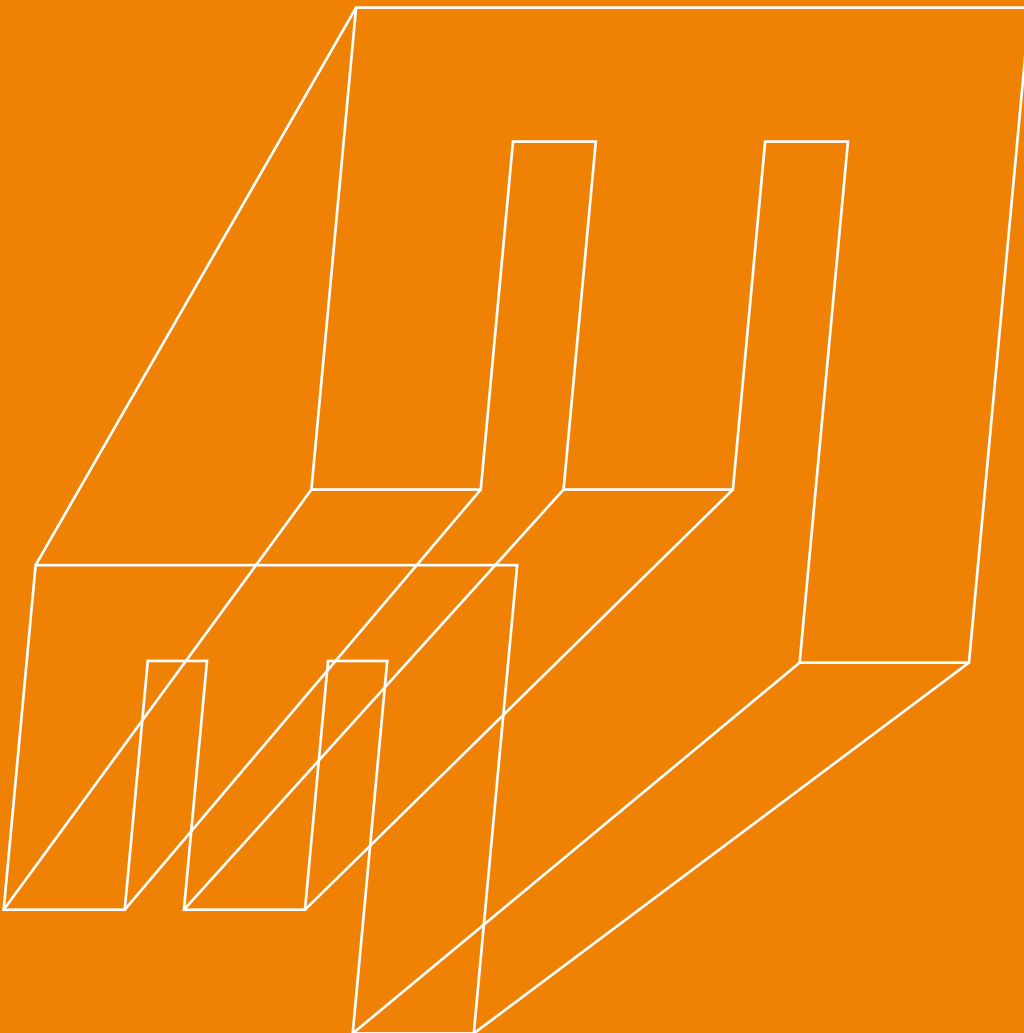




MPrint User Manual



Contents

MPrint Installation	01
Downloading	02
Setup	02
MPrint Interface	03
Using MPrint	05
Select Device	06
Select Slicer	06
Import Model	07
View Orientation	07
Move Model	08
Scale Model	08
Rotate Model	09
Set Printing parameter	09
Export	11
Multiple Model Slicing	13
Slicer setup	15
MBotslicer Parameter Setting	16
Slic3r Parameter Setting	20
Rectify printing size	29
Tips	31

01. MPrint Installation

MPrint Installation

Download

Open the below link to download the installation package

<http://www.mbot3d.com/software>

Supporting operating system: Windows Vista/Win7 or higher, Mac OS X.

Remarks:

1. This user manual is for MPrint 1.3.0 Version Window OS
2. If the anti-virus software prevents to install MPrint then chooses trust MPrint and continue. If it is necessary, then disable the anti-virus software temporarily during the installation.

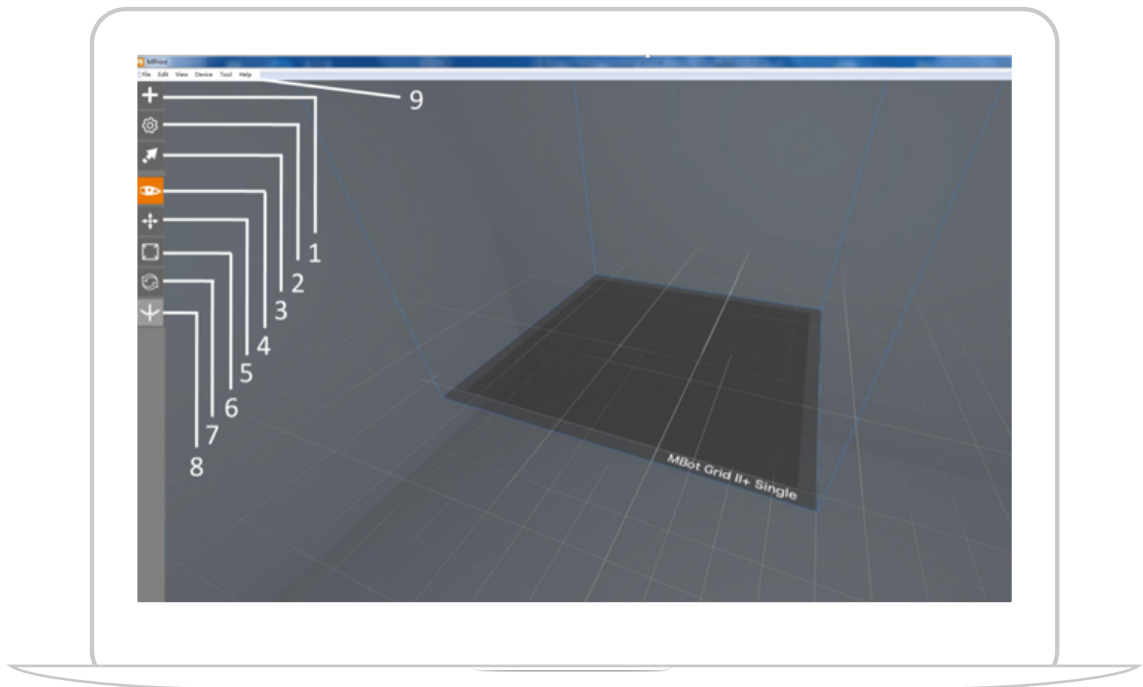
Setup

1. Double click the MPrintinstall.exe file to install the MPrint.
2. Follow the instructions. Installing in the default directory is suggested. After finish, MPrint icon is created on the desktop.
3. Start MPrint from the desktop icon or your applications folder.

