratio.



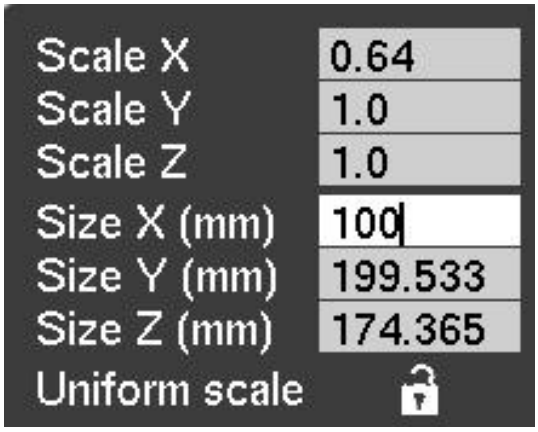

- Shift X of the model from 156.129 into 100 and scale displays that dimension is 0.64 times as long as the original one, and Y size and Z size will automatically shift to the corresponding value.




- If you only want to scale up/down a certain single dimension, Please click icon of uniform scale, and it allows to scale up/down a single direction.




- Icon of Uniform shifts to unlocked, X dimension change cause no change of Y scale and Z scale as well as Y size and Z size.



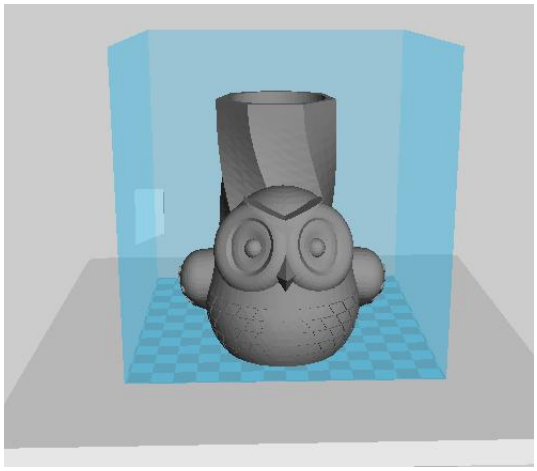
**Auto mirror:**  mirror

Mirror in X direction 

Mirror in Y direction 

Mirror in Z direction

### 3.2.6 View Model



- **View if model size is within printing range.**

Open model and usually it appears to be too large, as the right image displays, model size or model location surpasses printing range of 3D printer, and the model turns grey.

Change model size automatically by requirement or shift model position, until that the whole model turns yellow.

- **View Model Overhang**

Overhang usually exists in print model (Please check chapter six), and the following ways can be used to view overhang.

: View mode, it is in top right corner of Cura basic menu.

: Normal status view

: View overhang. Overhang that requires support appears in red.

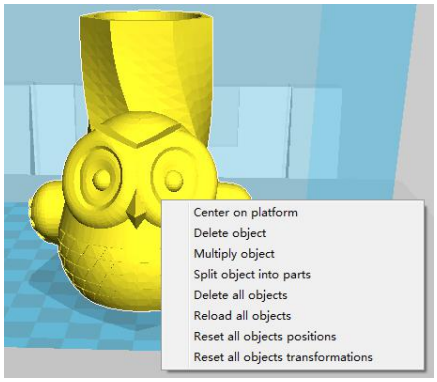
: Transparent mode

: X-Ray mode

: View layer by layer. It simulates printing process.

- Basic operation of model view

Select model, click right mouse button, and you can make the following operations to the model:




