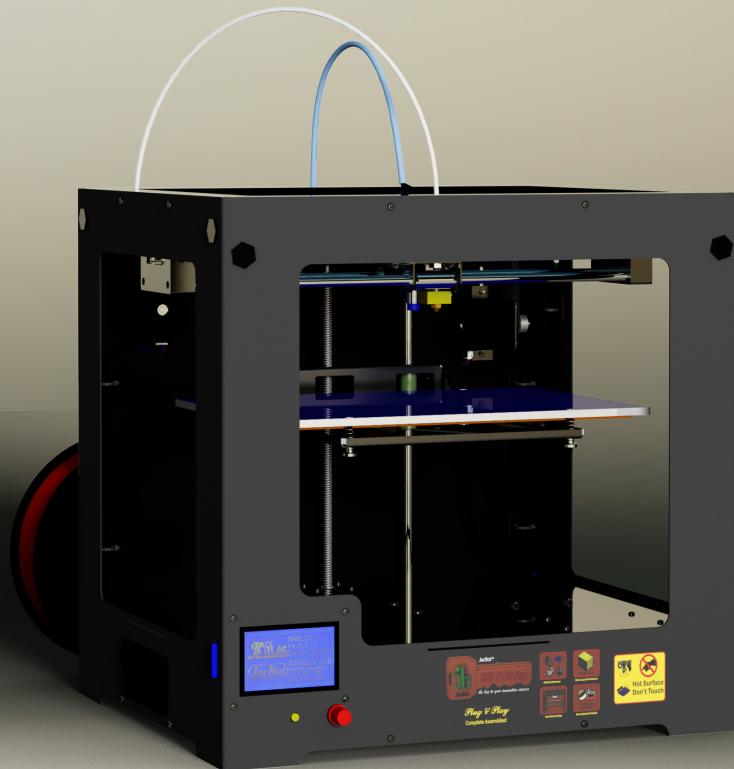




JacBot™ 3D Printer



USER Manual
BP1.7D

Contents

WELCOME

Features & Specifications -----	6
1. Features -----	6
2. Specifications -----	7
How does JACBOT work -----	8

UNPACK

Unpack the Machine -----	10
What in the Package -----	11
1. Machine -----	11
2. Accessories-----	11
3. Software and Sample files -----	12
4. Mount Spool Holder -----	13
Machine Preview -----	14

INSTALL SOFTWARE TO YOUR COMPUTER

Connect JacBot with Computer -----	18
1. Connection Via USB Cable-----	18
2. Connection on the Repetier Host -----	18
Install software to your computer-----	20
1. Install software from the SD card -----	20
2. Installation -----	20
3. Update Software -----	22
Printer Setting -----	23
1. Connection -----	23
2. Printer-----	24
3. Extruder -----	26
4. Printer Shape -----	26
5. Save the settings -----	26
Import JacBot Configuration files -----	27
1. Select a Slice Tool -----	27

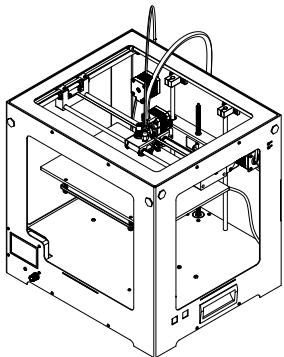
2. Import CuraEngine Configuration -----	27
3. Import Slic3r Configuration -----	28
4. Delete the Default Configuration -----	28

MAKE YOUR PRINT FILE

Work Process -----	30
Place Object-----	30
1. How to get a 3D model file? -----	30
2. Load file and Familiar with the useful tools -----	31
Slice Object-----	35
1. CuraEngine-----	35
2. Slic3r-----	37
Preview -----	39
1. Printing Statistics -----	39
2. Visualization -----	39
Print -----	41
1. Load / Change Filament-----	41
2. The First Print -----	42
3. SD Card Print -----	45

APPENDIX

G-Code -----	48
1. What is G-Code -----	48
2. Formate of G-Code -----	48
3. How the G-Codes work in the JacBot printer-----	49
4. Start G-Code -----	49
5. End G-Code -----	53
Example: -----	53
6. Table of Common G-Codes -----	54
Configuration of Slicer -----	57
1. Select a slice tool -----	57
2. Configure the slice tool-----	57
3. Make New Configuration -----	70
4. Import & Export the Configuration files-----	70



JacBot™

3D Printer

Welcome

Features & Specifications

1. Features

- All metal structure

Strong and stable structure makes printing precise and fast

- Larger printing volume

Up to 300x200x200mm

- Auto bed leveling

Make printing easier

- Powerful and efficient heater

Nozzle temperature can quickly go up to 260°C, bed temperature can speedy go up to 120°C; which ensure to print more materials such as PLA, ABS, Nylon, Rubber, wood etc.

- Plug & Play

Total assembled machine

WARNING: The JacBot 3D Printer generates high temperatures and includes moving parts that can cause injury. Never reach inside the Jacbot while it is in operation. Ensure that the JacBot has been cool down for 10 minutes before reaching inside.

CAUTION: If reach inside the JacBot for service, ensure that the power supply is turned off and the cord is disconnected.

2. Specifications

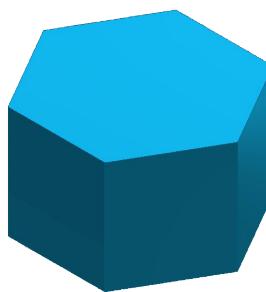
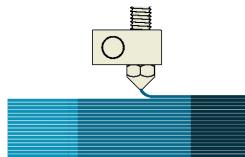
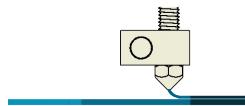
Printing		
Print technology	Fused deposition modeling (FDM)	
Print Size	300x200x200 mm	
Print Precision	Highest	0.05 mm
	High	0.1 mm
	Normal	0.2 mm
	Low	0.3 mm
	Lowest	0.4 mm
Locational Precision	XY direction	11 um
	Z direction	2.5 um
Filament Diameter	1.75 mm	
Nozzle Diameter	0.4 mm	
Software		
Firmware	Marlin	
Host	Repetier Host	
Slicer	Cura, Slic3r	
File Type	.stl, .obj, .3ds	
Operation System	Windows	
	Linus	
	Mac	
Physical Size		
Machine Dimension	425 x 415 x 320 mm	
Package Dimension	440 x 430 x 350 mm	
Net Weight / <u>Gross</u> Weight	20 kg. / 23 kg.	
Temperature		
Surrounding Temperature	15°C ~ 35°C	
Storage Temperature	0°C ~ 35°C	
Electric		
AC input	100~240V	
Working Power	12V - 25Amp; 100~240V - 2Amp	
Connection	USB Cable& SD Card	
Mechanic		
Box	All Steel	
Working Plate	Steel & Aluminum	
XYZ Axis	Chrome plated high precision axis	
Stepper Motor	18°step angle@1/16 micro step	

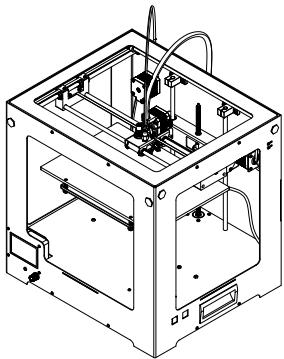
How does **JACBOT** work

JacBot makes objects from a digital file.

The creation of the object is achieved using additive processes. In an additive process an object is created by laying down successive layers of material until the entire object is created.

Each of these layers can be seen as a thinly sliced horizontal cross-section of the eventual object.



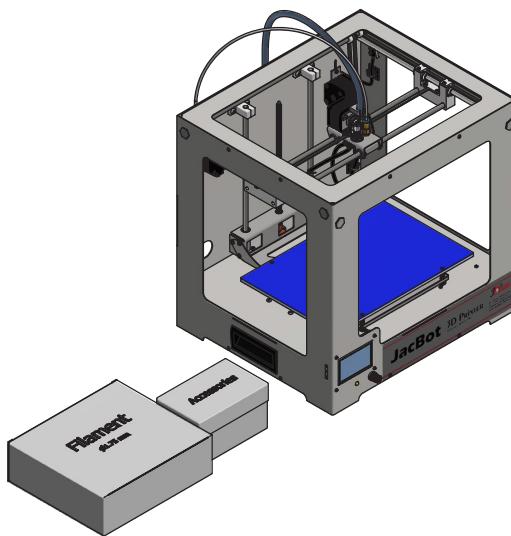
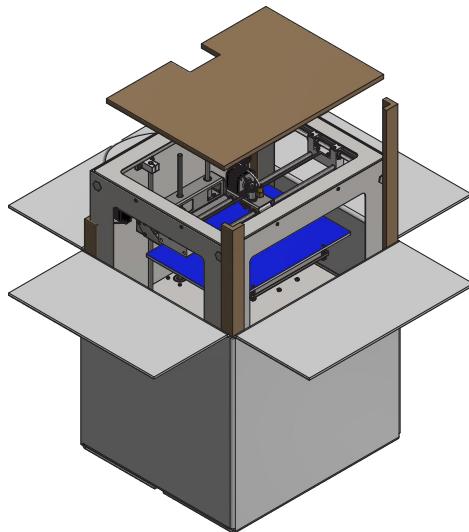


JacBot™

3D Printer

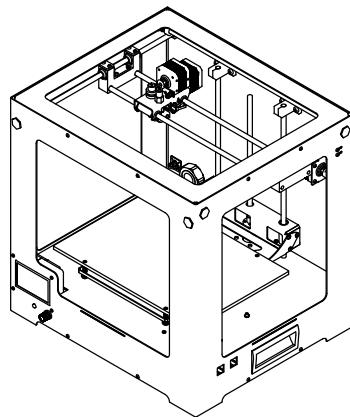
Unpack

Unpack the Machine

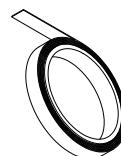
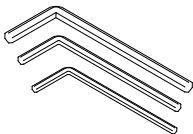
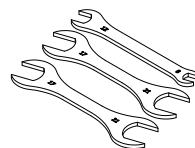
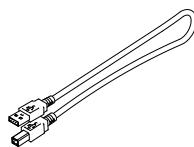
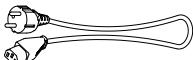
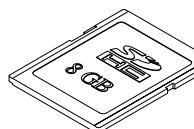
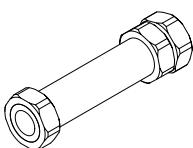


What in the Package

1. Machine

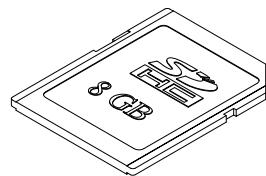


2. Accessories



3. Software and Sample files

Name	Date modified	Type	Size
Samples	18/03/16 08:38	File folder	
SoftWare	18/03/16 08:37	File folder	



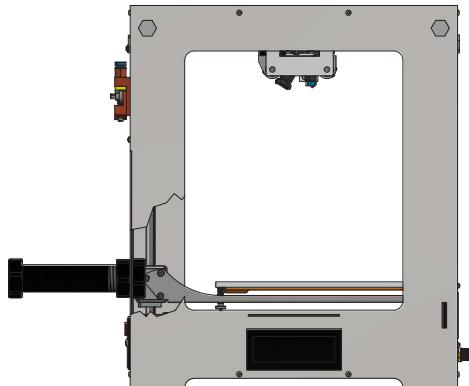
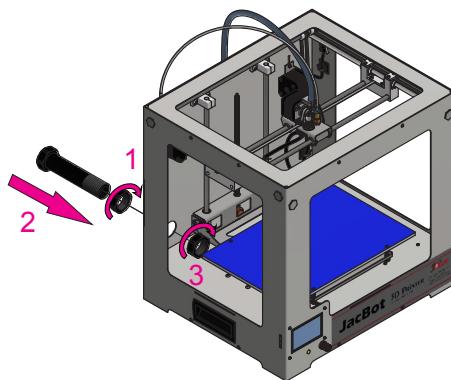
Software