 **Note: MPrint cannot detect and connect automatically on window 8 and 10 but need to connect manually.**

02. MPrint Interface

MPrint Interface



1--Import stl file

4--View orientation

7--Rotate tool

2--Parameter setting

5--Move tool

8--Dual head setting

3--Export x3g file

6--Scale tool

9--Menu bar

03. Using MPrint

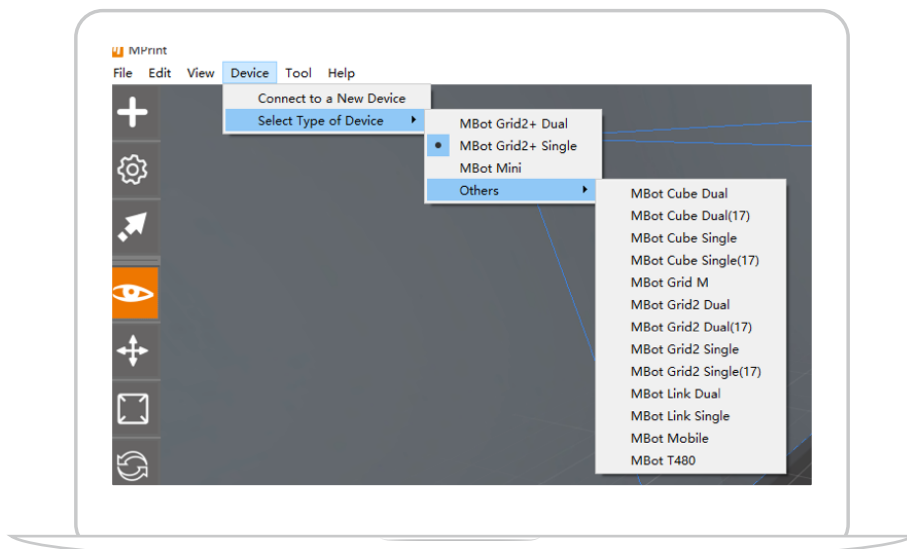
Using MPrint

5 steps to generate a printable x3g file from the stl file:

1. Select Device and slicer
2. Import model
3. Adjust model (Move, Scale and Rotate the model)
4. Slice and export file
5. Print

Select Device

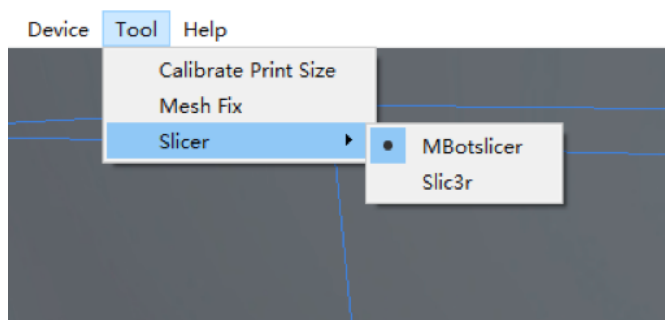
Select correct MBot 3D printer model on Device. Otherwise, as default device is MBot Grid 2+ Single Head.



Select Slicer

MPrint has integrated two slicers: MBotslicer and Slic3r. Choose one of them.
Tool => Slicer => MBotslicer/Slic3r.

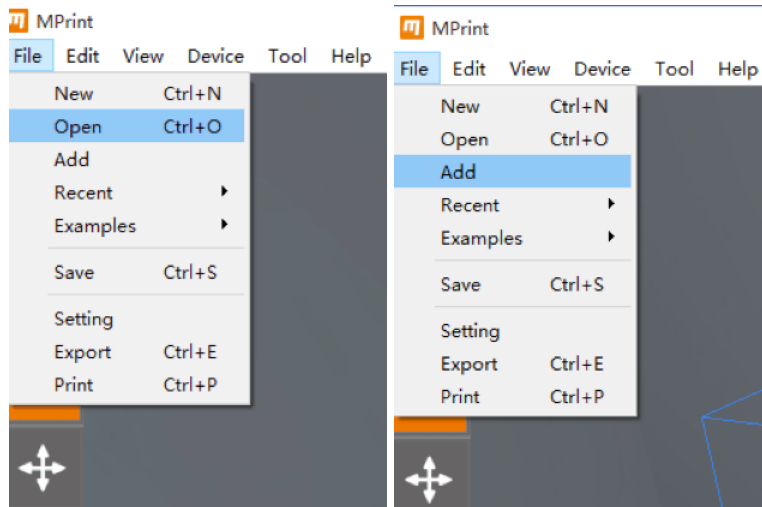
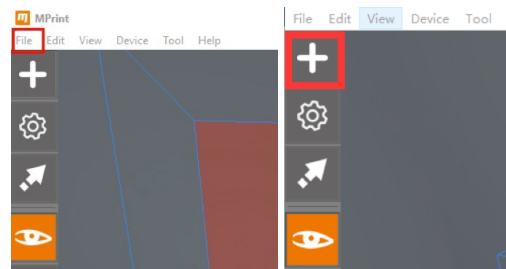
⚠ Note: Slic3r slicer only supports Single Head



Import Model

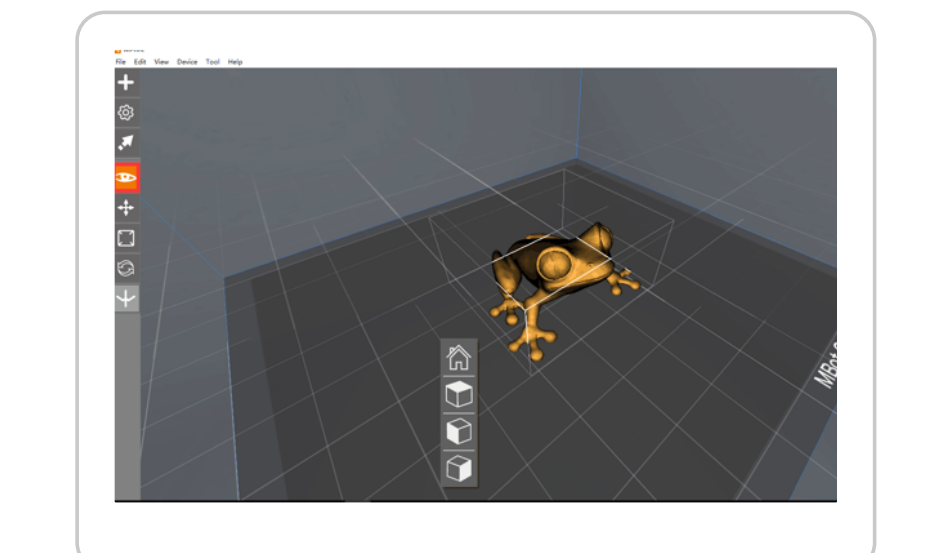
- Menu: File => open => select model => open
- Left Menu: Add => Select model => open

! Note: Model already exists in the window and if the new model is open then it will replace previous model.



View Orientation

Select the View (fourth shortcut key) on the left menu. Hold the left mouse button and move to view the model in different angles.

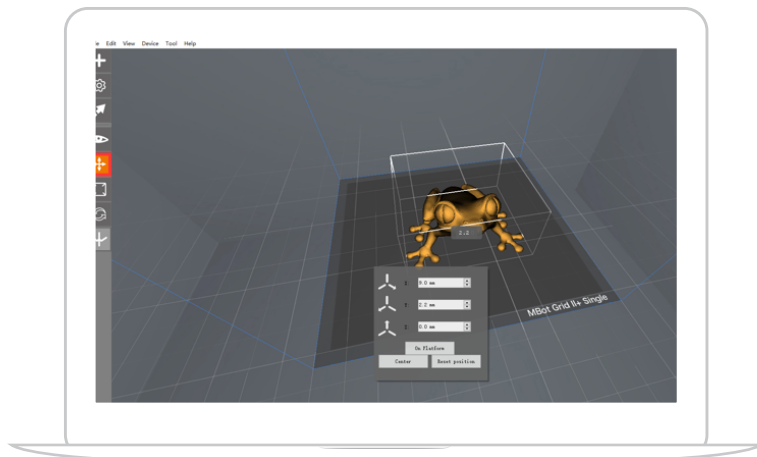


Move Model

Select the Move (fifth shortcut key) on the left menu. Click the right mouse button and X, Y, Z position window will pop-up. Change the value of X, Y, Z position according to as a requirement.

Hold the Right mouse button and move the mouse to view the model in different angles.

! Note: Make sure the model is on the platform, and if there is a single model. It is suggested to place at the center of the platform.

