

FULLSCALE XT PLUS

Desktop 3D Printer

— User Manual —



Contents

WELCOME 1

Safety 2

Specifications 3

How it Works 4

SETUP 5

Unpacking 6

Accessory 9

Tool Box 10

Setting Up Mankati Fullscale XT Plus 11

Fullscale XT Plus Outlook 14

LCD INSTRUCTION 15

LCD Menu Structure 16

LCD Menu 17

SOFTWARE 19

Download and Install 20

Interface 21

Parameter Definition 22

FIRST RUN 28

Stick the Kapton tape 29

Leveling Build Plate 31

Feed the Filament 34

Print from SD Card 35

Unload the Filament 36

MAINTENANCE 37

Maintenance 38

TROUBLESHOOTING 40

Setting and Calibration Problems 41

Hardware Problems 42

Printing Problems 44

How to 46

1

WELCOME TO THE MANKATI FULLSCALE XT PLUS

This User Manual will guide you to quick start with the Mankati Fullscale XT Plus easily. Being a thrilling reform to the next level, Mankati Fullscale XT PLUS will provide you with higher resolution, bigger printing volume and more user friendly functions.

Print your ideas right from now on!

Safety



GROUND CONNECTION

CAUTION: Make sure the power is grounded, if not, fire accident or wires melting may be caused.



HEAT and HIT

WARNING: Mankati Fullscale XT Plus 3D Printer generates high temperatures and moving parts contained may cause injury. To avoid scalding, please never touch the extruder or the heated aluminum block during preheating the nozzle or printing.



CAUTION: Do not leave your machine unattended for a long time during the printing. Do follow these guidances if you have to do it :

- Ensure that the first layer of your printing is properly landed on the heat bed;
- Ensure that filament is twined tightly on the spool, with no knot or tangle, for smooth feeding.
- Monitor your print periodically.

CAUTION: Filament melting will emit plastic odor during printing. Please keep ventilation in the work place.

CAUTION: In case of emergency disconnect power supply from wall socket.

CAUTION: Remoulding or modification by yourself may cause damage or performance exception. Any remoulding or modification, not approved by Mankati , will make your machine no longer under the right of warranty or post-sale services.

COMPLIANCE: Your Mankati Fullscale XT Plus has been tested strictly by SGS with the permission of FCC, CE, IC, C-Tick and ROHS.

Mankati products complies with rules and limits of EMC, MD and ROHS, to reasonably protect human against harm from electromagnetic compatibility, mechanical structure and chemical component.

Users have the right to be informed of the testing report from Mankati.



Specifications

Fullscale XT Plus

Forming Tech.	Fused deposition modeling
Extruder Number	1 or 2 (Dual extruders Perfect supported)
Build Size	260*260*300 mm / 10.2*10.2*11.8 inch

Specification

Precision	X,Y axis 0.01 mm, Z axis 0.015 mm
Layer Resolution	0.04 mm ~ 0.4 mm
Nozzle Diameter	0.4 mm
Max. Travel Speed	250 mm/s
Max. Print Speed	180 mm/s
Max. Extruder Temperature	300 °C / 572 °C
Max. Hot Bed Temperature	110 °C / 230 °C
Max. Extrude Speed	100 mm/s

Filament

Filament Type	ABS, PLA(Spec.), PVA, PS, Nylon
Filament Diameter	3 mm
Filament Temperature	150–300 °C / 302–572 °C
Filament Colors	Multicolor

Hardware

Controller	Arduino ATmega 2560 R3 Micro Controller
Mother Board	Mankati Stable Main Board V1.0
Stepper Motor	X, Y axis 42*48, 1.3 A; Z axis 42*40, 1.3 A
Geared Motor	E42*42, Ratio 1:10
Power Input	100–230 V, 1.5 A
Power Output	24 V, 10.5 A

Software

Operating System	Windows XP/Vista/7/8, Mac OS, Ubuntu
File Format	STL, OBJ, GCode
Printing Software and Slicer	MankatiUM V6,5
Software Language	English, Deutsch, French, Nederlands, Spanish, Polish

Machinery

Dimensions	384*434*535 mm
Weight	46 kg

How it Works

The Mankati Fullscale XT Plus 3D Printer makes solid, three-dimensional objects out of melted Filament. Your 3D designed files are read from SD card and translated into instructions by Mankati Fullscale XT Plus. The machine then heats the filament and through a nozzle squeezes it out onto heat bed to build a solid object, layer by layer. This method is called **Fused Deposition Modeling [FDM]**.



Preparing the model

Use a 3D modeling software to output stl or obj format 3D model file.



MankatiUM

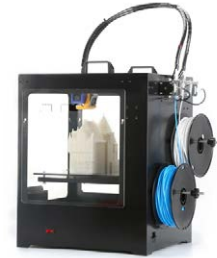
MankatiUM is a software to prepare the gcode file for your Fullscale XT Plus.

In MankatiUM, you can set printing parameters for your model, such as layer thickness, printing speed and nozzle temperature, then output a gcode file for printing.