Repetier Host for Windows, Mac & Linux

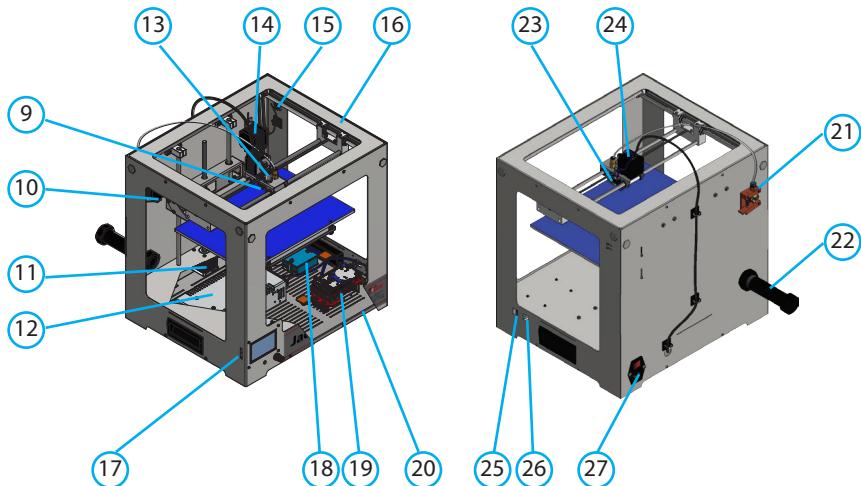
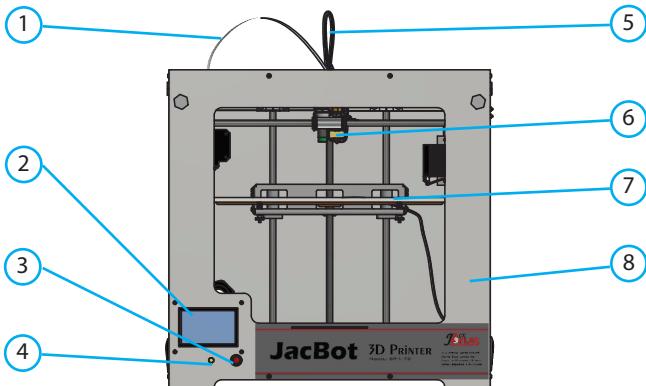
[Http://www.repetier.com/download-now/](http://www.repetier.com/download-now/)

Samples

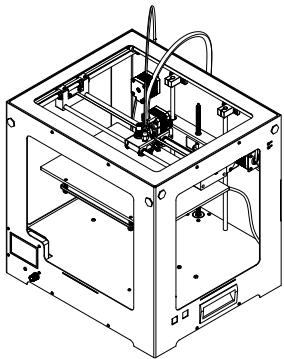
4. Mount Spool Holder



Machine Preview



-
- 1. Filament Tube
 - 2. Display
 - 3. Rotate and Push Button
 - 4. Emergency Stop Button
 - 5. Extruder Cable
 - 6. Extruder
 - 7. Working Plate
 - 8. Frame
 - 9. Fan
 - 10. Motor E
 - 11. Motor Z
 - 12. Power 12V
 - 13. Limited Switch Z
 - 14. Motor Y
 - 15. Limited Switch Y
 - 16. Top Cover
 - 17. SD Card Slot
 - 18. Relay
 - 19. Mini Computer
 - 20. Bottom Cover
 - 21. Filament Feeder
 - 22. Spool Holder
 - 23. Limited Switch X
 - 24. Motor X
 - 25. Power output 5V
 - 26. USB Cable Connector
 - 27. Power Connector and Switch 110~230V

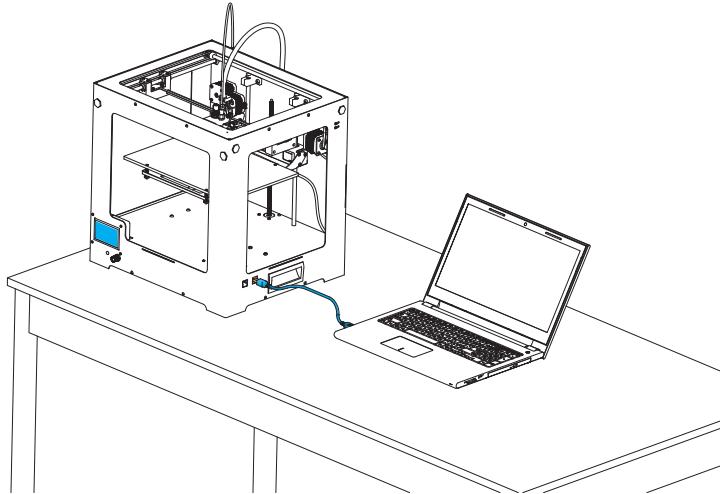


3D Printer

Install Software to
your Computer

Connect JacBot with Computer

1. Connection Via USB Cable

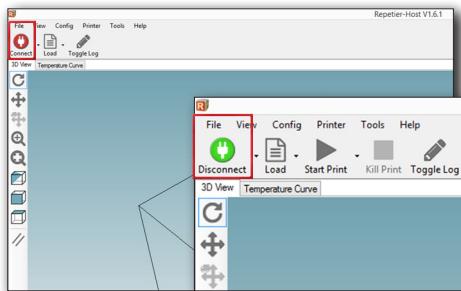


When the JacBot is connected with your computer, the LCD screen on the JacBot is light on, even though the machine did not connect AC Power. Because The LCD Display gets power from your computer.

Note: The power from your computer is just able to supply the power for the Micro-computer inside and the LCD display but can not drives motors.

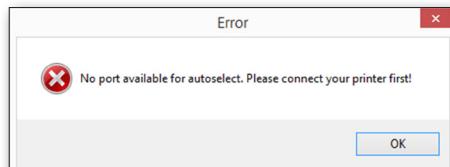
2. Connection on the Repetier Host

Both calibration of the machine and printing via USB cable request connection for communications between the JacBot and your computer.



Click the Red Icon 'Connect' at the left-Top of the Host and wait for seconds. If the connection established, the color of the Icon becomes to green.

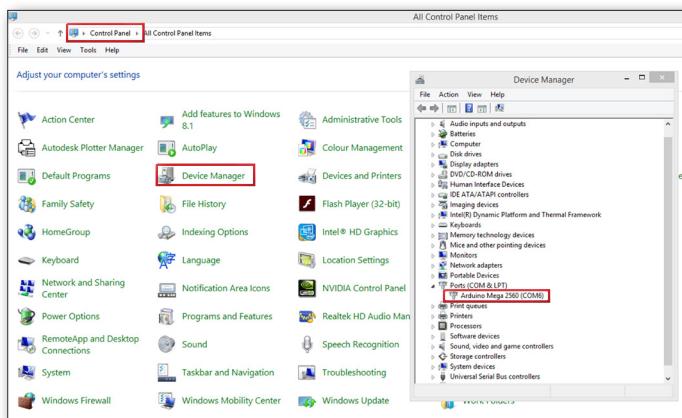
If the connection is fault, the Error message will pop out.



Solutions:

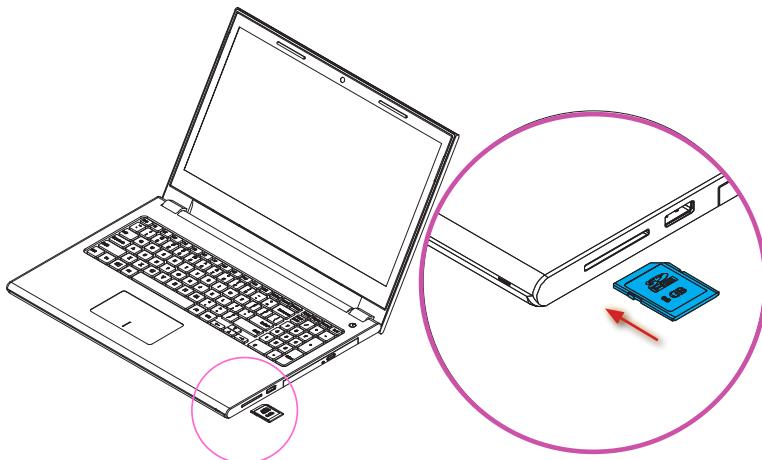
- Unplug USB cable and plug again;
- Close the Software and Open again;
- Check if your setting is correct;
(See '[1. Connection](#)' on Page 23).
- Change the 'Port' from 'Auto' to COM number

To find the port number, check in your computer through Control Panel, Device Manager, Ports(COM & LPT) then you can find 'Arduino Mega2560 (COMx)' (Here is COM6, it maybe different from yours.)



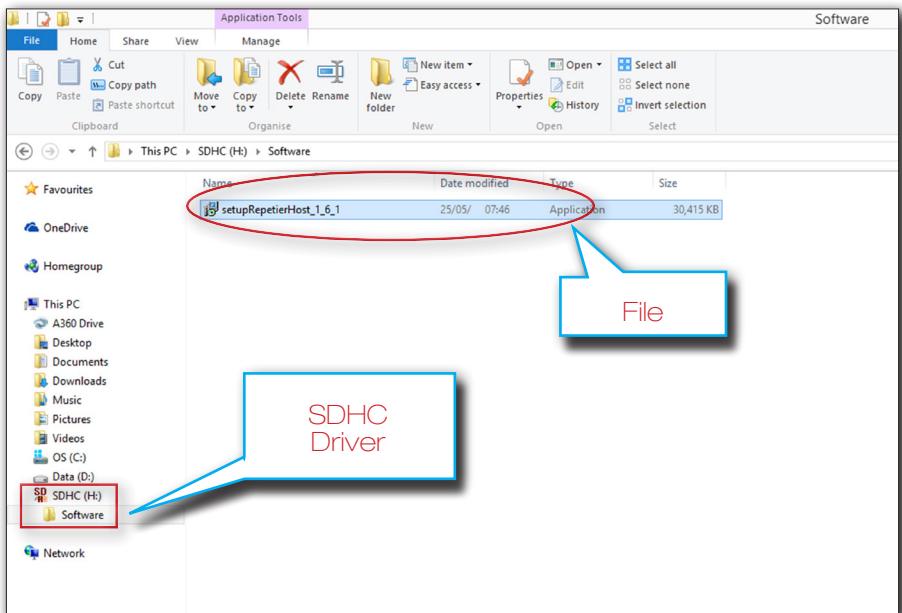
Install software to your computer

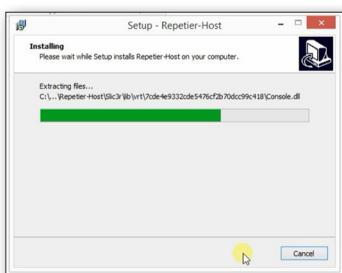
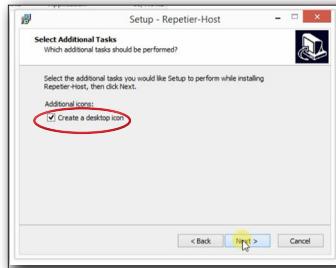
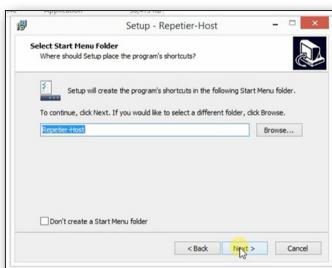
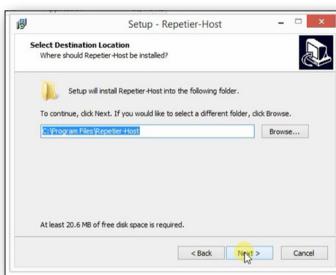
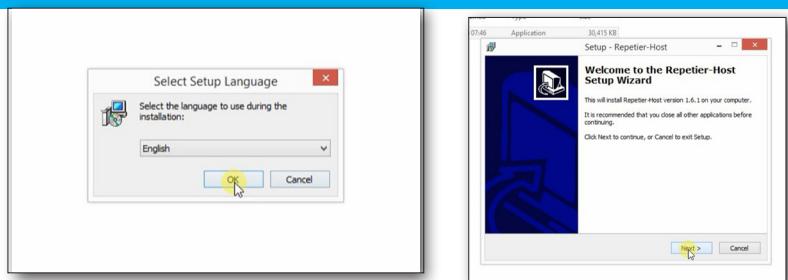
1. Install software from the SD card

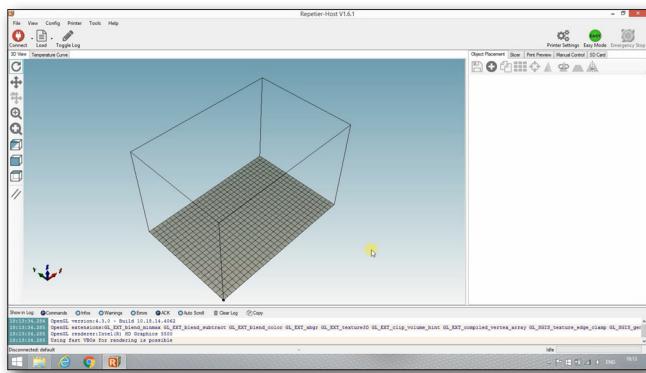


2. Installation

Go to “SDHC Driver” enter the “Software” folder, double click the file name to start the installation.







3. Update Software

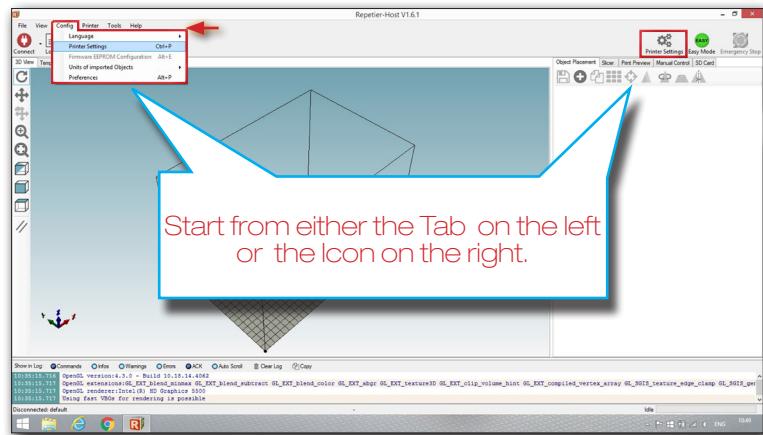
Visit the following Internet website to download the free software if there is newer version available.