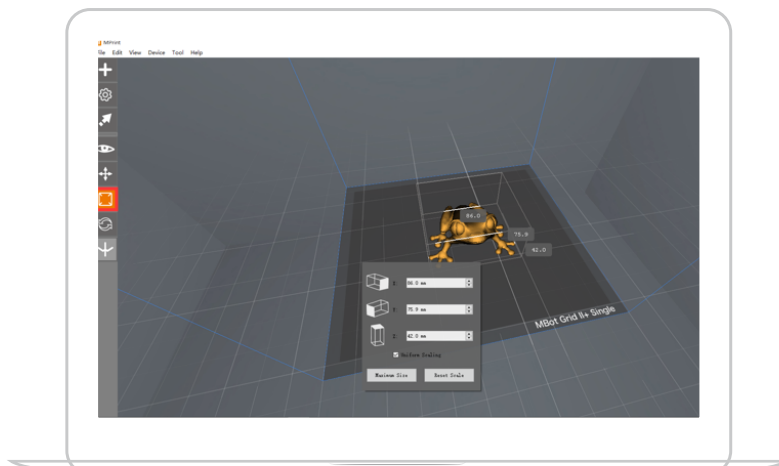


Scale Model

Select the Scale (sixth shortcut key) in the left menu and click left mouse button to select the model then size will show. Scroll up to increase and scroll down to decrease the size of the model. Click the right mouse button and X, Y, Z scale window will pop-up. Change the value of X, Y, Z scale according to as a requirement.

Uniform scale is selected as default. Don' t select the uniform scale when you want to change a single value in X, Y, or Z axis. Maximum size is the largest uniform scale the printer can hold.

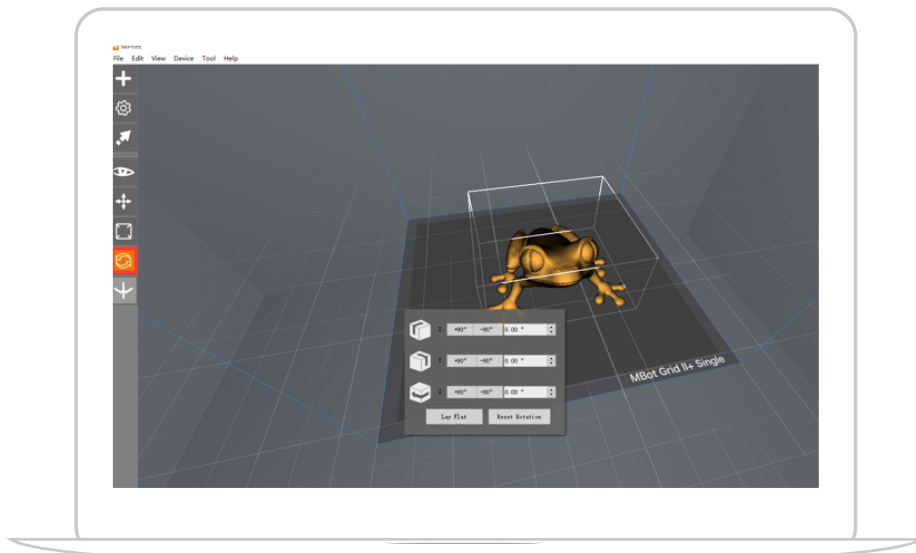
Hold the Right mouse button and move the mouse to view the model in different angles.



Rotate Model

Select the rotate tool (seventh shortcut key) on the left menu. Point the mouse cursor on the model and click the right mouse button and X, Y, Z rotation angle window will pop-up. Change the value of X, Y, Z rotation angle according to as a requirement.

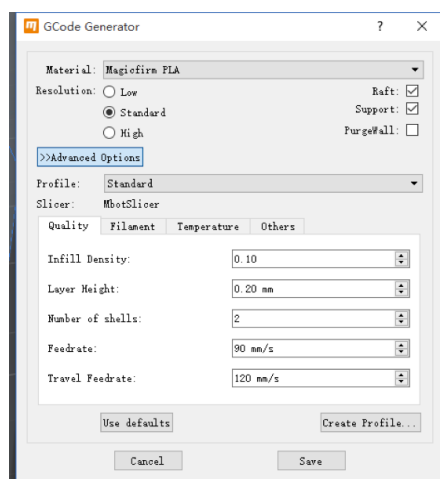
Hold the Right mouse button and move the mouse to view the model in different angles.



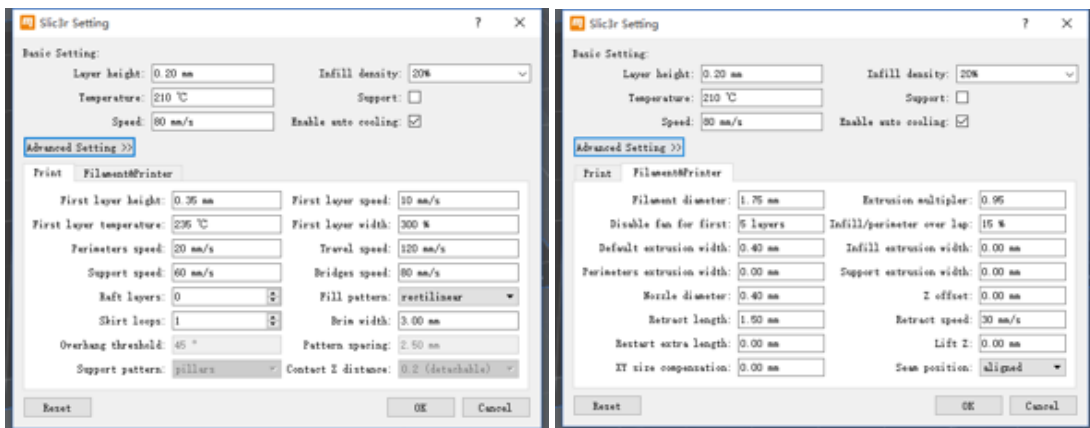
Set Printing parameter

Common printing parameters

Click on setting (the second shortcut key) on the left menu, and the printing parameter setting window will open. Here you can set the filament type, layer resolution, raft, support, purge wall. Temperature, density, speed, layer height can be set up on the advanced setting. (The parameters' detail functions are explained in the Slicer setup)



MBotslicer slicer

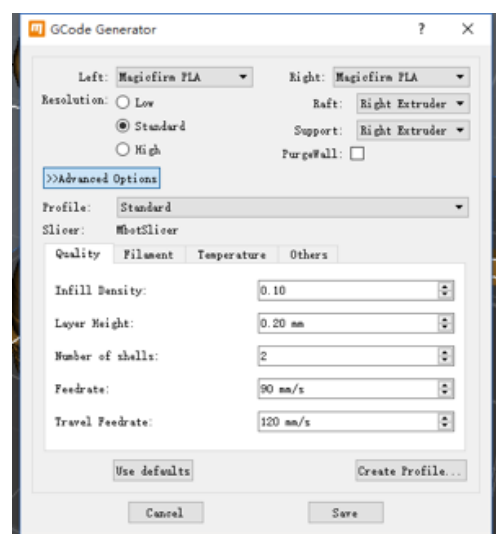
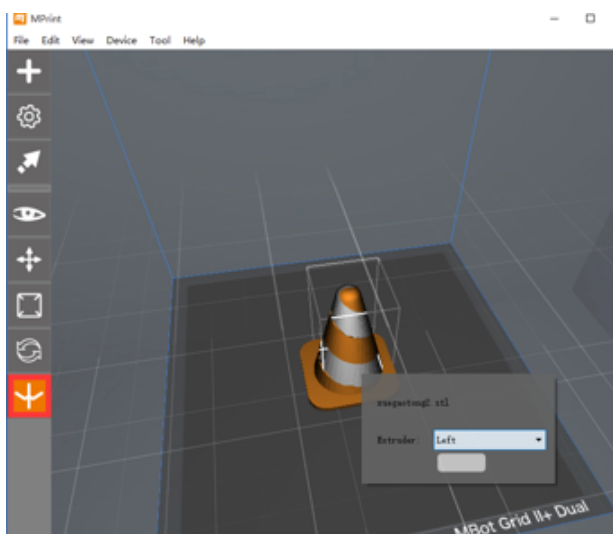


Slic3r Slicer

Double Head setup

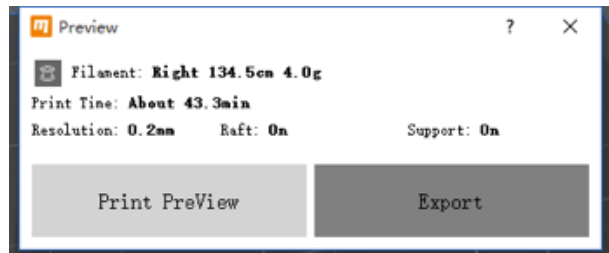
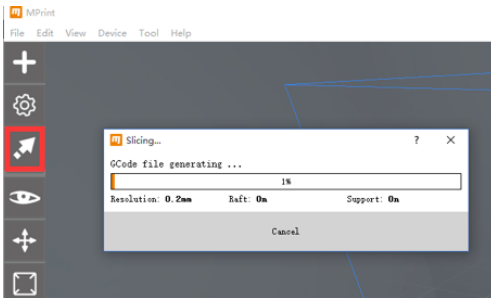
Click eighth (the last shortcut key) in the left menu. Identify which part will print by which printer head in the pop-up window.