**Center on platform**：move the model to the center of the printing platform.

**Delete object**: Delete unnecessary model

**Multiply object**: Multiply model quantity.

**View model from various angles:** Click and hold right mouse button, and move mouse

**Move model**: Click and hold left mouse button and move mouse

**Zoom in / out to view model**: Scroll the wheel of mouse

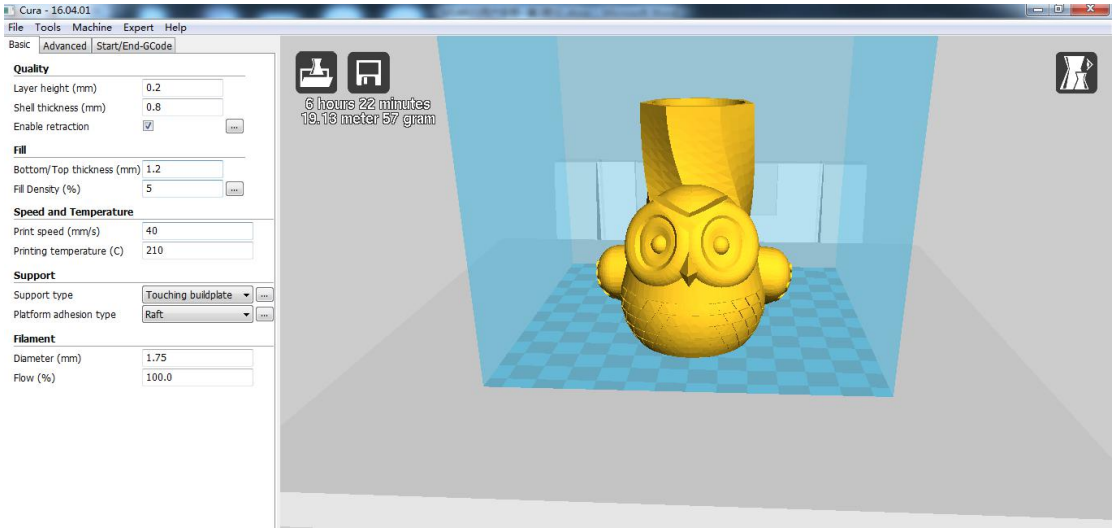
# 4 Printing Exercise

Last chapter we already know function of slicing software and now make a practical operation!

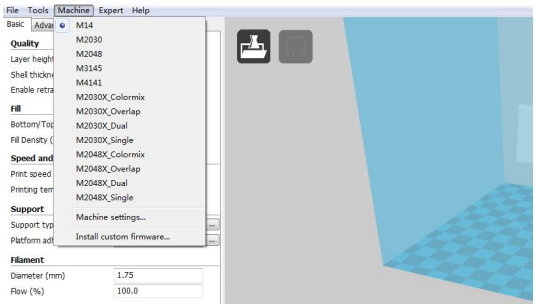
To finish a printing, please follow the following process 'STL file' → 'Gcode file' → '3D printer'.

The practical operation is in the following:

- Open Cura, click icon  or select 'Load model file' of 'File (column)', choose the STL file which you want to print, click and open it, which is displayed in the following image.

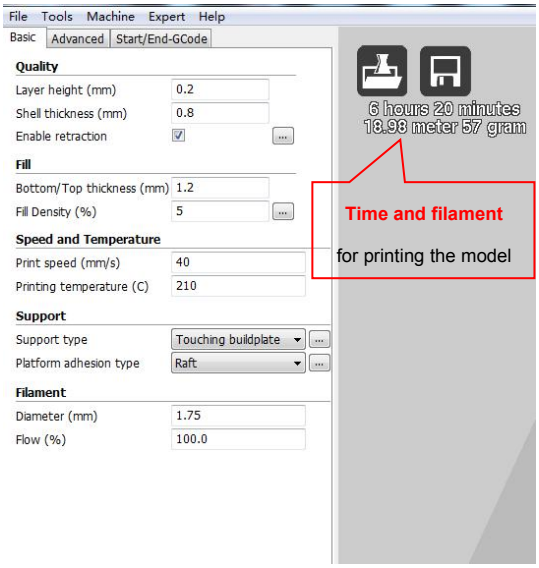



- Choose M14 from column 'Machine'



- Set required parameters for printing, and adjust location and angle of the model manually. once slicing is finished, Cura will show running time and filament weight for this printing.