[Http://www.repetier.com/download-now/](http://www.repetier.com/download-now/)

Printer Setting

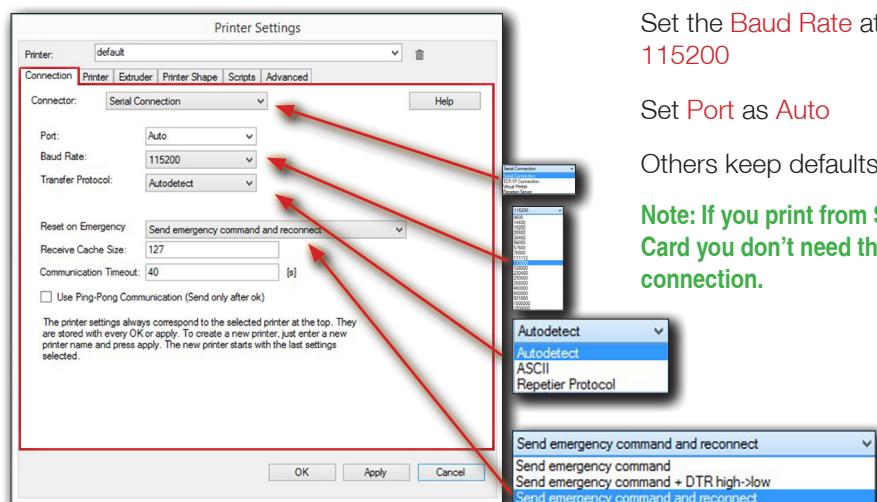
To make your JacBot Printer work properly, you shall make setting on the Repetier Host first.

Follow the below illustrated instruction to set your Jacbot step by step.

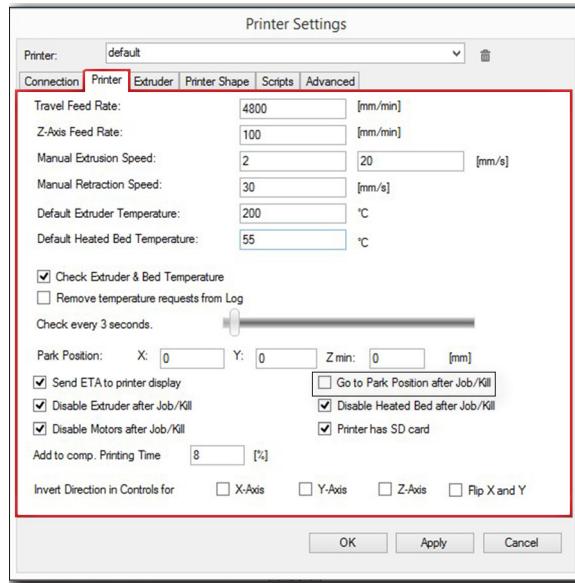


1. Connection

The Connection setting below is to establish a communication between your JacBot and the Host, Which uses for calibration the Printer and printing via USB cable.



2. Printer



The setting in this page is used for manual control via USB cable connection.

Travel feed rate

4800

A non-print speed of the Extruder moving at X and Y direction;

Z-axis feed rate

100

A non-print speed of the Working Plate moving up and down,

Manual Extrusion Speed & Manual Retraction Speed

The speeds of filament moving for-

ward or backward.

Default Extruder Temperature

200°C

Default Bed Temperature

55°C

Check Extruder & Bed Temperature

Checked

Park Position

The positions of the Extruder and

the Work Plate will be parked.

X=0, Y=0, the Extruder moves toward right and back sides;

Z=0, the Work Plate moves up.

If the difference between calculated time and real print time is great, you can increase the percentage here.

Send ETA to Printer Display

Checked

Invert Direction in Controls for

Invert move direction in specific axis.

Go to Park Position after Job/Kill

Unchecked

*Don't check the **Go to Park after Job/Kill**, otherwise your printed object probably collide against the extrude nozzle before you move the object away from the Work Plate.*

Disable Extruder after Job/Kill

Checked

Disable Heated Bed after Job/Kill

Checked

Disable Motors after Job/Kill

Checked

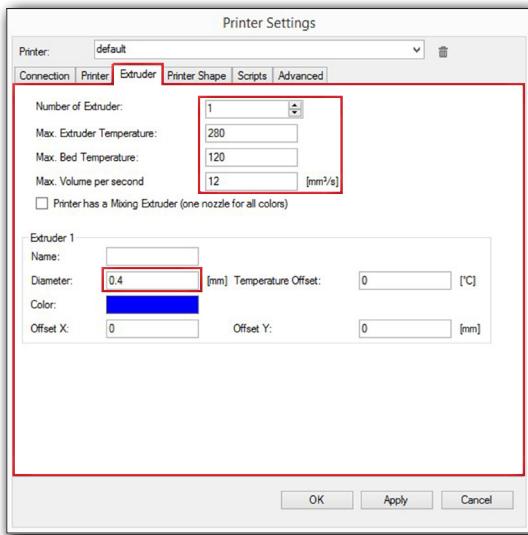
Printer has SD card

Checked

Add Comp. Print Time

8%

3. Extruder



Extruder Number

1

Max Temperature

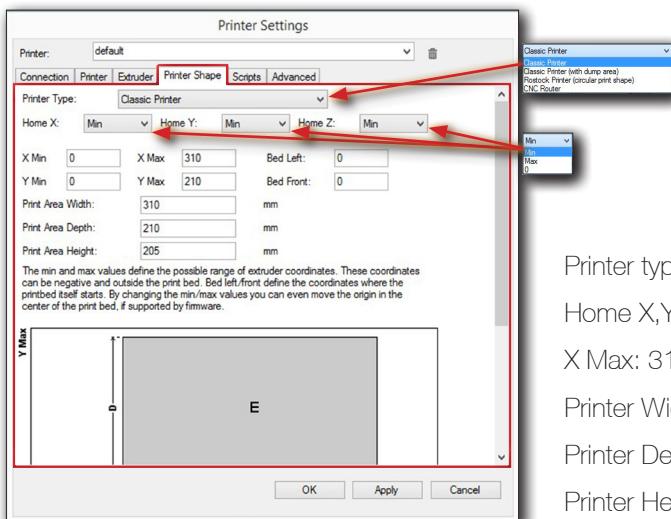
Extruder: 280

Bed 120

Extruder Diameter

0.4 mm

4. Printer Shape



Printer type: Classic Printer

Home X,Y,Z : Min

X Max: 310, Y Max: 210

Printer Width: 310

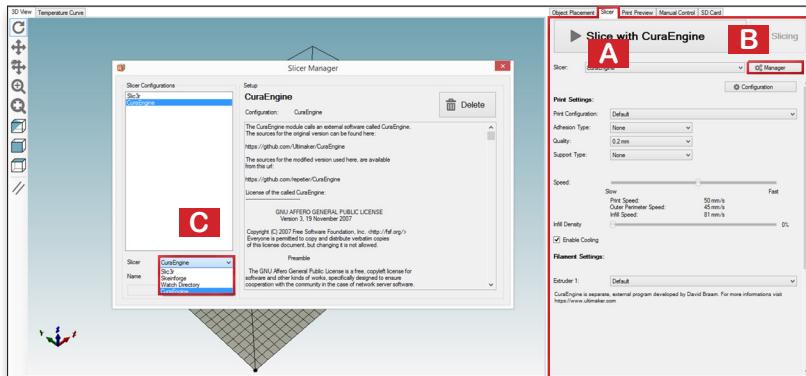
Printer Depth: 210

Printer Height: 200

5. Save the settings

Click OK or Apply button to save all your setting.

Import JacBot Configuration files



1. Select a Slice Tool

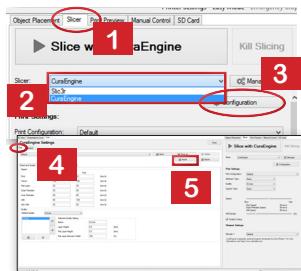
There are many tools to slice object in the Host.

A. Go to **Slice** TAB

B. Click **Manage** button

C. Click **Slicer** Drop List Box to select one or more slice tools.

2. Import CuraEngine Configuration



Print

Import Print configuration

[1] **Slicer** TAB

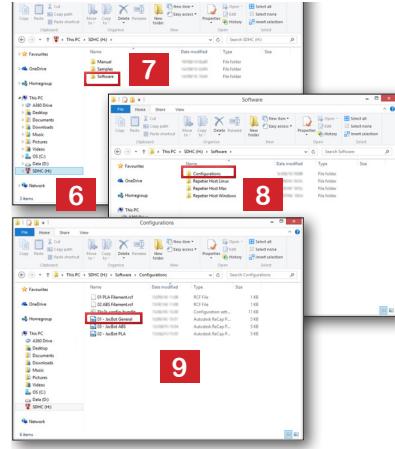
[2] Slicer: **CuraEngine**

[3] **Configuration** button

[4] **Print** TAB

[5] **Import** button

[6] SD Card position



[7] Software folder

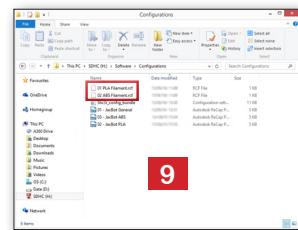
[8] Configuration folder

[9] 01 JacBot General

Filament

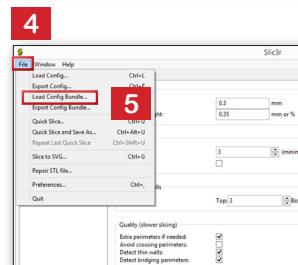
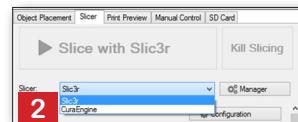
Import Filament configuration

- [1] Slicer TAB
- [2] Slicer: Cura Engine
- [3] Configuration button
- [4] Filament TAB
- [5] Import button
- [6] SD Card position
- [7] Software folder
- [8] Configuration folder
- [9] 01 PLA Filament & 02 ABS Filament



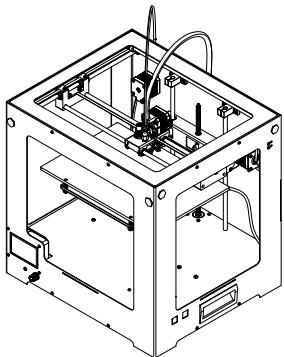
3. Import Slic3r Configuration

- [1] Slicer TAB
- [2] Slicer: Slic3r
- [3] Configuration button
- [4] File TAB
- [5] Load Config Bundle...
- [6] SD Card position
- [7] Software folder
- [8] Configuration folder
- [9] Slic3r_Config_Bundle



4. Delete the Default Configuration

Since the default setting in both the CuraEngine and the Slic3r are not suit with the machine, in order to avoid mistake, it is better to delete the default setting in both slicers.



JacBot™

3D Printer

Make your Print File

Work Process

1. Object Placement

Load one or more 3D model files.

2. Slice

Cut the 3D model into horizontal layers and generate G-Code which indicate tool-paths to fill them.

3. Preview

Check print path layer by layer.

4. Print

Print with computer directly or Print standalone by SD card .

Note:

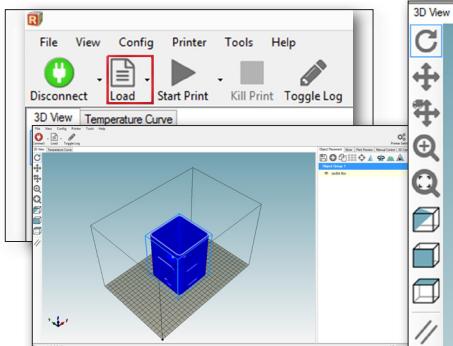
We recommend print by SD card, it is more stable. Print with computer is good for calibration only.

Place Object

1. How to get a 3D model file?

- Download from INTERNET website
- Scan a object by 3d scanner
- Draw by yourself with 3D software

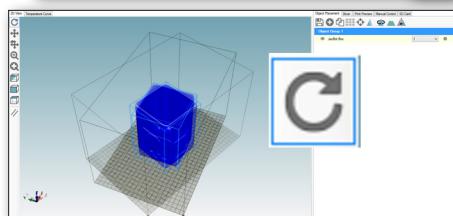
2. Load file and Familiar with the useful tools



Load

Load file from computer

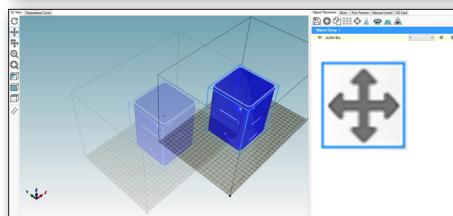
E.g. "Jacbot-Box.stl" from [Samples] folder in SD card.