! Note: Firstly, make sure you have selected double head machine in the device, and chosen slicer as MBotSlicer in the tool.

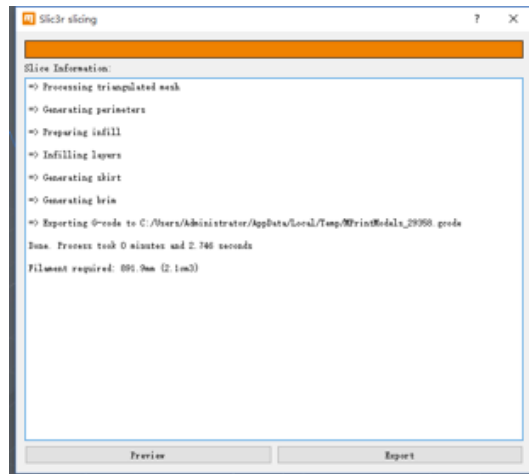


Export Model

After all the parameters are setup, click Export (the third shortcut key) on the left menu to generate the x3g file. Once the slicing process is finished, Preview window will pop-up. There are two options Print preview and Export.



MBotslicer



Slic3r

MBotslicer and slic3r both will show approximate the filament consumption and printing time.

Click Preview, to preview the print model layer by layer and examine the parameters before print.

- Print From SD Card (recommend)

Copy exported x3g file into SD card and print from SD card.

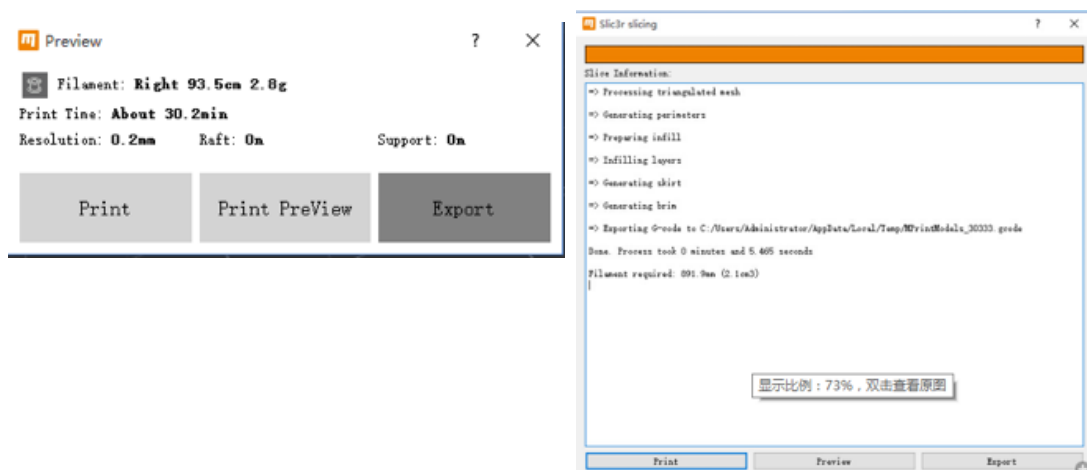
● Print From computer

MBot Grid II+ and MBot T480 are automatically detected when connecting with the computer by MPrint but another model has to manually select.

Selecting for another model MBot 3D printer. On Menu: Device => select type of device (select correct model) and Device => connect to a new device => COM

After the device is successfully connected, the machine type will be displayed at the bottom right corner of the window.

Once the device is connected, there will be a print choice in the export message, so that you can print directly from the computer.



Printing process is shown at the bottom right corner. It can be paused or cancelled from the computer.



04. Multiple model slicing

Multiple model slicing

copy model

If you want to print a batch of the same model, copy or Ctrl + D

Delete Model

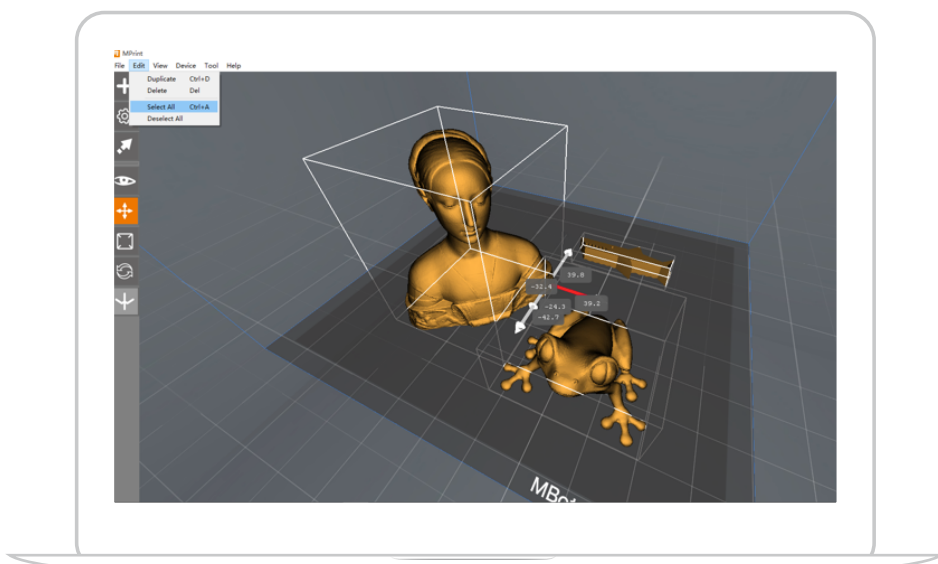
Delete or Backspace

Save Model

Save or Ctrl + S

Synchronous slicing multiple models

Select all the models (on menu: Edit => Select all or Ctrl + A) or press Ctrl and select each model. any operations will be done on all the selected models.



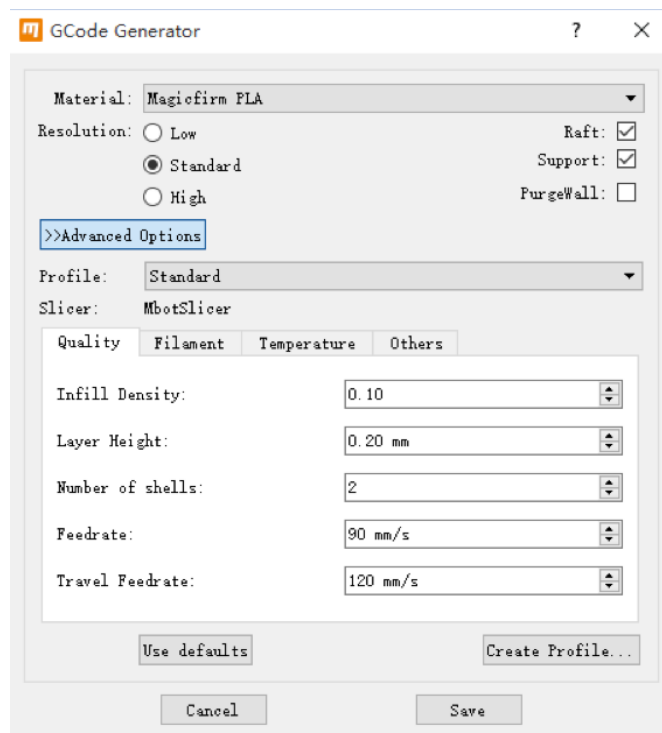
05. Slicer Setup

Slicer Setup

MBotslicer slicing parameter setting

Choose MBotslicer (Menu: Tool => slicer => MBotslicer)

Click setting (the second shortcut key) on the left menu or from Menu file => setting.
The parameter setting window will pop-up.