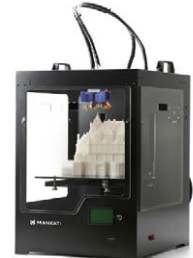
Transfer the GCode file

Save the GCode file into the SD card, and insert it into the SD card reader of the printer.



Prepare the printer

Before 3D printing, you need to load the filament for the printer.



Print it!

Operate the knob beside the LCD and choose the model from the SD card inserted into the printer.

2

SETUP

Every Mankati product has been passed stringent tests before leaving factory, and packaged very carefully to deliver. Hope you would take your time reading this chapter and get your machine set up.

Unpacking



1. Place Mankati Fullscale XT Plus on the floor. Leave enough available space for unpacking.



2. Your machine is packed with carton, cardboard, foam and sealed bag to ensure the perfect status when you received it.

NOTE: You can download a PDF copy of this User manual at the support page: <http://www.mankati.com/download.html>



3. Remove the first cardboard layer. Open the plastic bag and lift it out by pulling it up gently at the left and right frames of your machine.

Caution: Be careful NOT to grip the belts or shaft of the motion system.

Unpacking



4. "Hello, Mankati Fullscale XT Plus!"

Place your machine on a stable surface, and leave enough space surrounding it to check accessories.

Caution: When taking your 3D printer out of the box, please be careful NOT to grip the belt or shaft of the motion system.



5. Removing the stretch film off, you can see all the accessories, tools and filaments.



6. Take all the accessories out.

Unpacking



8. Taking out the filaments:

For easy and safe transport, spools of filament are fixed tightly between the build plate supporters.

Take them out and follow the 3 steps below:



- 1) Connect power wire to your machine;
- 2) Turn it on;



- 3) Enter into LCD panel's menu: "Prepare > Autohome"; Then the two build plate supporters will lift to the top of Z axis;
- 4) Tacking out the spools of PLA filament and feeding devices.

Accessory



Taking out accessories, tool box and filament and check if below things are included:

- 1) 2 spools of PLA filament, diameter 3mm, 1kg/spool;
- 2) 1 Build plate.



- 3) 1 tool box: including the necessary tools for installation and other accessories.
- 4) 2 feeding devices: constituted by gear motor and feeder, with the advantages of high accuracy and easy operation.



- 5) 3 PC cover boards: installed on 3 sides(front, left, right) to keep the temperature of inside printing space constant.



Tool Box



Check if the following tools are complete. If anything described is missing, please contact:

support@mankati.com.

- 1) 1* diagonal pliers: to dismantle the support underneath your printing model or to prune filament.
- 2) 4* Spanners: installation tools.
- 3) 1* Glove pairs: as gift for the user.
- 4) 1* Lubricant oil: to lubricate XY axis and guide rails regularly.

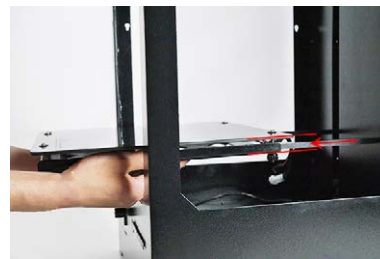
- 5) 1* Kapton tape: stuck on the build plate to fix your printing model on it.
- 6) 1* Scraper: to shovel out the model and clean the build plate.
- 7) 1* Screwdriver: the installation tool.
- 8) 1* Art knife: to improve the surface texture of your model printed out.
- 9) 1* Tweezer: to clean filament extruded out by the nozzle.
- 10) 2* Filament holders: holding filament spools.
- 11) 2* Printer handles: installed on your machine for moving it easily.
- 12) 3* Inner hexagonal spanner (L-shape wrench): installation tools.
- 13) 1* Drill: to clean the nozzle.
- 14) 1* Plastic scraper: to clean the heat bed after printing.
- 15) 1* SD card to store Gcode file for printing.
- 16) 2* Screw & Nut bags: as spare parts.
- 17) 1* Power wire: supplying power.
- 18) 1* USB cable: connecting the computer and your printer when printing or upgrading firmware.

Setting Up Mankati Fullscale XT Plus

1. Install the Build Plate



1) Turn on your Fullscale XT Plus, enter "Prepare > Move axis > Move 1mm> Move Z", lower Z axis a little(e.g., 30mm) to avoid collision of nozzle and build plate during installation.



2) Insert the build plate's holder into the middle of plate.

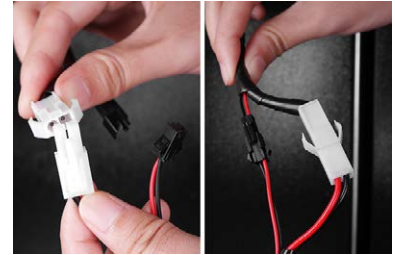
Note: The wire connection side should be put toward your machine.

3) Take out the nuts ⑥ and screwdriver from the tool box to fix the platform firmly.



4) After installing the plate, please tighten the 4 screws under it properly to lower it and avoid knocking the extruders in the plate leveling operation.

Note: Do not screw the screws too tight!
Or your build plate glass maybe cracked.