Rotate View

It changes the view direction.

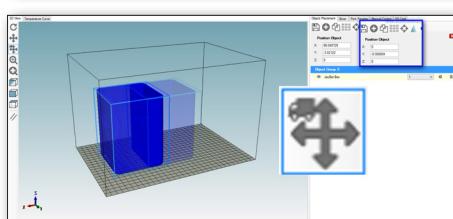
The object is not rotated.



Move View Point

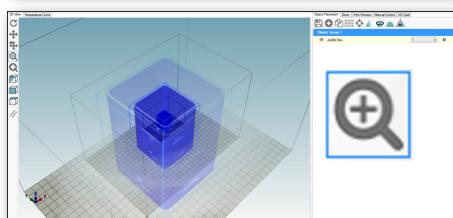
Move the point which is the center for rotate and zoom.

The object is not moved.



Move Object

The object is moved on the work plate.

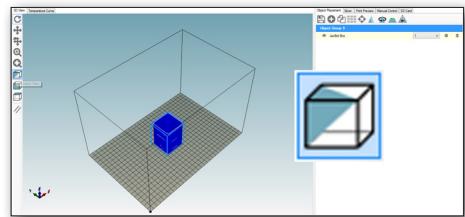


Zoom

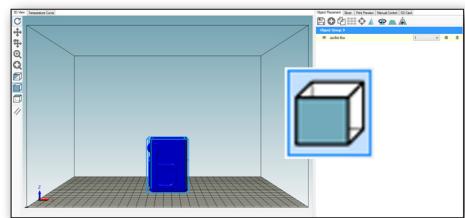
Zoom in or out.

The scale of the object is not changed.

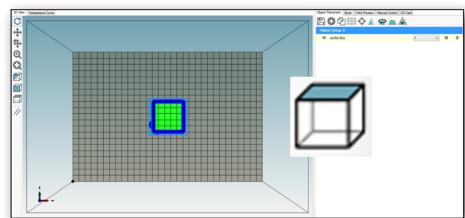
Isometric View



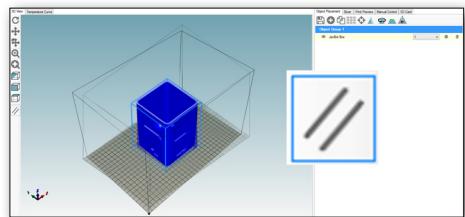
Front View



Top View



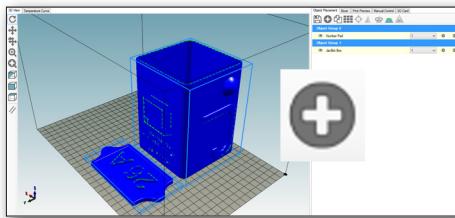
Parallel Show





Export

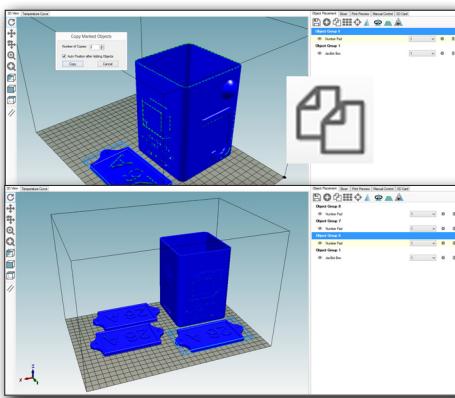
It is often used to save the file to **.stl** format when the file is placed here with other format.



Add

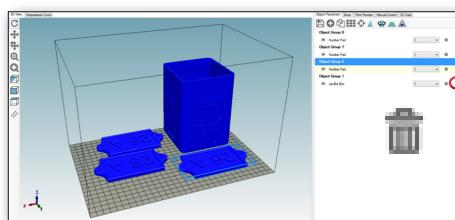
Add other object.

E.g. add "Number Pad.stl" from [Samples] folder in the SD card



Copy

Highlight the object you want to copy and fill the number of copies.

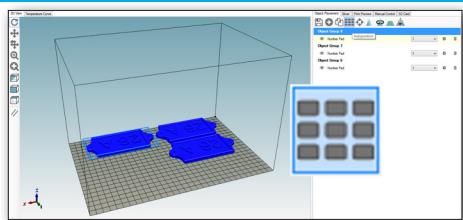


Remove

Highlight the Object you want to remove then click ashbin icon to remove the object.

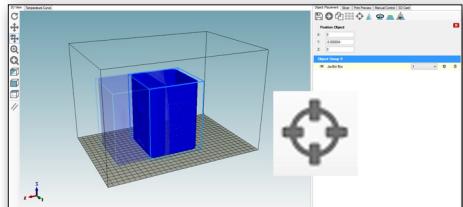
Auto Position

Move objects in right position automatically.



Center Object

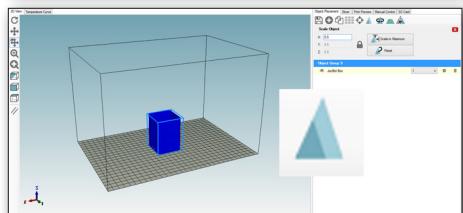
Move objects in center.



Scale Object

Grow or shrink object.

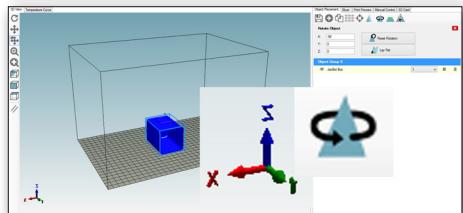
It changes the volume of the object. 0.5 is 50% of original



Rotate Object

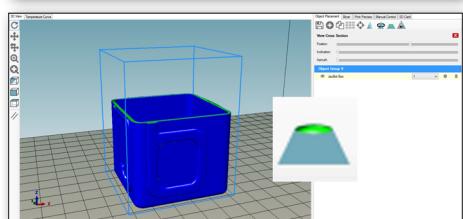
Rotate object round x, y, z shaft.

Ensure stability, avoid adding support and improve quality on specific surface.



Section View

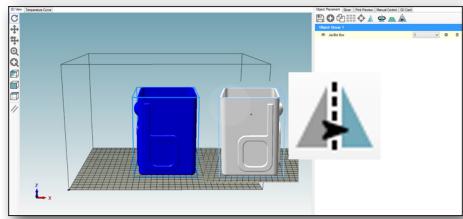
View the section of object.



Mirror Object

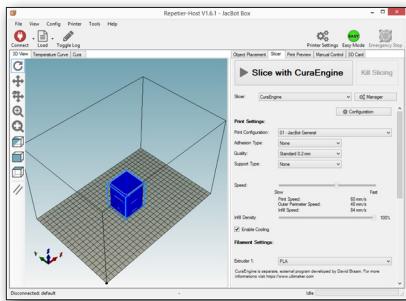
It makes the object mirrored.

It is useful to make symmetrical object with one file.



Slice Object

You can use either CuraEngine or Slic3r to generate G-Codes. CuraEngine is faster to slice, Slic3r has more details in generated G-codes.



1. CuraEngine

Slicer:

Cura Engine

Print Configuration:

01 JacBot General

If you cannot find it from the list, please import it from SD card. See '2. Import CuraEngine Configuration' on Page 27

Adhesion Type:

- None
- Brim
- Raft

Brim adds a belt around bottom of object, helps object to adhere on the work plate. The size of the Brim is changeable, see Appendix 'Skirt & Brim' on Page 63

Raft adds lays under object to pro-

tect the bottom of the object. The size of the Raft is changeable, see Appendix 'Raft' on Page 64

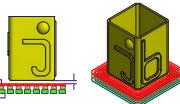
None



Brim



Raft



Quality:

- Standard 0.2 mm
- Best 0.1 mm
- Fast 0.3 mm

Best provides high quality but takes long time, **Fast** prints fast but loses quality.

Standard is recommended.

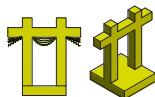
Support Type:

- None
- Touching Bed
- Everywhere

Supports are used when models have steep overhangs or unsupported areas.

Checked

None



Touching Bed



Everywhere



Extruder 1:

- PLA
- ABS

If you cannot find it from the list, please import it from SD card. See '2. Import CuraEngine Configuration' on Page 27

PLA is PLA filament with 190 °c Extruding temperature and 60 °c bed temperature.

ABS is ABS filament with 240 °c Extruding temperature and 110 °c bed temperature.

To change the setting, please make new configuration, see Appendix 'Filament' on Page 69

Speed:

60 mm/s is a standard speed

The lower of the speed, the higher of the quality.

The speed range is changeable, see Appendix 'A. Speed and Quality' on Page 59

Infill Density:

0 -- No infill

20%~30% is a standard

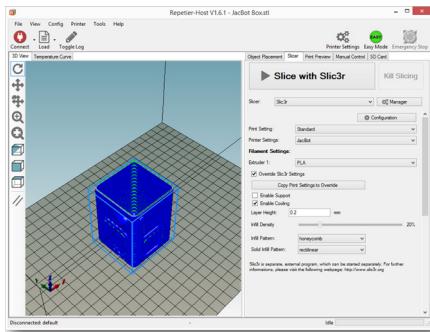
100% is a solid

Higher density of infill makes strength of object higher, but consumes more filament and time.

20%~30% is common used density.

Enable Cooling

2. Slic3r



Slicer:

Slic3r

Print Settings:

- Best 0.1 mm + Brim
- Best 0.1 mm + Raft
- Best 0.1 mm
- Fast 0.3 mm + Brim
- Fast 0.3 mm + Raft
- Fast 0.3 mm
- Standard 0.2 mm + Brim
- Standard 0.2 mm + Raft
- Standard 0.2 mm

If you cannot find it from the list, please import it from SD card. See '3. Import Slic3r Configuration' on Page 28

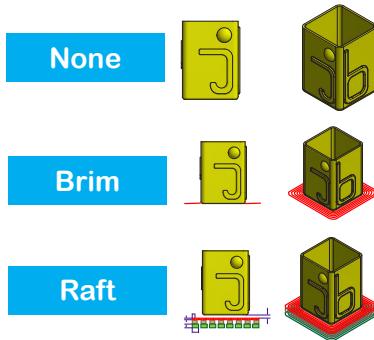
Best provides high quality but takes long time, **Fast** prints fast but loses quality.

Standard is recommended.

Brim adds a belt around bottom of

object, helps object to adhere on the work plate.

Raft adds lays under object to protect the bottom of the object



Printer Settings:

JacBot

Extruder1:

PLA

ABS

PLA is PLA filament with Extruding temperature for the first layer at 200 °c , others at 190°c and bed temperature for the first layer at 60 °c, others no heating.

ABS is ABS filament with Extruding temperature for the first layer at 240 °c , others at 230°c and bed temperature at 110°c for all layers

To change the setting, please make new configuration.

Override Slic3r Settings:

Checked

and less than 0.4 mm.

Settings below are not available if the option is unchecked. All setting below keep last settings.

Copy Print Settings to Override

Click the button to restore original setting, otherwise all settings below are still the last one which may not be expected this time.

Enable Support

No Support



Support



Checked

Support everywhere

Unchecked

No Support

Enable Cooling

Checked

Layer Height:

Keep original setting or input new height which shall be more than 0

Infill Density:

0 -- No infill

20%~30% is a standard

100% is a solid

Higher density of infill makes strength of object higher, but consumes more filament and time.

0 % is useful for printing object's shell.

20%~30% is common used density.

Note: Not all Infill Patten supports 100% of the infill density.

Infill Patten:

Keep original setting

If you selected 100% Infill Density above, you shall choose the rectilinear type patten.

Solid Infill Patten:

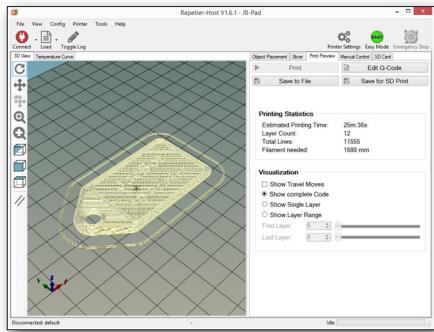
Keep original setting

If you selected 100% Infill Density above, you shall choose the rectilinear type patten.

Preview

After slice object, the page is opened

1. Printing Statistics



Estimated Printing Time:

The time is estimated, real time may be longer.

To adjust the time, please see 'Add Comp. Print Time' on Page 25

Layer Count:

Number of layers into which object is sliced

Total Line:

The number of line's segments on the extruder's path.

Filament Needed:

The length of filament consumed.

2. Visualization

Show Travel Moves

Checked

Display the travel path.

Unchecked

Display extrusion path only.

Show Complete Code

Display printed object

Show Single Layer

Display the path on current layer

Show Layer Range

Display the path on a range of layers.

- Print button

Click the button to print via USB cable.

Print via USB cable is not stable and easily interfered by computer's working environment.