Material

Choose printing material type.

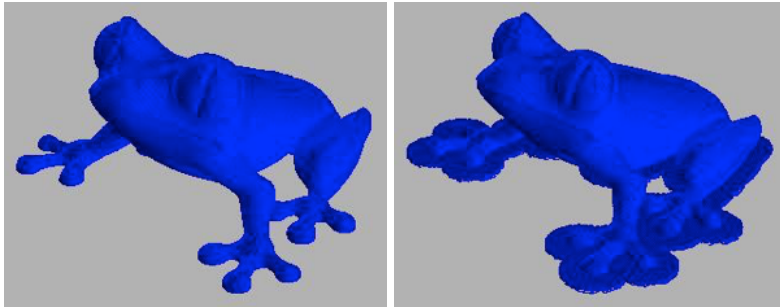
Resolution

Choose Low, Standard or High resolution to specify the surface quality of your 3D print.

- Objects sliced with the standard resolution profile will be printed using the default settings standard resolution prints will print quickly and have good surface quality.
- Objects sliced using the low resolution profile will be printed with thicker layers and will print faster.
- Objects sliced with the High resolution profile will have finer layers and will print more slowly.

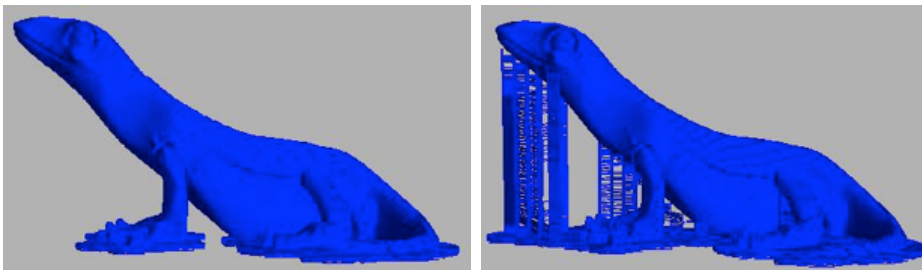
● Raft

The Raft act as a base for your object and any support structure, and ensures that everything adheres well to the build plate. The raft will be easily removable once you remove your finished object from the build plate.



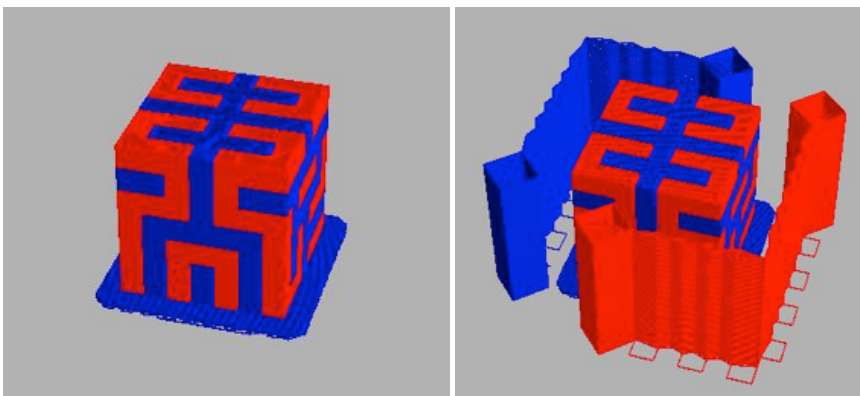
● Support

MPrint will automatically generate supports for any overhanging sections of your object. Support will be easily removable once you remove your finished object from the build plate.



● Purge Wall

Purge wall is most useful for the print by dual head machines, and models with the small cross section. It can prolong the cooling time of the print layer so that the layers are well shaped.

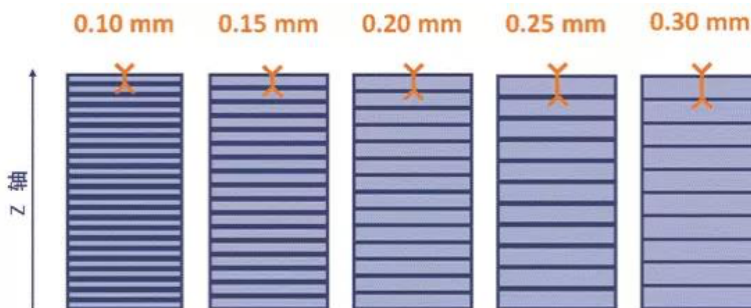


■ Infill Density

Density is the percentage of inner fill. The infill can support and shape the model. The higher density creates the stronger model. At the same time, the more filament and are consumed. Select the density according to necessary.

■ Layer Height

Layer Height is the height of each layer. The lower value of layer is the finer surface of the model. It is usually a benchmark of the print resolution.



■ Number of shells

A number of shells means is a value that sets the number of outlines printed on each layer of your object. The default value is 2.

■ Feed Rate

Feed-rate is the speed of printing head when extruding filament. Usually the value should be 40 to 100mm/s, and the slower the printing head moves, the better the print is.

■ Travel Feed-rate

Travel feed-rate is the speed of printing head when not extruding, that is when the printing head moving to the next print part. The usual value is 90 to 150mm/s. This speed should be faster than the feed-rate. It saves time and reduces the excess melt filament dropping onto the model surface where there shouldn't be any extrusion.

■ Filament Diameter

MBot 3D printer' s usage 1.75mm

- Cooling Fan

The front cooling fan cooling the melted filament extruded from the nozzle. It can be disabled when printing high shrinkage filament like ABS.

- Extruder temperature

The temperature should be set up according to filament manufacture recommendation. For example, PLA usually is printed at 200 to 230°C, ABS: 220~250°C. MBot 3D printers can print as high as 260°C, almost covering all the materials in the market.

- Intelligent Compensation

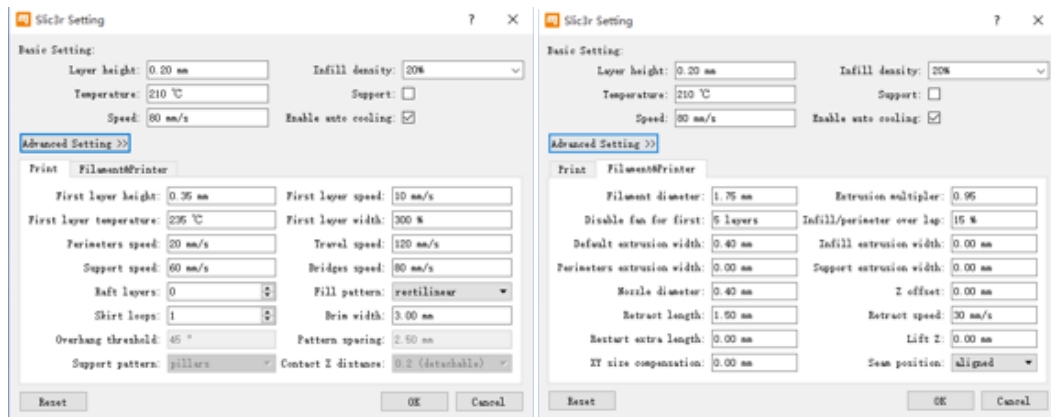
Intelligent compensation is only available for the MBot Grid II+ Single Head machines with firmware 7.7.1. With Intelligent compensation function on, the printing head can detect three probing positions on build plate which automatically control the leveling accurately and reduce warping by adjust its height and tilt angle to fit the printing head. This function can reduce the problems caused by the uneven plate, such as rough raft warping, and scratch between the printing head and printing plate.