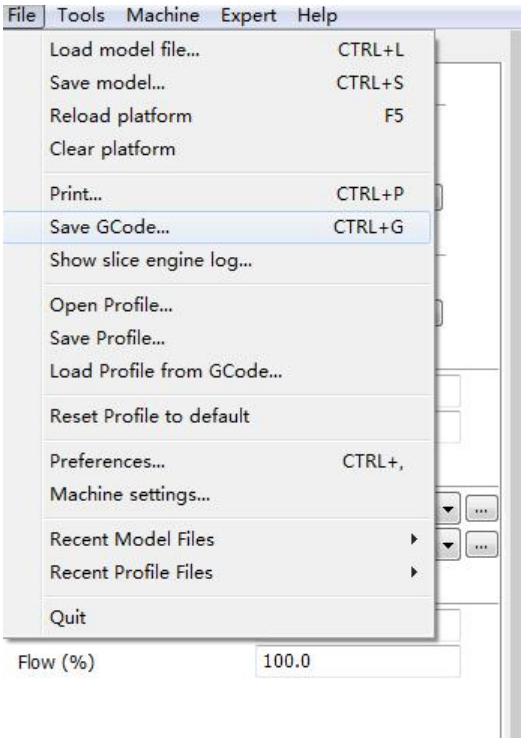
**Notice:** check if filament is enough and if buildplate is leveled by information provided by Cura.



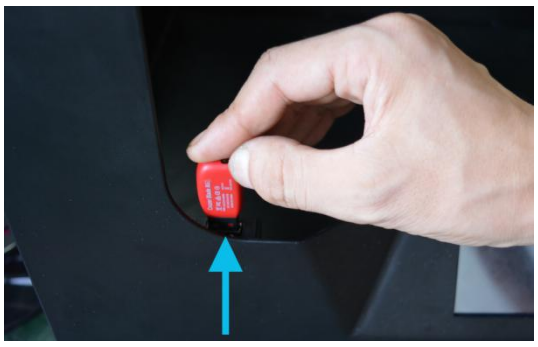

- Click icon 

and it can directly save gcode into U flash disk.

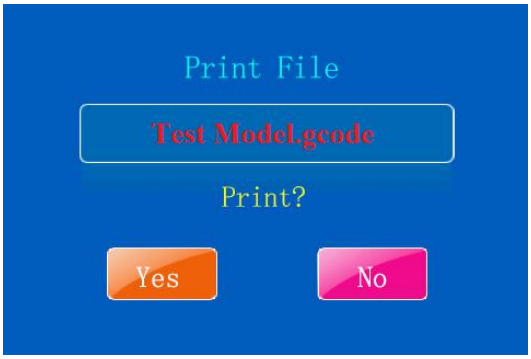
  If you want to save gcode in other places, or if you want to rename gcode, please select 'save Gcode' in 'file', and save it in a certain file.




- Insert USB flash disk into USB port.



- Select 'print'. Find the Gcode which is saved previously and select 'print'.



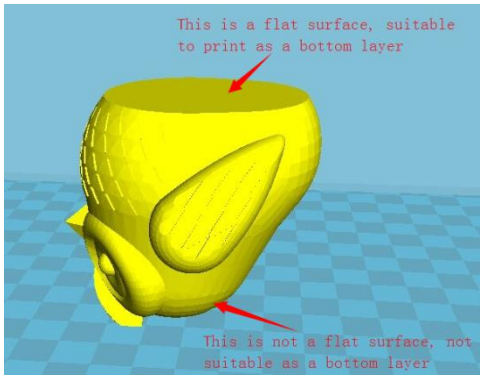

- Wait till model starts to be printed, and then the model will be finished by printing. **Notice:** The model will adhere to buildplate after finishing printing. Please use a shovel to remove model bottom slowly, and, please heat the buildplate if it adheres too strongly and then remove it.

# 5 Printing Technique

FDM format 3D printer has its own print features,mastering those features can print out perfect graphic. Summarized as follows:

## 5.1 Flat Bottom

When start to print, choose the flat bottom side,because it starts to print from the bottom and then all the layers up, so the bottom decide the whole print quality. Therefore, it is essential to print the bottom layer, choosing a good flat bottom as the bottom layer is a good start.