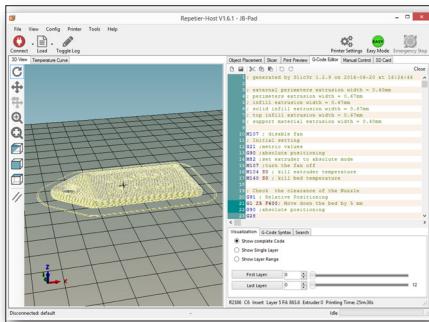
It's often used for machine calibration and small object print.

To print via USB cable, click connection icon on top-left if not connected. See 'Connect JacBot with Computer' on Page 18

LCD display on the machine will light on while USB cable is connected, but the power is not enough to drive the machine. Don't forget to connect AC power and switch it on.

- Edit G-Code button

Click the button, G-Code Editor page is displayed.



If you don't satisfy with the G-code which is generated by the slicer automatically, you can edit the G-Code here.

- Save to File

Click the button to save the G-Code file to computer.

This G-Code file doesn't include Start and End codes.

- Save for SD Print

Click the button, a dialog window pop out.

Include "Start" and "End" Code

Checked



Include Job Finished Commands

Checked

Save in Binary Format

Unchecked

*Click **Save** button, find your SD card, give a name then to save.*

The name's length is better to limit within 18 characters (includes blank) otherwise it will be over LCD display range.

Take out the SD card from your computer and insert the card into your printer, now you can print independently from computer.

Print

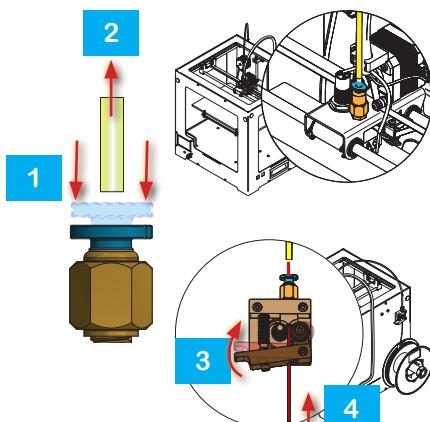
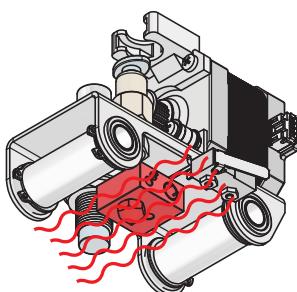
1. Load / Change Filament

Heat Extruder

Temperature

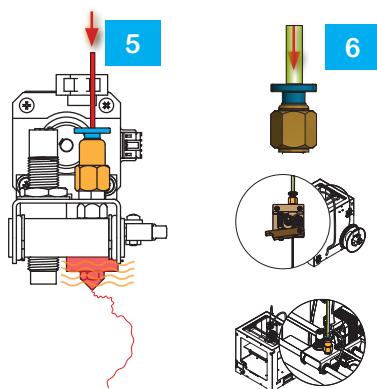
PLA : 230°C

ABS: 250°C



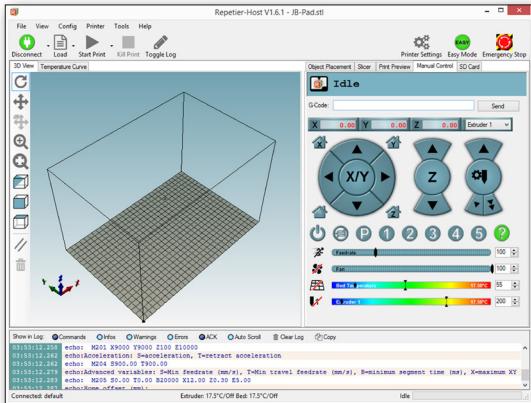
Load filament

- [1] Press the head down
- [2] Pull out the filament tube
- [3] Open the guide wheel
- [4] Insert the filament through the feeder and into the tube
- [5] Insert the filament into the extruder till it is extruded out smoothly.
- [6] Insert the filament tube back into the connectors.



2. The First Print

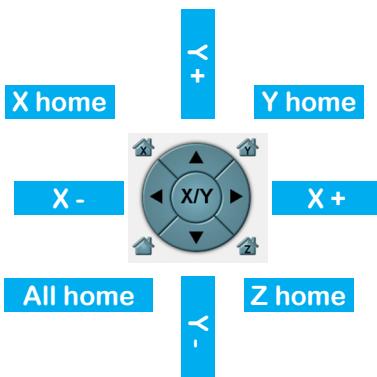
Manual Control



G -Code

Input G-Code, then click send button,
Input G28, the extruder and bed will go home; input G29, auto bed leveling

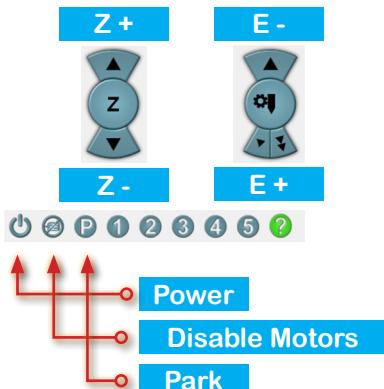
Home & X/Y Movement



Z (Bed) Movement

E (Filament) Movement

Turn on/off Power

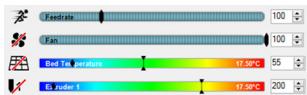


Disable/Enable Motors

Disable motors when you move a axis by hand.

Park

The Extruder will go to park.



Feed Rate

Print Speed

Fan

Fan On/Off and Speed

Bed Temperature

On/Off and degree

Extruder Temperature

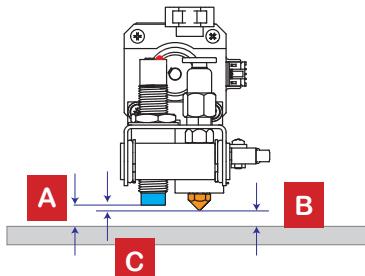
On/Off and degree

Bed Leveling

After go home, input G29 and click send button to make auto bed leveling, read the z positions at different point from log window (If the window disappear, click Toggle Log icon on the top to display the window.)

Adjust the distance by driving screws on the bed to try make each point position as close as possible.

Z Probe



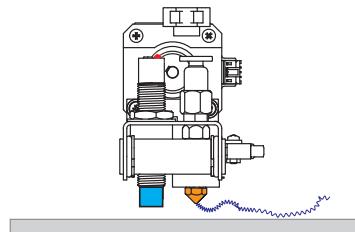
- The probe maximum detect distance is about 2 mm.

A is trigger distance about 1.5 ~ 2mm

B is distance between the nozzle and the bed. 0.2 mm is better distance to stick extrusion on the bed.

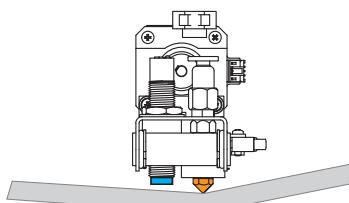
C is distance of the probe over the nozzle, at least 0.5 mm.

Increase A will make B shorten; reduce A will make B large.



A too small

A too small, the nozzle becomes too high.

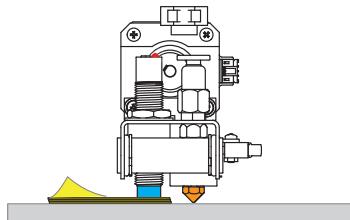


A too large

A too large, the nozzle touches the bed. In this case, turn off the power immediately.

- Choose Correct Distance

Release the screws of the probe to adjust the distance.



Turn off the power, lift the bed to touch the nozzle by hand.

Turn on the power, put 3~6 pieces of papers under the probe till the light on the probe is off, remove one or two pages, the light on again. Fix the screws.

Let machine goes home to check if the probe can stop the bed, then print a small object to see if extrusion can stick on the bed.

If the probe can stop the bed, but the first layer is too thin or thick or can not stick on bed at all, then we can solve the problem by edit G-Code.

- Adjust the distance by G-Code

At the end of the Start G-Code, between the two star lines, there is code like this,

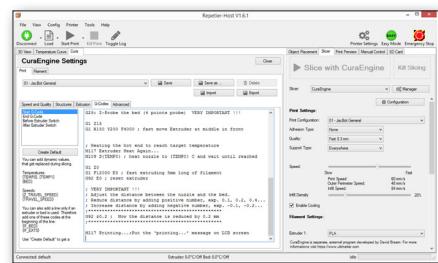
G92 Z0.2

This means the distance between the nozzle and bed will be reduced by 0.2 mm

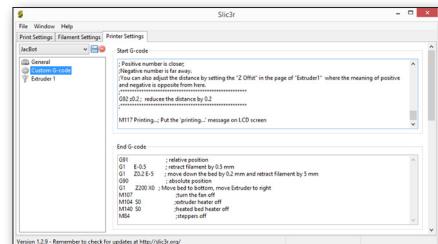
G92 Z-0.1

This means the distance between the nozzle and bed will be increase by 0.1 mm

Start G-Code in CuraEngine



Start G-Code in Slic3r

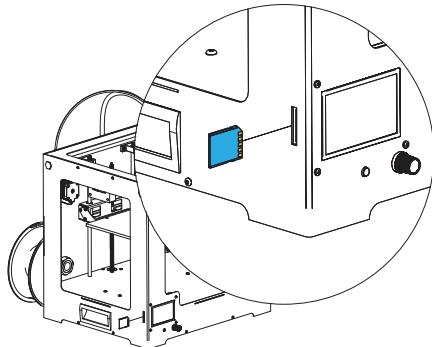


It is very useful to adjust so small distance that physical adjustment is very difficult.

See Appendix ‘Improve our Start Code:’ on Page 50

3. SD Card Print

Insert SD Card into SD Card Slot



Push Encode button

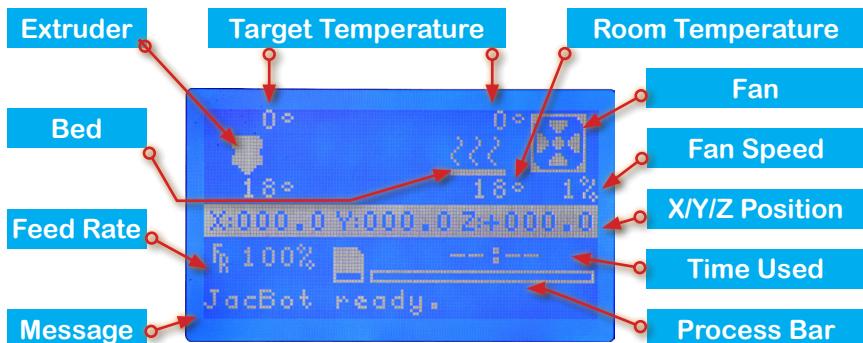
Push (Rotate and Push Button) to enter menu

Push again to select or confirm.

Turn the Encode button

On Main page set Feed Rate

In Menu page, Browse



Target Temperature is Extruder & bed temperature your set.

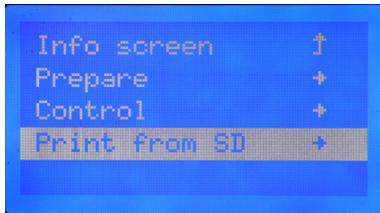
Message displays information such as error, file name, etc.

Feed Rate is manual speed control.

Print from SD Card

Press to enter menu

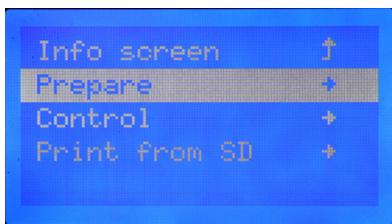
Turn to browse to **Print from SD**, press again to enter **File dictionary**, choose a file and press to confirm for printing



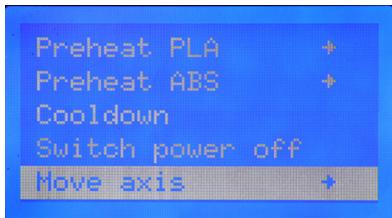
Move X.Y.Z.E axis

Press to enter menu

Turn to browse, press again to enter **Prepare** sub-menu



Turn to browse, press again to enter **Move Axis** sub-menu

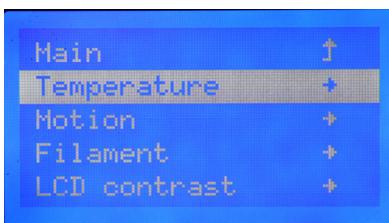


Follow the menu to control axis movement.

Set Temperature

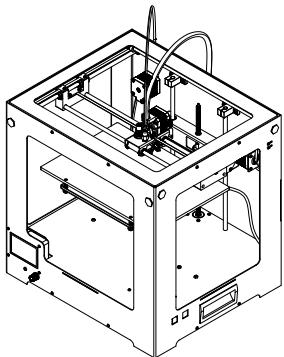
Press to enter menu

Turn to browse, press again to enter **Control** sub-menu



Turn to browse, press again to enter **Temperature** sub-menu. Follow the menu to set temperature.

This often uses to heat extruder for load or change filament, or adjust temperature during printing.