- Pause at ZPos

Pause at Zpos is to set the printer to stop at a specific Z-axis height. Multiple pause points should be separated with a comma.

Slic3r slicing Parameter Setting

Slic3r is similar to MBotslicer, including basic setting and advance setting is shown as below. Please refer to the operation of MBotslicer.



First layer setting

The First layer is the base of a model, so it is significant to print well. Here are some setting tips:

- First layer height:

A thicker layer height will provide more flow, and consequently more heat, making the extrusion adhere to the build plate. It also gives the benefit of giving more tolerance for the levelness of the build plate. It is recommended to raise the first layer height to match the diameter of the nozzle, e.g. a first layer height of 0.35mm for a 0.35mm nozzle.

- First Layer Speed

The speed should be low 10 to 30mm/s is recommended.

- First Layer Temperature

To enhance the attachment onto the printing plate, the temperature of the first layer should be 10~20°C higher than the normal printing temperature.

- First Layer Extrusion Width

The wider the first layer adheres strongly to the plate. For example, If the first layer height is 0.3mm, the first layer extrusion width is 300% and the final first layer extrusion width on the plate is 0.9mm.

Speed

Print Speed is the infilling speed. If rigidity is not required, the speed can be as fast as 100mm/s. The recommended range is 60 to 100mm/s.

■ Perimeters Speed

The outline of the model may benefit from being printed slightly slower so that the outside skin of the print has fewer blemishes. The speed should be as low as 20 to 40mm/s.

■ Infill Speed

As the infill is hidden this can be extruded a little faster. Take care though not to go too fast as higher speeds results in thinner extrusions, and this may affect how the extrusions bond

■ Travel speed

The jump between the end of one extrusion and the next should usually be performed as quickly as the printer will allow in order to minimize any mess caused by material oozing from the nozzle.

■ Support speed.

Support speed is the speed of printing supports. Since high resolution is not necessary, print can be faster than the normal print. Recommend range is 80 to 100mm/s

■ Bridges Speed.

Bridge speed is subject to the cooling effect and filament curdling feature. Too low will make the bridge loose and too fast may not shape a line. Recommend speed is 60 to 100mm/s.

- Raft Layers.

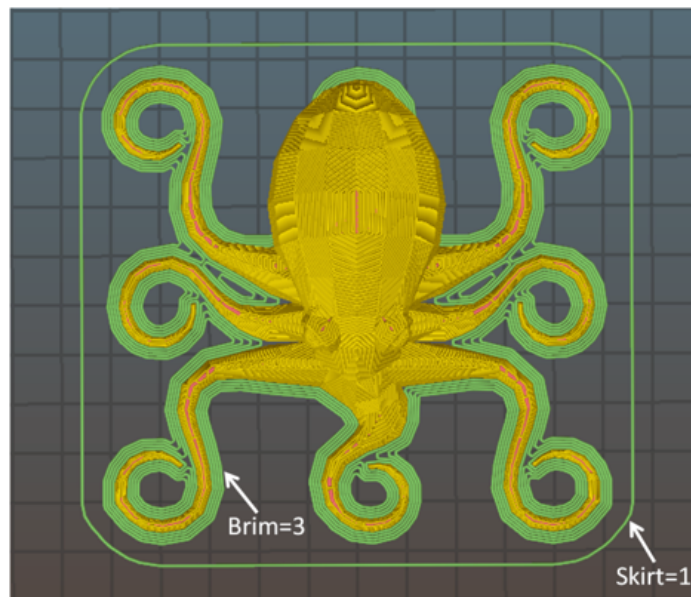
The raft can help the print adhere to the build plate better especially when there is not heating bed or too rough to leave marks. The raft is automatically generated when print supports. But raft needs to be removed and sometimes it can be annoying. So we suggest value:0.

- Skirt Loops.

Skirt loops is a line of filament extruded before the real print begins. It is to check that the nozzle is not clogged. Recommend value: 1. Please refer to the below pic.

- Brim Width.

Brim width is used to add more perimeters to the first layer, as a base flange, in order to provide more surface area for the print to stick to the bed with in order to reduce warping (see picture below). The brim is then cut away once the print is finished and removed from the bed. Suggest value: 3 to 5.



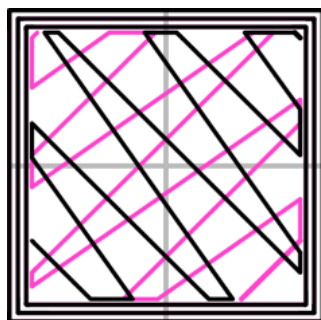
Infill Setting

There are several considerations when choosing an infill pattern: object strength, time and material, personal preference. It can be inferred that a more complex pattern will require more moves, and hence take more time and material.

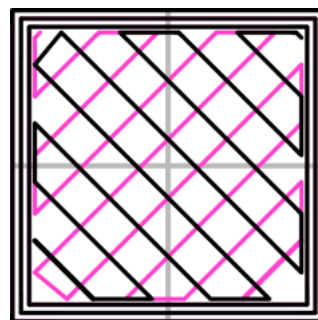
■ Fill pattern

Slic3r offers several infill patterns, four regular, and three more exotic flavors. The numbers given in brackets below each figure are a rough estimate of material used and time taken for a simple 20mm cube model.

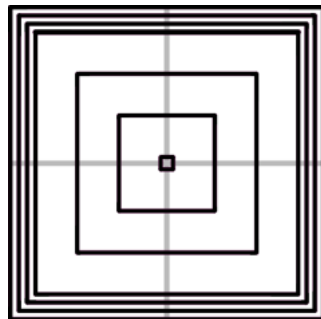
Note that this is only indicative, as model complexity and other factors will affect time and material.



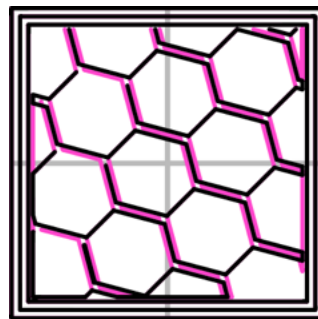
Line (344.51mm / 5m:20s)



Rectilinear (350.57mm / 5m:23s)



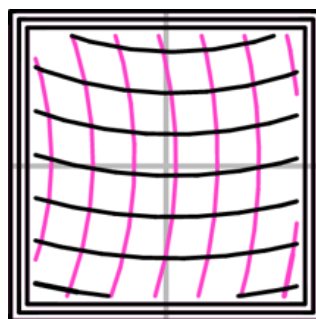
Concentric (351.80mm / 5m:30s)



Honeycomb (362.73mm / 5m:39s)



Hilbert Curve
(332.82mm / 5m:28s)