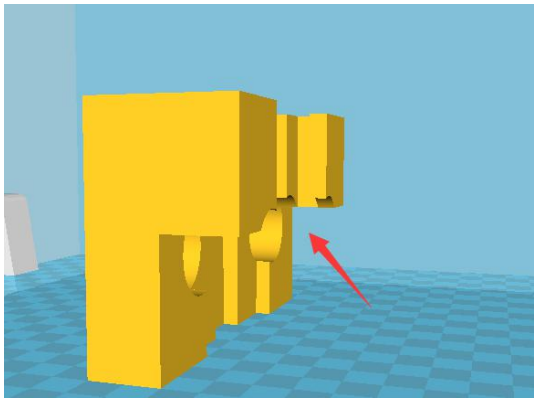



## 5.2 Avoid Overhang

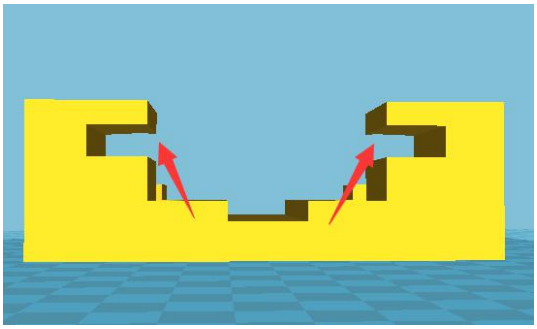
Because the filament that come out from the nozzle is fluid during printing, it will drop down for gravity reason, if there is any overhang in the printed image, the filament will drop down, therefore, we need to use software to form a support when print, and remove it after print is done. It is same as building a house, because the concrete is fluid which will drop down, so we need stake it off, and remove the wooden stakes after the concrete is consolidated.
Same as construction, after removing the wooden stakes, it will leave the stake marks. So after print finish, the support mark will be there. So if you want to improve the print quality, you need to avoid overhang as much as possible to improve, the following two aspects must be done:

### 5.2.1 Avoid overhang by changing printing location

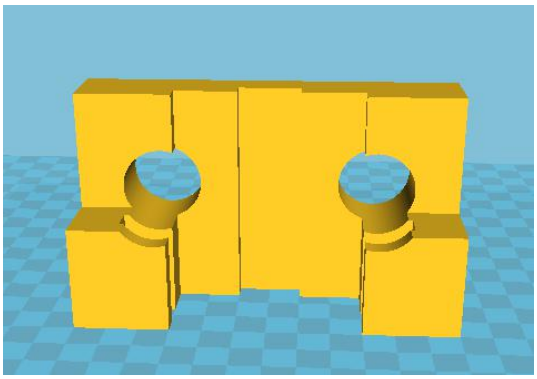
- Overhang appears which red arrow points to in the right image.



- In this direction, 2 overhangs appear (Please see the right image)



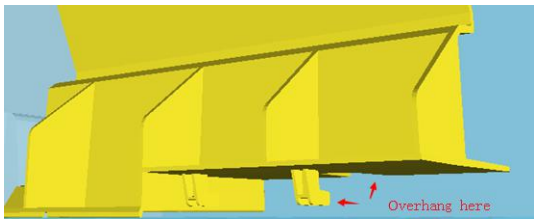
- At last, it is decided to print in this direction.



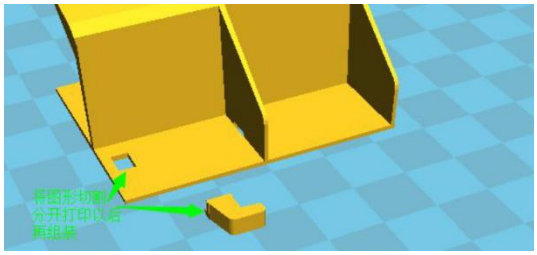
### 5.2.2 Avoid overhang in design

- Overhang appears in this model and overhang cannot be avoided by changing location. So divide overhanging incline and pillar from main part, print them separately and stick them together with glue in the end.

- Overhang appears in the bottom of this model, and it belongs to assembling part.




- Since it belongs to assembling part, please separate it and print it directly, assemble it and it is done.



## 5.3 Tolerance for Assembling Models

If printing model like screw, nut, gear and these assembling models needs to be assembled, please increase the tolerance a little higher, due to expand with heat and contract with cold of plastic in printing and inflated edge caused in printing model bottom. Usually it is set as 0.4mm, and, to be specific, it is set by actual model.

## 5.4 Big Size Model Print

If the print volume is relatively big, PLA filament are needed, the feature of ABS filament decide that it is not suitable for big volume graphic, it will easily split and out of shape. So PLA is recommended for printing big volume graphic.

If the big volume graphic is still easy to get edge warping during printing, you can use 502 glue to stick the bottom, but do not put 502 to the nozzle.