JacBot™

3D Printer

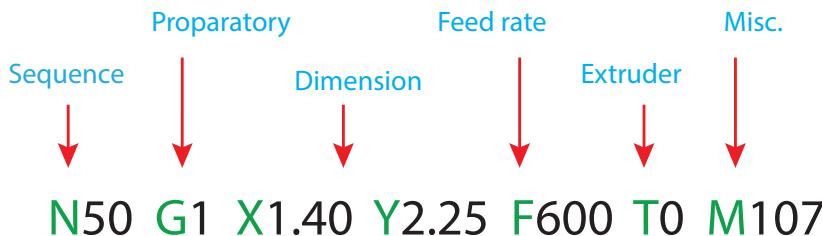
Appendix

G-Code

1. What is G-Code

G-Code is the generic name for a control language for 3D printer. It indicates the machine to move to various points at a desired speed, control the temperature, turn on and off fan, and all sorts of other things. It is fairly standard, and is a useful tool.

2. Format of G-Code



- N - Sequence or line Number
- G - Preparatory Functions

G Codes specify the mode in which the printer's extruder or work plate is to move along its programmed axis.

- X/Y/Z/E - Distance or Position in X/Y/Z/E.

E is filament extrusion distance or position

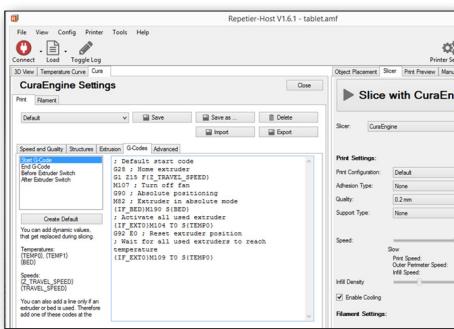
- M - Miscellaneous

M Codes specify 3D Printer functions not related to dimensions or axis movements.

- F - Feed Rate
- T - Extruder Number

3. How the G-Codes work in the JacBot printer

The G-Codes are generated by Slice software such as Cura, Slic3r and so on, and control the printing from the beginning to the end. They consist of three sections: Start Codes, Printing Codes and End Codes.

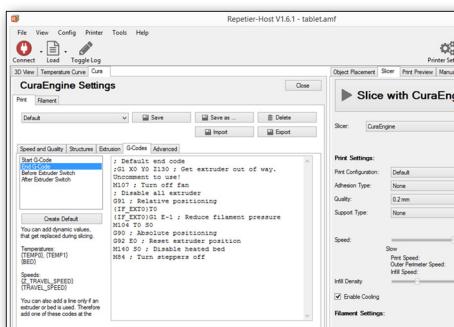


4. Start G-Code

In default setting of both the Cura and the Slic3r are very simple.

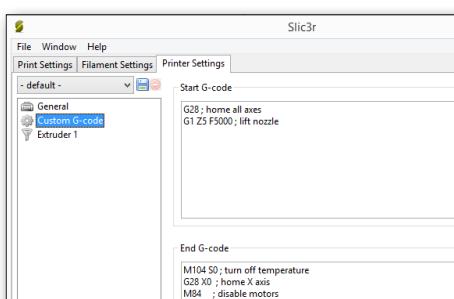
Cura:

- Go home
- Move down the work plate
- Set positions



Slic3r:

- Go Home
- Move Down the work plate



Both of them have not Auto Bed Leveling function. (G29)

Without the G29 word, the machine works as normal printer, you shall level the work plate (Bed) manually before printing, which often result in failure of printing.

Improve our Start Code:

- **Initial setting**

Unit shall be metric (G21);

- **Clean the extruder nozzle**

Leave a space to check the cleanliness of the nozzle. Since the best distance between the nozzle and bed is about 0.2 mm, so any hard filament attached on the nozzle probably resist the bed up to the distance in case of HOME and AUTO BED LEVELING.

- **Heat the Bed**

Set the bed temperature automatically, you don't need to set the temperature manually in Repetier Host each time.

- **Auto Bed Leveling (G29 after Home)**

G29 only activate after G28

G29 makes Z probe at 4 or 9 points of the bed to remember the difference of the distances. Thus the compensation can be made to keep the bed leveling by moving the bed up or down during printing.

Without the G29, the printer will work as normal printer, and the bed level shall be adjusted manually.

- **Heat the extruder nozzle**

Set the nozzle temperature automatically, you don't need to set the temperature manually in Repetier Host each time.

- **Reset the filament start position and bed height**

The distance between the nozzle and the bed is so important that it determines whether the first layer of printing can be adhered on the bed successfully.

G92 Z0.1 (or 0.2 or 0.45 or 0.5) means shorten the distance by 0.1 mm (or 0.2 or 0.45 or 0.5 mm);

G92 Z-0.1 (or -0.2 or -0.3) means increasing the distance by 0.1mm (or 0.2 or 0.3 mm)

Example:

; Initial setting

G21 ;metric values
G90 ;absolute positioning
M82 ;set extruder to absolute mode
M107 ;turn the fan off
M104 S0 ; kill extruder temperature
M140 S0 ; kill bed temperature

; Check the clearance of the Nuzzle
G28 ; Go home
G1 Z15 F800; Move down the bed by 15 mm
G1 X150 Y200 F4000 ; fast move Extruder at middle in front

; Heating the heat bed
M140 S60 ;Set Bed Temp 60 C

; Preheating the hot end
M109 S180 B230 F1;activate auto temp min 180 max 230 scaling factor 1 (for PLA)

; Auto bed leveling
G28: Home Axis
G29; Z-Probe the bed (4/9 points probe) **VERY IMPORTANT !!!**

; Heating the hot end to reach target temperature
M109 S210 ; heat nozzle to 210 C and wait until reached

G1 F12000 E5 ; fast extruding 5mm long of filament
G92 E0 ; reset extruder

; Adjust the distance between the nozzle and the bed, **VERY IMPORTANT !!!**
; Adjust the distance between the nozzle and the bed.
; Reduce distance by adding positive number, exp. 0.1, 0.2, 0.4...
; Increase distance by adding negative number, exp. -0.1, -0.2...
; *****
G92 z0.4 ; decrease the distance by 0.4 mm
; *****
M117 Printing...;Put the 'printing...' message on LCD screen

Note:

The semicolon “;” means comment, and words followed it will be ignored by program. You can write in any language and any words which help you to understand the meanings of your codes.

Add the semicolon “;“ before the codes same as deleting the code, it is good way to test your code and easy to change back by removing off the semicolon. For

instance, we don't need the Preheating the Hot-End, just add ";" before the M109 S108 B230 F1

; M109 S108 B230 F1

Above example is setting for PLA filament, to set for ABS filament just edit the temperature setting.

Example: For ABS Filament

; Initial setting

G21 ;metric values

G90 ;absolute positioning

M82 ;set extruder to absolute mode

M107 ;turn the fan off

M104 S0 ; kill extruder temperature

M140 S0 ; kill bed temperature

; Check the clearance of the Nuzzle

G28 ; Go home

G1 Z15 ; Move down the bed by 15 mm

G1 X150 Y200 F4000 ; fast move Extruder at middle in front

; Heating the heat bed

M140 S110 ;Set Bed Temp 110 C

; Preheating the hot end

M109 S210 B260 F1 ;activate auto temp min 210 max 260 scaling factor 1 (for ABS)

; Auto bed leveling

G28; Home Axis

G29; Z-Probe the bed (4/9 points probe) **VERY IMPORTANT !!!**

; Heating the hot end to reach target temperature

M109 S240 ; heat nozzle to 240 C and wait until reached

G1 F12000 E5 ; fast extruding 5mm long of filament

G92 E0 ; reset extruder

*; Adjust the distance between the nozzle and the bed, **VERY IMPORTANT !!!***

; Adjust the distance between the nozzle and the bed.

; Reduce distance by adding positive number, exp. 0.1, 0.2, 0.4...

; Increase distance by adding negative number, exp. -0.1, -0.2...

*; ******

G92 z0.4 ; decrease the distance by 0.4 mm

*; ******

M117 Printing... ;Put the 'printing...' message on LCD screen

5. End G-Code

When the print is finished, we hope:

- Moves the extruder from the object we printed;
- Retract filaments a little to avoid filament leak from nozzle
- Turn off fan and temperature
- Disable the stepper motors

Understand the G90 and G91.

G90 is absolute position which measures a distance from home position.
G91 is relative position which measures a distance from last position.

Example:

```
G91          ; relative position
G1 E-0.5    ; retract filament by 0.5 mm
G1 Z0.2 E-5  ; move down the bed by 0.2 mm and retract filament by 5 mm
G90          ; absolute position
G1 Z201 X0 Y0 F800 ; Move bed to bottom, move Extruder to home
M107        ; turn the fan off
M104 S0     ; extruder heater off
M140 S0     ; heated bed heater off
M84         ; steppers off
```

6. Table of Common G-Codes

	Parameters	Description	Example
G Code			
G0	Axis [X/Y/Z] Position	Rapid Movement <i>Move at maximum speed</i>	G0 X50
G1	Axis [X/Y/Z/E] Position Feed [F]	Coordinated Movement (XYZ E) <i>Move at preset feed rate speed</i>	G1 F150 X10
G2	I, J	Clockwise Arc <i>Center offset from current position</i>	G2 I10.5 J10.5
G3	I, J	Counter-clockwise Arc <i>Center offset from current position</i>	G2 I10.5 J10.5
G4	Time in ms [P]	Dwell / Wait	G4 P500
G10	None	Retract filament according to settings of M207	
G11	None	Retract recover filament according to settings of M208	
G20	None	Set units to inch	G20
G21	None	Set units to mm	G21
G28	X/Y/Z	Home	G28 X Y
G29	None	Z Prob <i>For auto bed leveling. Only activated after G28</i>	G29
G90	None	Absolute Positioning	G90
G91	None	Relative Positioning	G91
G92	Axis [X/Y/Z/E] Value	Set Current Position as Absolute Position <i>It is much useful for adjust the distance between Extruder Nuzzle and Work Plate. Increase value to shorten the distance; decrease the value to increase the distance.</i>	G92 E0 G92 Z0.2 G92 Z0.4 G92 Z-0.1
M Word			
M0	None	Stops everything after buffer is empty	M0
M17	None	Enable all stepper motors <i>Move axis by hand heavily</i>	M17
M18	None	Disable all stepper motors <i>Move axis by hand freely</i>	M18
M20	None	List files at the root folder of the SD Card	M20
M21	None	Initialize (mount) SD Card	M21
M22	None	Release (unmount) SD Card	M22
M23	Filename	Select File for Printing	M23 print.gco
M24	None	Start / Resume SD Card Print (see M23)	M24
M25	None	Pause SD Card Print (see M24)	M25
M26	Bytes[S]	Set SD Position in bytes	M26 S12345
M27	None	Report SD Print status	M27

	Parameters	Description	Example
M28	Filename	Write program to SD Card	M28 print.gco
M29	Filename	Stop writing program to SD Card	M29 print.gco
M40	None	Eject part (if possible)	M40
M41	None	Loop Program(Stop with reset button!)	M41
M42	None	Stop if out of material (if supported)	M42
M43	None	Like M42 but leave heated bed on (if supported)	M43
M80	None	Turn on ATX Power (if necessary)	M80
M81	None	Turn off ATX Power (if necessary)	M81
M82	None	Set extruder to absolute Mode (default)	M82
M83	None	Set Extruder to relative mode while in Absolute Coordinates (G90) mode	M83
M84	None	Stop idle hold <i>DO NOT use while printing!</i>	M84
M92	Steps_per_unit[X]	Program set S steps per unit (resets)	M92 X123
M101	None	Set extruder 1 to forward (outdated)	M101
M102	None	Set extruder 1 to reverse (outdated)	M102
M103	None	Turn all extruders off (outdated)	M103
M104	Temperature[S]	Set extruder temperature (not waiting)	M104 S100
M105	None	Get extruder Temperature	M105
M106	PMW [S 0-255]	Set Fan Speed to S and start	M106 S123
M107	None	Turn Fan off	M107
M108	None	Set extruder speed (outdated)	M108
M109	Temperature[S]	Set extruder Temperature (waits till reached)	M109 S123
M110	Line Number[N]	Set current line number (next line number = line no. +1)	N123 M110
M111	Debug Level [S]	Set Debug Level	M111 S6
M112	None	Emergency Stop (Stop immediately)	M112
M113	PMW[S]	Set Extruder PWM to S (or on-board potent. If not given)	M113 S0.7
M114	None	Get Current Position	M114
M115	None	Get Firmware Version and Capabilities	M115
M116	None	Wait for ALL temperatures	M116
M117	None	Display Message on LCD	M117 Hello ... M117 Z-Probe
M119	None	GET Ends-top Status	M119
M126	Time[P]	Open extruder valve (if available) and wait for P ms	M126 P500
M127	Time[P]	Close extruder valve (if available) and wait for P ms	M127 P500