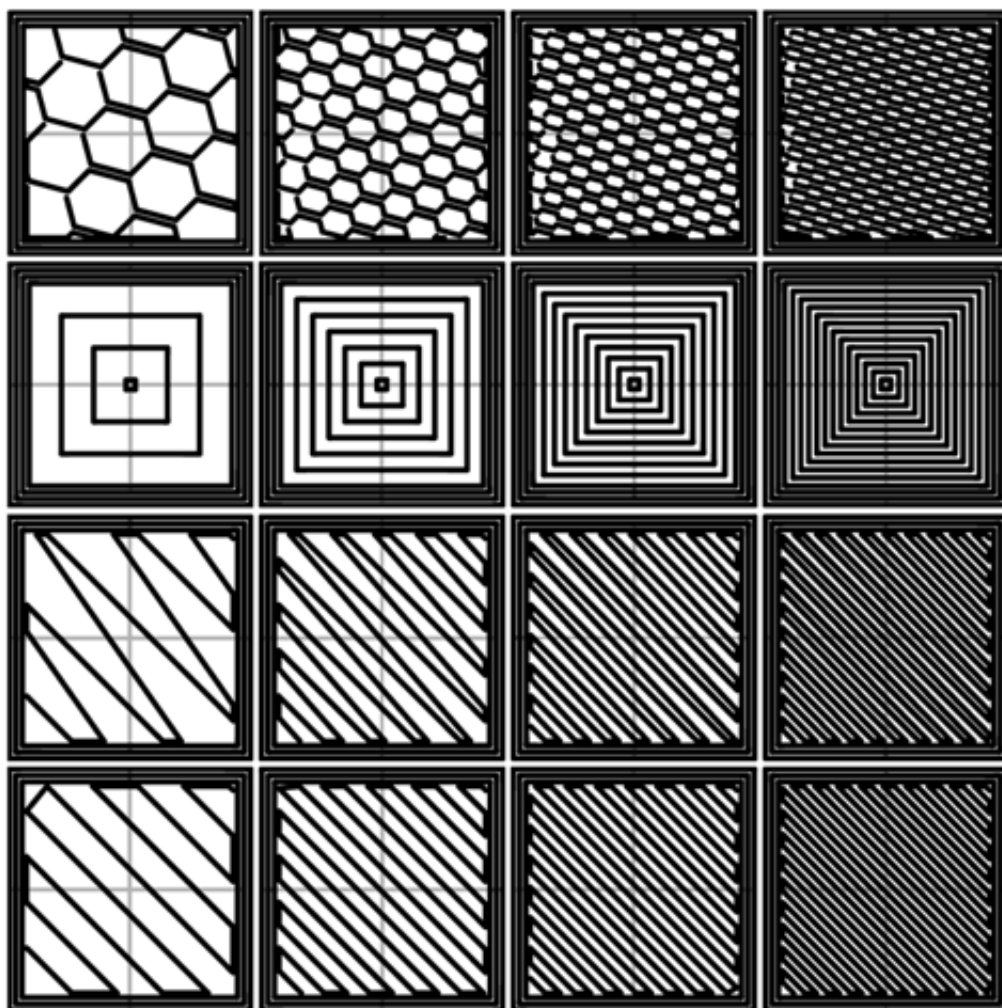
Archimedean Chords
(333.66mm / 5m:27s)



Octagram Spiral
(318.63mm / 5m:15s)

Infill Setting

Density can affect the model's strength, so if you need a strong model, the density should be high. otherwise, low density is recommended. We recommend 10% to 30% for a normal showcase model. Below is the infill effect pic of Honeycomb, Concentric, Line, Rectilinear, Hilbert Curve with a density of 20%, 40%, 60%, and 80%.



support setting

Support is to hold the overhang parts. It is generated when the overhang slopes at an angle or higher. The default value of this angle is 45°.

Since support costs time and filament, and it leaves marks on the model surface, we should avoid as less support as possible.

 **Note: All the following settings are only available on advanced setting**

■ Overhang threshold

Overhang threshold is the angle an overhang slopes from the horizontal surface. If the model is complicated, it's hard to set one angle for all situations. We suggest you to set the overhang threshold as 0, and then the software will decide itself where to add a support.

■ Pattern spacing

Pattern spacing is the space between the support pillars, that is the density of the supports. Suggest value is 2.5mm. If too low, it will be weak.

■ Contact Z distance

Contact Z distance is the distance between the model surface and the support surface. There are two options: 0 (soluble) mm, and 0.2 (detachable) mm. The former choice is for the water soluble support. The later one is for other materials.

Extrusion Setting

Support is to hold the overhang parts. It is generated when the overhang slopes at an angle or higher. The default value of this angle is 45°.

- Default extrusion width

If set to 0, the software will calculate the value, which is recommended.

- Infill extrusion width

A thicker extrusion for infill will produce faster prints and stronger parts. When set as 0, the infill extrusion width is taken as the default value.

- Perimeters extrusion width

A lower value will produce thinner extrusions which in turn will produce more accurate surfaces. When set as 0, the infill extrusion width is taken as the default value.

- Support extrusion width

As with the infill options, a thicker extrusion will speed up print time. When set as 0, the infill extrusion width is taken as the default value.

- Infill/perimeters overlap

Infill/perimeter overlap is the coincident part of infill and perimeter. The recommended value is 15%, that is the overlap width is 15% of the perimeter extrusion width.

Retract Setting

Once the filament is melt, it spills out easily from the nozzle, thus the surplus filament will be buttered on the surface, or spins around. Some parameters can be adjusted in Slic3r to relieve this problem.

- Retract length

The number of millimeters to retract. Note that the measurement is taken from the raw filament entering the extruder. A value of between 1 and 2mm is usually recommended. Bowden extruders may need up to 4 or 5mm due to the hysteresis introduced by the tube.

- Retract speed

The speed at which the extruder motor will pull back the filament. The value should be set to as quick as the extruder can handle without skipping steps, and it is worth experimenting with this value to find the quickest retraction possible.

- Restart extra length

Adds an extra length of filament after the retraction is compensated after the travel move. This setting is rarely used, however should the print show signs of not having enough material after travel moves then it may be useful to add a small amount of additional material.

- Lift Z

Raises the entire extruder on the Z axis by that many millimeters during each travel. This can be useful to ensure the nozzle will not catch on any already laid filament, however it is usually not necessary and will slow the print speed. A value of 0.1mm is usually sufficient.

Filament Setup

- Filament diameter

Default value: 1.75mm, cannot be changed.

- Extrusion multiplier

The Extrusion multiplier setting allows the fine tuning of the extrusion flow rate, and is given as a factor, e.g. 1 means 100%, 1.5 would mean 150%. Whilst the value should ideally be set in the firmware it can be useful to test slight changes to the rate by altering this value. It varies the amount of plastic proportionally and should be changed in very small steps (e.g. +/- 0.05) as the effects are very visible.

- Temperature

Printing temp should be adjusted according to the filament type. The recommended temp for PLA is 190 to 220°C. 230 to 250°C for ABS.

⚠ Note: The first layer temp should be 5 to 15°C higher than its printing temp.

- Enable auto cooling

Cooling fan can help to cool the extruded layer instantly and shape a good base for the next layer, thus the surface of the model won't be twisted. Especially, It's effective when printing models with cantilever, bridge, or segment of small cross section. But for ABS and other high shrinkage material, it is not necessary to enable the cooling fan since it may cause too much shrink and twist the model surface.

- Disable fan for first layers

It is very important the first layer well adhesion on the build plate. So it is suggested to disable the cooling fan when printing the first three layers, leaving enough time for the filament adhere to the build plate.

Printer setup

There are several considerations when choosing an infill pattern: object strength, time and material, personal preference. It can be inferred that a more complex pattern will require more moves, and hence take more time and material.

- Nozzle diameter

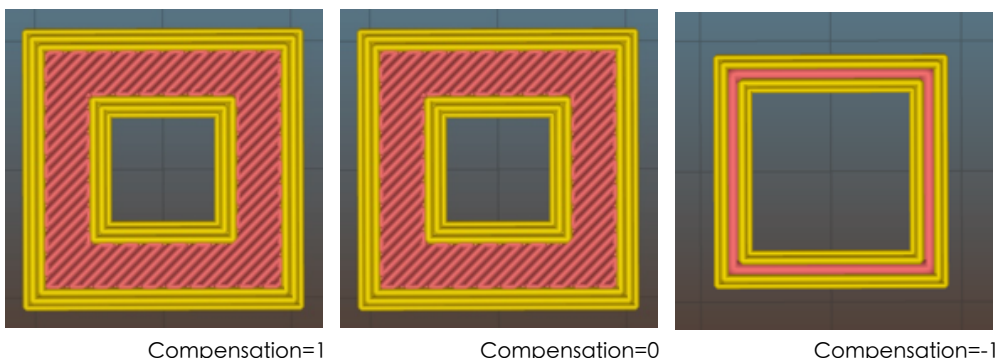
Default value: 0.4mm. Cannot be changed.

- Z offset

Z offset can be used to compensate for an incorrectly calibrated Z end-stop. If the nozzle stops slightly too far from the bed, then adding a negative value will offset all layers by that amount. The correct solution however is to fix the end-stop itself.