	Parameters	Description	Example
M128	PWM[S]	Set internal extruder pressure S255 eq max	M128 S123
M129	Time[P]	Turn off extruder pressure and wait for P ms	M129 P500
M140	Degrees[S]	Set heated bed temperature to S (not waiting)	M140 S55
M141	Degrees[S]	Set chamber temperature to S (not waiting)	M141 S30
M142	Pressure[S]	Set holding pressure to S bar	M142 S1
M143	Degrees[S]	Set maximum hot-end temperature	M143 S275
M160	No.[S]	Set number of materials extruder can handle	M160 S4
M190	Degrees[S]	Set heated bed temperature to S (waiting)	M190 S55
M203	Offset[Z]	Set Z offset (stays active even after power off)	M203 Z-0.1
M226	None	Pauses printing (like pause button)	M226
M227	Steps[P/S]	Enables Automatic Reverse and Prime	M227 P1500 S1500
M228	None	Disables Automatic Reverse and Prime	M228
M229	Rotations[P/S]	Enables Automatic Reverse and Prime	M229 P1.0 S1.0
M230	[S]	Enable / Disable wait for temp.(1 = Disable 0 = Enable)	M230 S1
M240	None	Start conveyor belt motor	M240
M241	None	Stop conveyor belt motor	M241
M245	None	Start cooler fan	M245
M246	None	Stop cooler fan	M246
M300	Freq.[S] Duration[P]	Beep with S Hz for P ms	M300 S300 P1000

T Word

T	No.	Select extruder no. (Starts with 0)	T0

Configuration of Slicer

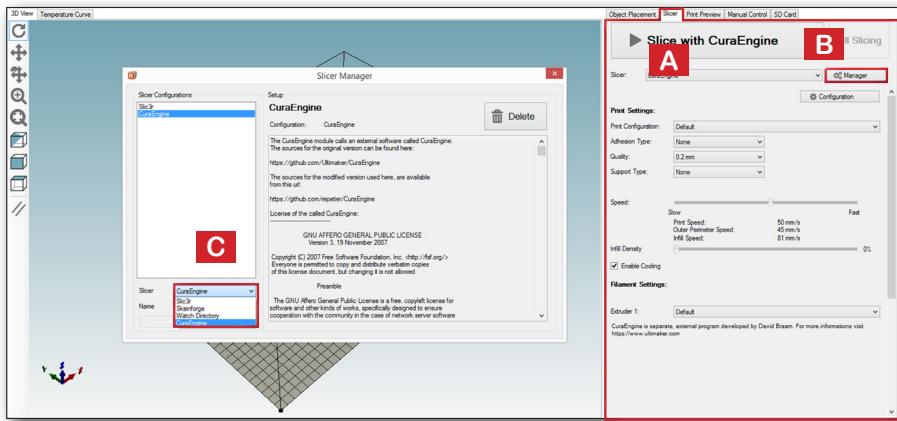
1. Select a slice tool

There are many tools to slice object in the Host.

A. Go to **Slice TAB**

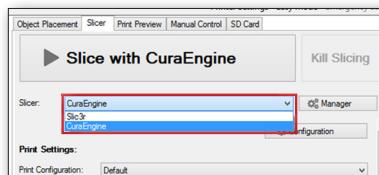
B. Click **Manage** button

C. Click **Slicer** Drop List Box to select one or more slice tools.

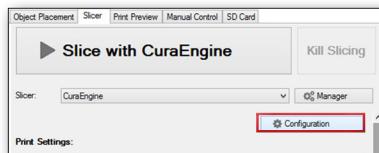


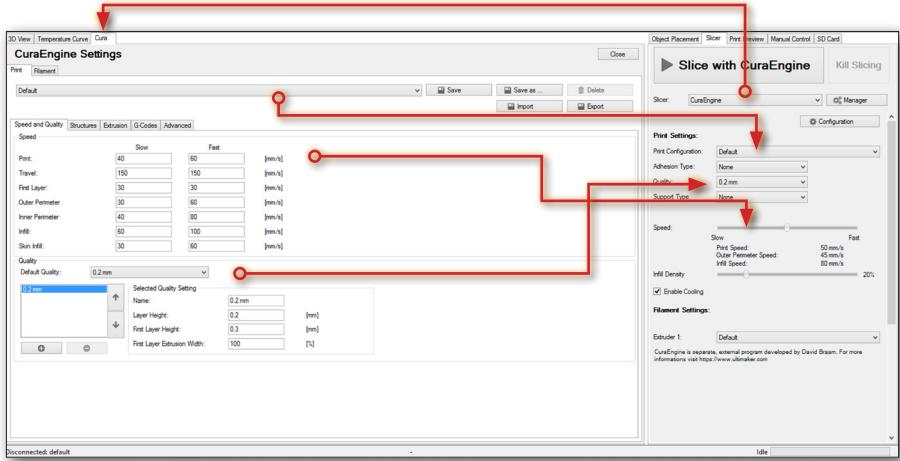
2. Configure the slice tool

Here we select CuraEngine as our slice tool.



Before we use the CuraEngine, we need to configure it first.





Print

A. Speed and Quality

Speed

- **Print**

Here set the print speed range, the “Low” is minimum speed, the “Fast” is Maximum speed.

- **Travel**

Here set a non-printing move speed.

- **First Layer**

This determines how fast the printer will extrude during the first layer. Normally 50% of printing speed.

- **Outer/Inner Perimeters and Infill**

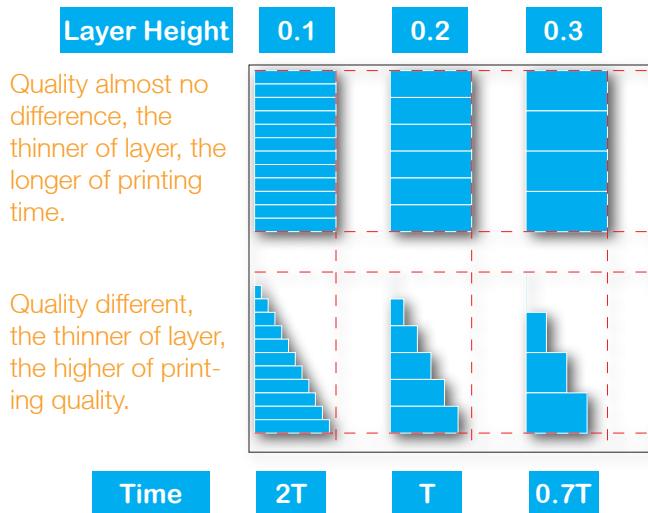
Infill speed can be higher than outer perimeter speed. Inner perimeter speed is better between of them.

- **Shin Infill**

No more than the Print speed.

A. Speed and Quality

Quality



- Name

Such as **Best, Standard and Fast**; or **0.1mm, 0.2mm and 0.3 mm**.

- Layer Height

$0.1\text{mm}, 0.2\text{mm}, 0.3\text{mm}$

The minimum height can be 0.05mm , but print time will be much longer. The maximum height can be as big as the size of nozzle's diameter i.e. 0.4 mm . It is fast, but low quality.

The maximum height normally is about 80% of nozzle's diameter. In case you print very big or small object, you can change nozzle size and adjust the layer height accordingly.

- First Layer Height

$0.2\text{--}0.4\text{ mm}$

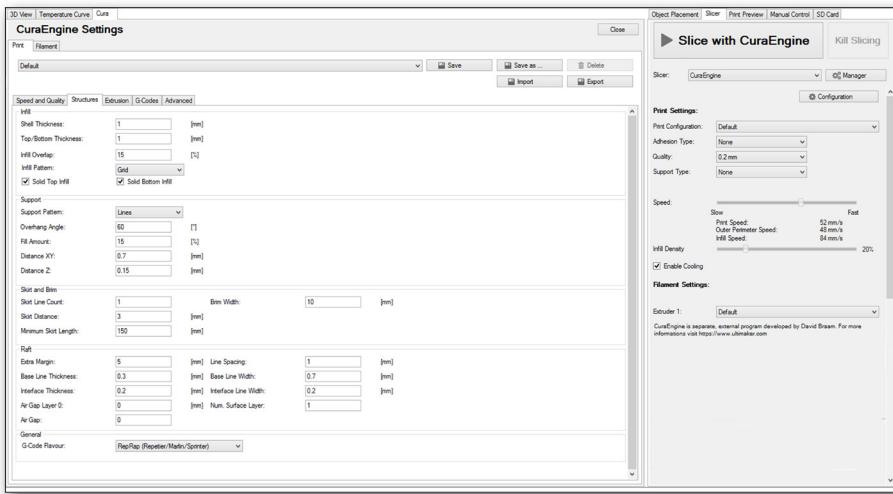
This height shall be 0.1 to 0.2 mm higher than the layer height

- First Layer Extrusion Width

100% Means the width is as big as the size of nozzle diameter.

Print

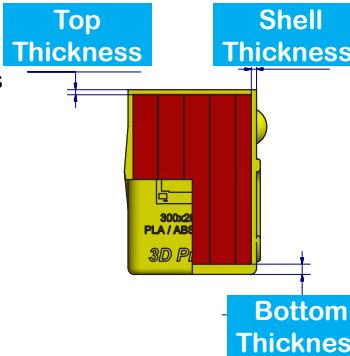
B. Structures



Infill

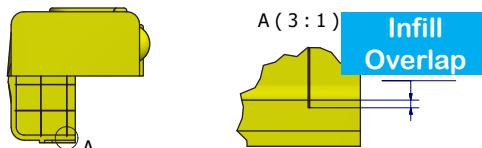
- **Shell Thickness**

Since the nozzle diameter is 0.4 mm, so thickness of 0.8 mm will run 2 rounds along perimeter; the thickness of 1.2 mm will run 3 rounds along perimeter.



- **Top/Bottom Thickness**

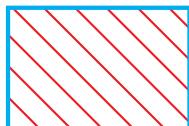
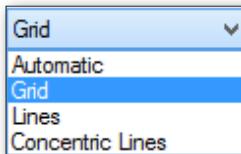
The Top/Bottom shell needs at least 3 layers to cover infill, so set the thickness according to the lay height.



- **Infill Overlap**

The setting determines how long the infill insert into the shell.

- Infill Pattern



Lines



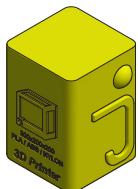
Grid



Concentric

We recommend **Grid** type, it makes object more stable.

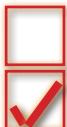
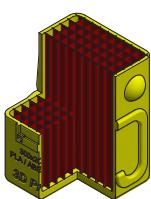
- Top/Bottom Infill



Solid Top Infill

Solid Bottom Infill

Both Top and Bottom are covered .

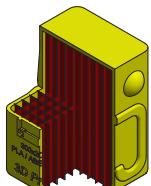


Solid Top Infill

Solid Bottom Infill

Bottom is covered,
Top is opened.

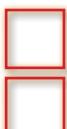
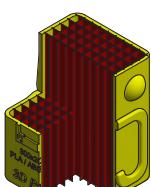
This is useful to print vase without infill.



Solid Top Infill

Solid Bottom Infill

Bottom is opened,
Top is closed.



Solid Top Infill

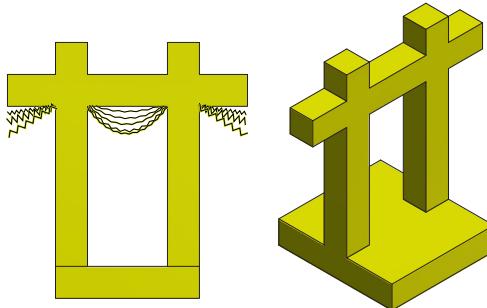
Solid Bottom Infill

Both Top and Bottom are opened .

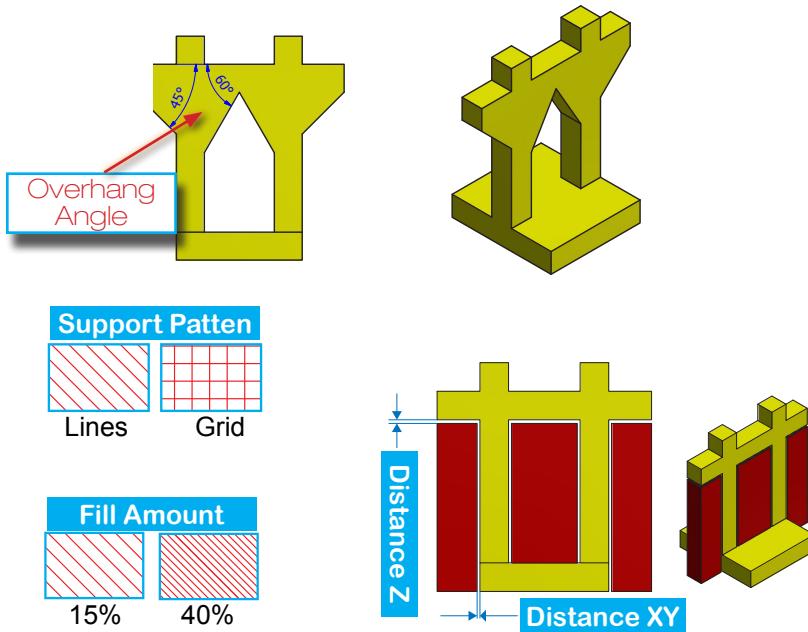
B. Structures

Support

Supports are used when models have steep overhangs or unsupported areas.



Normally, the printer can support the overhang when the overhang angle is over 45° because majority of the plastic layer is supported by the previous layer below it.



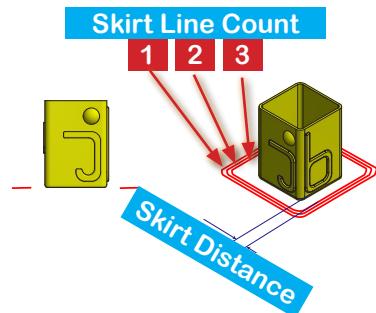
- **Support Pattern**
Lines or Grid
- **Overhang Angle**
More than 45°
- **Fill Amount**
15% is better
- **Distance XY**
0.7 mm
- **Distance Z**
0.15~0.2 mm

B. Structures

Skirt & Brim

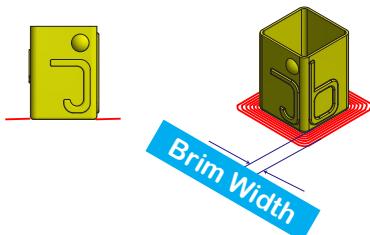
Skirt

To help prime your extruder and establish a smooth flow of filament, an outline surrounds your part is extruded before starting to print your model.



Brim

Brim connects your parts and extends outward to stabilize small parts or “islands”

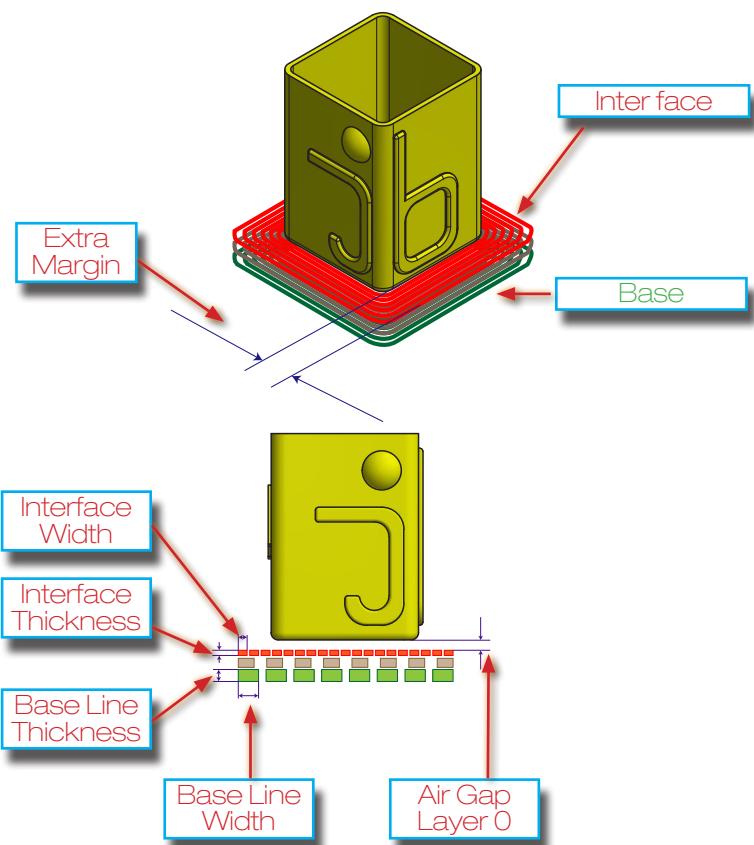


- **Skirt Line Count**
1 to 3
- **Skirt Distance**
3~5 mm
- **Min. Skirt Length**
150 mm
- **Brim With**
3~10 mm

B. Structures

Raft

Raft is layers under your parts. It helps to keep precision of the bottom of your parts after you remove it and cleanup.



- **Extra Margin**

2~10 mm

This determines the distance around the outside of your object that the raft is created. Can be helpful for ensuring no warping of the lower layers.

- **Line Spacing**

0.5~2 mm

A small spacing makes the support structures closer together improving strength of the raft, but uses more material.

- **Base Line Thickness**

0.3 mm

- **Base Line Width**

0.7 ~ 2 mm

- **Interface Thickness**

0.05~0.2 mm

- **Interface Line Width**

0.05~0.2 mm

- **Mum. of Interface Layer**

1~3

- **Air Gap 0**

0.15~0.2 mm

Larger distance is easy to remove the raft, but reduce quality of the parts at bottom.

- **Air Gap**

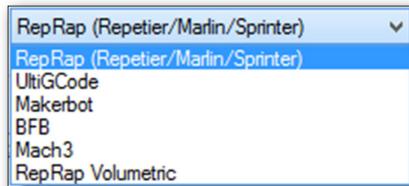
0 mm

B. Structure

General

G-Code Flavor :

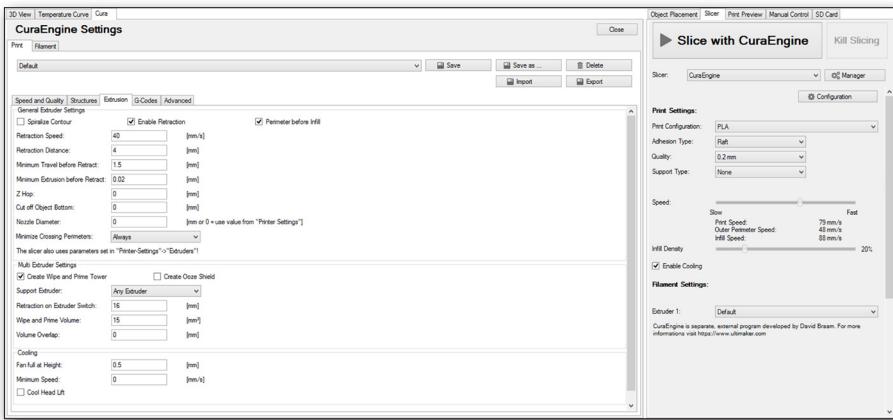
Reprap (Repetier/Marlin/Sprinter)



Print

C. Extrusion

General Extruder Setting

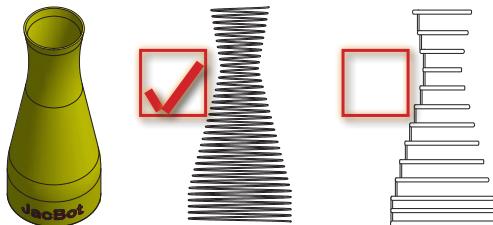


Spiralize Contour

The option is used for printing single-wall object, not matter if there is infill.

The option generates huge number of layers along the spiral path, no break from a layer to next layer.

If the object is solid, It prints outside contour; if the object is opened, it prints both sides contours.



Enable Retraction

This option shall be checked if Spiralize Contour option is not checked.



Perimeter before Infill

This option shall be checked if Spiralize Contour option is not checked.

- **Retraction Speed**
The speed of filament retract.
30~60 mm/s is better.
- **Retraction Distance**
The length of filament retract
0 means no retraction.
4~8 mm is good setting to avoid string hanged between shells of object.
- **Minimum Travel before Retract**
The minimum non-printing travel distance to active the retraction
- **Minimum Extrusion before Retract**
The minimum length of filament extruded to active the retraction.
- **Z Hop**
Increase the distance between nozzle and object printed during non-print traveling.
- **Cut off Object Bottom**
Sinks object into the platform
- **Nozzle Diameter**
Set 0 as the setting at '3. Extruder' on Page 26
- **Minimize Cross Perimeter**
Always
It may take long trip but less string.

Always
Never
Always
Always Except for Skin

Print

C. Extrusion

Cooling

- **Fan Full at Height**

The height at which the fan turns on at maximum speed.

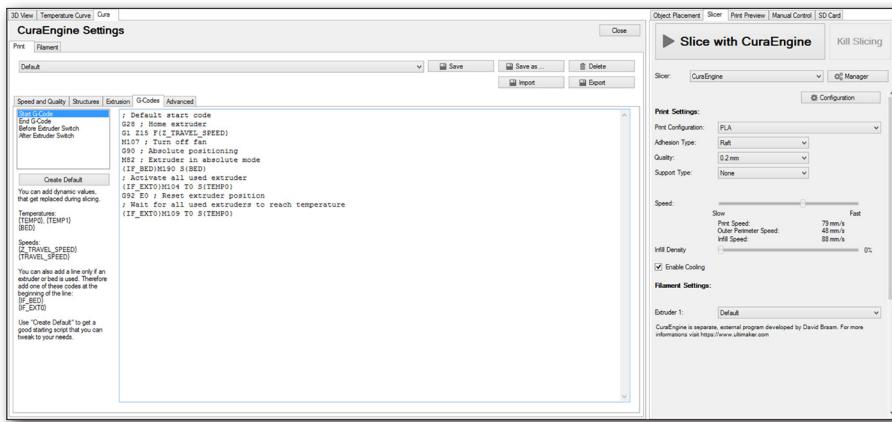
- **Minimum Speed**

It forces the fan to turn on at full speed upon the printing speed lower than the minimum speed setting.

Print

D. G-Code

G-Code is the generic name for a control language for 3D printer. It indicates the machine to move to various points at a desired speed, control the temperature, turn on and off fan, and all sorts of other things. It is fairly standard, and is a useful tool. See Appendix 'G-Code' on Page 48



Start Code

The Start Code is applied to make everything ready for printing.

The default code is very simple, we shall make modification to meet our demands.

To realize the auto bed leveling, G29 shall be added follow the line of G28.

To adjust the distance between the nozzle and work plate, G92 Znn shall be added at the end of the start code. Here nn is a number such as 0.1, 0.2, ... -0.1, -0.3, ...

See appendix '4. Start G-Code' on Page 49

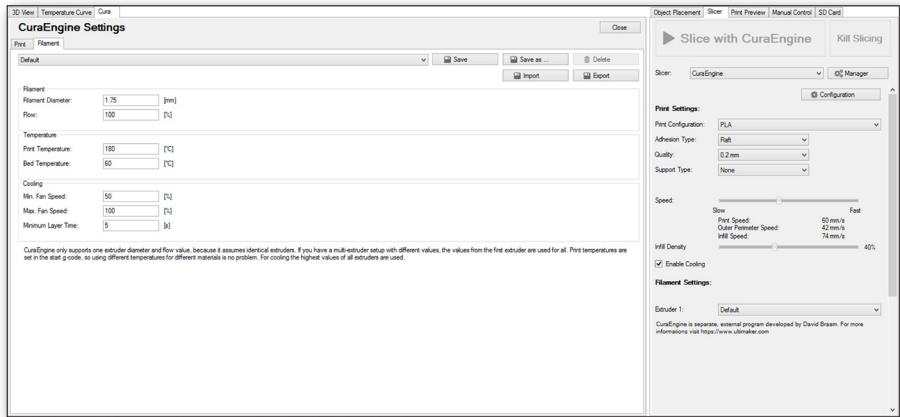
End Code

The End Code is applied to restore the printer to original state.

See appendix '5. End G-Code' on Page 54

Filament

Filament



- **Filament Diameter**

1.75 mm

- **Flow**

100%

If extrusion is too much, you can reduce the amount, if too less increase the amount. But you shall combine this with the temperature and speed in consideration.

Cooling

- **Min. Fan Speed**

50 %

- **Max. Fan Speed**

100%

- **Minimum Layer Time**

5 seconds

It force print speed to slow down when print very small object.

Temperature

- **Print Temperature**

PLA: 180~230 °C

ABS: 220~260 °C

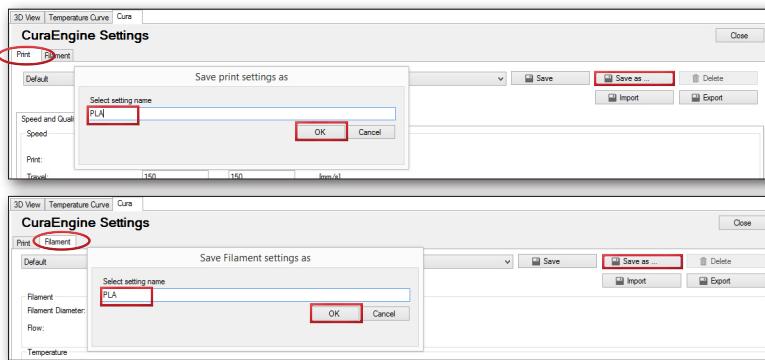
- **Bed Temperature**

PLA: 60 °C

ABS: 110 °C

3. Make New Configuration

To make new configuration, click **Save as ...** button,



Give a name, then Click **OK** button.

Edit the default setting, when finished, click **Save** button.

Note: Any change will be not affective until you save it.

4. Import & Export the Configuration files

Sometime you want to use the configuration in different computers, you can export one or all your configuration to a flash disk then import from other computers.