- XY size compensation

XY Size Compensation option was also introduced that allow to grow/shrink object shape in order to compensate for the measured error. Supposing your holes are smaller by 0.1mm, you can just enter -0.05 in this option to get them compensated (negative sign means shrink inwards).



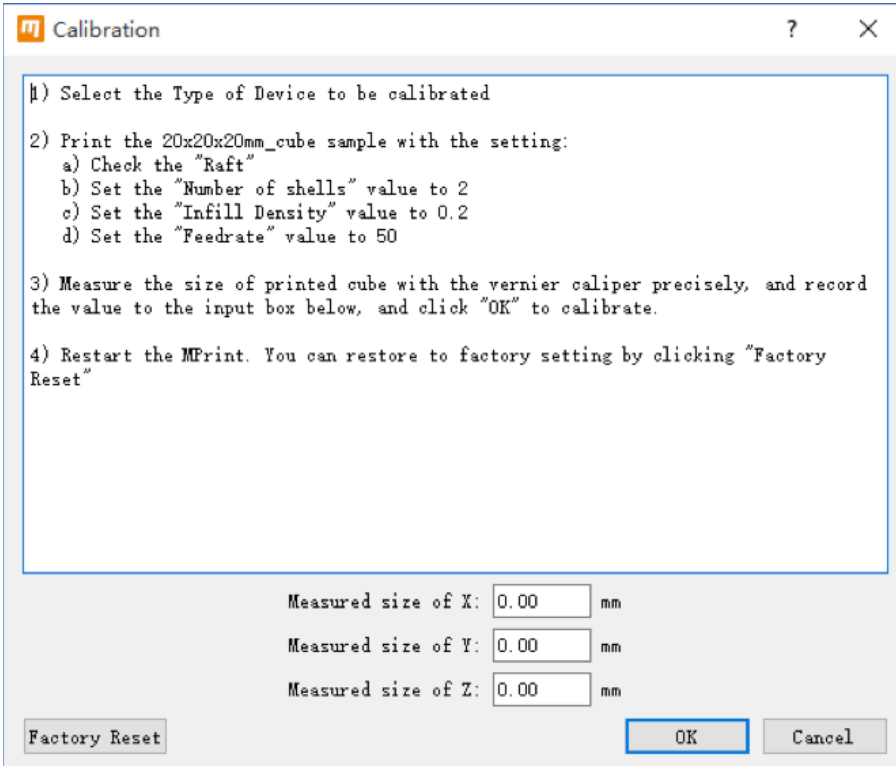
- Seam position

(Layers and perimeters - Vertical shells) - This setting determines the start point of perimeter loops, and thus the position of the potentially visible vertical seam on the side of the object. The available options are: Aligned、Random、Nearest。

06. Rectify printing size

Rectify printing size

Choose Rectify from Tool, and you will see a pop-up window shown as below, just follow the instructions.



The screenshot shows a window titled "Calibration" with a question mark icon and a close button. The window contains a list of instructions for calibrating the printer. Below the instructions are three input fields for measured sizes of X, Y, and Z, each with a "mm" unit. At the bottom, there are three buttons: "Factory Reset", "OK", and "Cancel".

Calibration

1) Select the Type of Device to be calibrated

2) Print the 20x20x20mm_cube sample with the setting:

- a) Check the "Raft"
- b) Set the "Number of shells" value to 2
- c) Set the "Infill Density" value to 0.2
- d) Set the "Feedrate" value to 50

3) Measure the size of printed cube with the vernier caliper precisely, and record the value to the input box below, and click "OK" to calibrate.

4) Restart the MPrint. You can restore to factory setting by clicking "Factory Reset"

Measured size of X: mm

Measured size of Y: mm

Measured size of Z: mm

07. TIPS

Tips

Q1 When printing the raft, the filament is not printed evenly on the build plate, and sometimes the plate is even scratched. Why?

Ans Because the plate is not leveled. We suggest you calibrate the build plate when to encounter such situations

Q2 How to avoid edge warping when printing a model of the big base.

Ans

1. Stick double sided tape or textured rubber to enhance the friction,
2. Adjust the build plate upper closer to the plate. Once it is done, the lines printed on the build plate is flat.
3. Increase printing temperature.
4. Shut down the front cooling fan when printing the raft.
5. Open brim function.

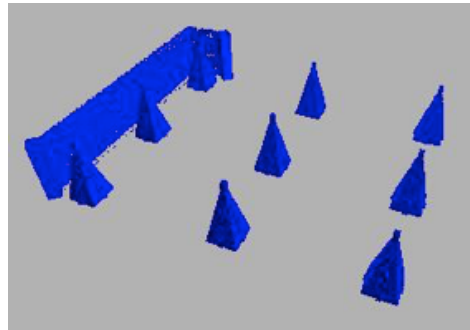
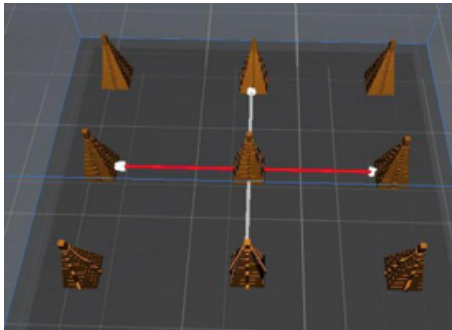
Q3 How to reduce the melt filament thread netting in the hollowed-out model

Ans When printing a hollowed-out model, reduce the temp by 5 to 15 centigrade. Or optimize the Retract parameter setting.

Q4 How to print well when printing models with sharp peaks? Sometimes, the outer cap is not sealed.

Ans The holes can be caused by many factors such as the model's section size, printing speed, printing temperature, density, cap thickness, and cooling effect. Usually, if the section size is too small to leave enough time to allow the layer cool down, the new layer will be added on destroying the former level, that's why some sharp peaks are twisted. So we can improve in the following ways:

1. Decrease temperature. 2. decrease printing speed. 3. Add a purge wall. 4. Print more models and spread them within a big area. 5. Increase density.



Q5 When printing with a dual head machine, the part printed by one head doesn't match the other one on the x-axis, like the following pic shows.



Assumed position



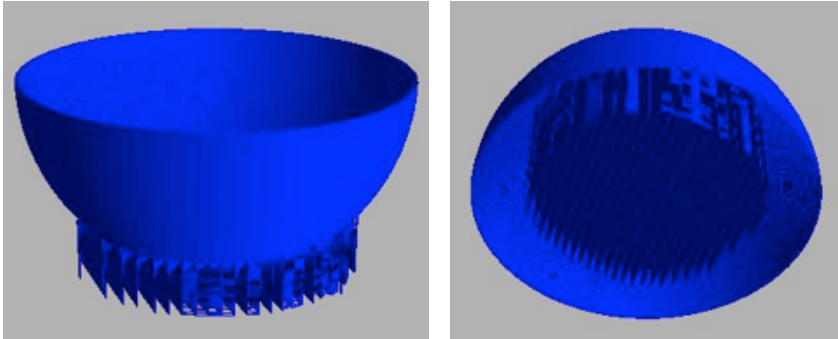
Actual position

Ans For an example MBot Grid II+ dual head printer. It's print volume is 220*220*180mm and the print head can travel as far as 220mm on the X axis. But for the dual printed head, the left one cannot reach the right end, and vice versa. So deducting the width of both print head (35mm), the shared effective X-axis printing range is 150mm. So when printing something big, the model had better be put in the center zone shown as below.



Q6 How to optimize support ?

Ans The surface attached to support is rough, so unnecessary support should be avoided. Take the following model as an example, the left position is recommended since it needs less support, thus consuming less filament and time. Besides, it's easier to take off the support.





Magicfirm

Address: Block A, HuaYe Building, 511 JianYe Road, Binjiang District,
Hangzhou, China 3100052

Tel:

86-571-88021266

[Mon-Fri 9:00 to 18:00 (UTC+8)]

Web:

www.mbot3d.com

Technical Support:

support@mbot3d.com