5) Connect the power wire and data cable of platform with the connectors.

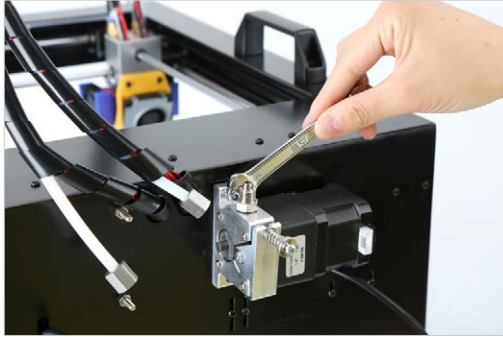
Caution: Make sure the connection fully. Otherwise, it can cause the platform heating problems or connectors overheated.

6) Done.

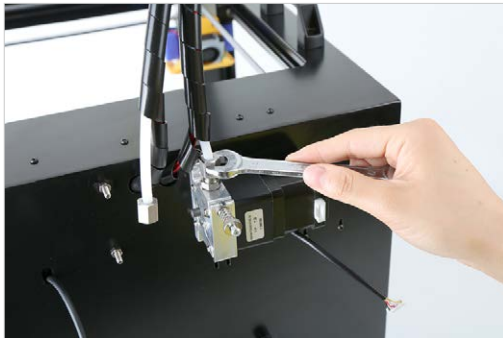
Setting Up Mankati Fullscale XT Plus

3. Feeding Device Installation

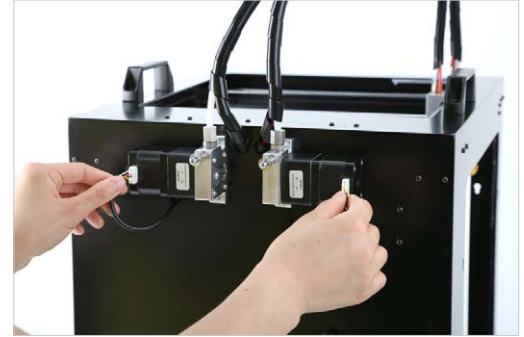
1) Feeding motor:
using the small
spanner to install No.
1 feeding motor on
position 1.



2) Insert the end
of feeding tube
into the socket of
feeding motor and
tighten it firmly.



3) Connect the data cable
to the feeding devices.



4) Feeding device
installation done.



Setting Up Mankati Fullscale XT Plus

4. Filament Spool Holder Installation

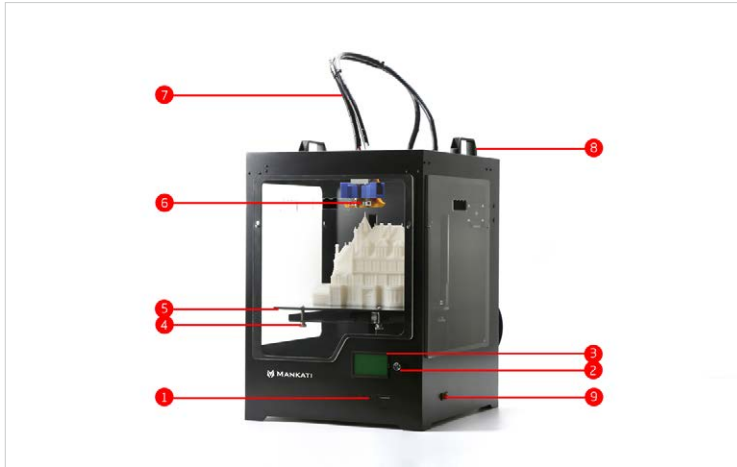
1) Take the two spool holders out of the tool box.



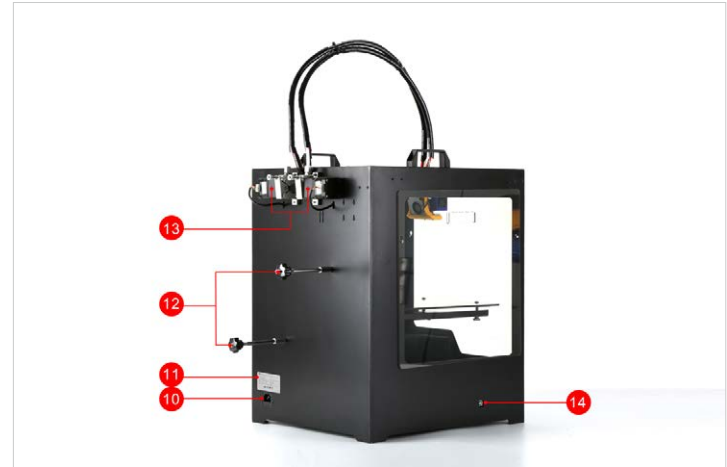
2). Install filament spool's holder.



Fullscale XT Plus Outlook



- 1. SD-card slot
- 2. Push and rotate knob
- 3. LCD screen
- 4. Build plate screws
- 5. Build plate
- 6. Print head
- 7. Filament guide tube
- 8. Printer holder
- 9. Power switch button



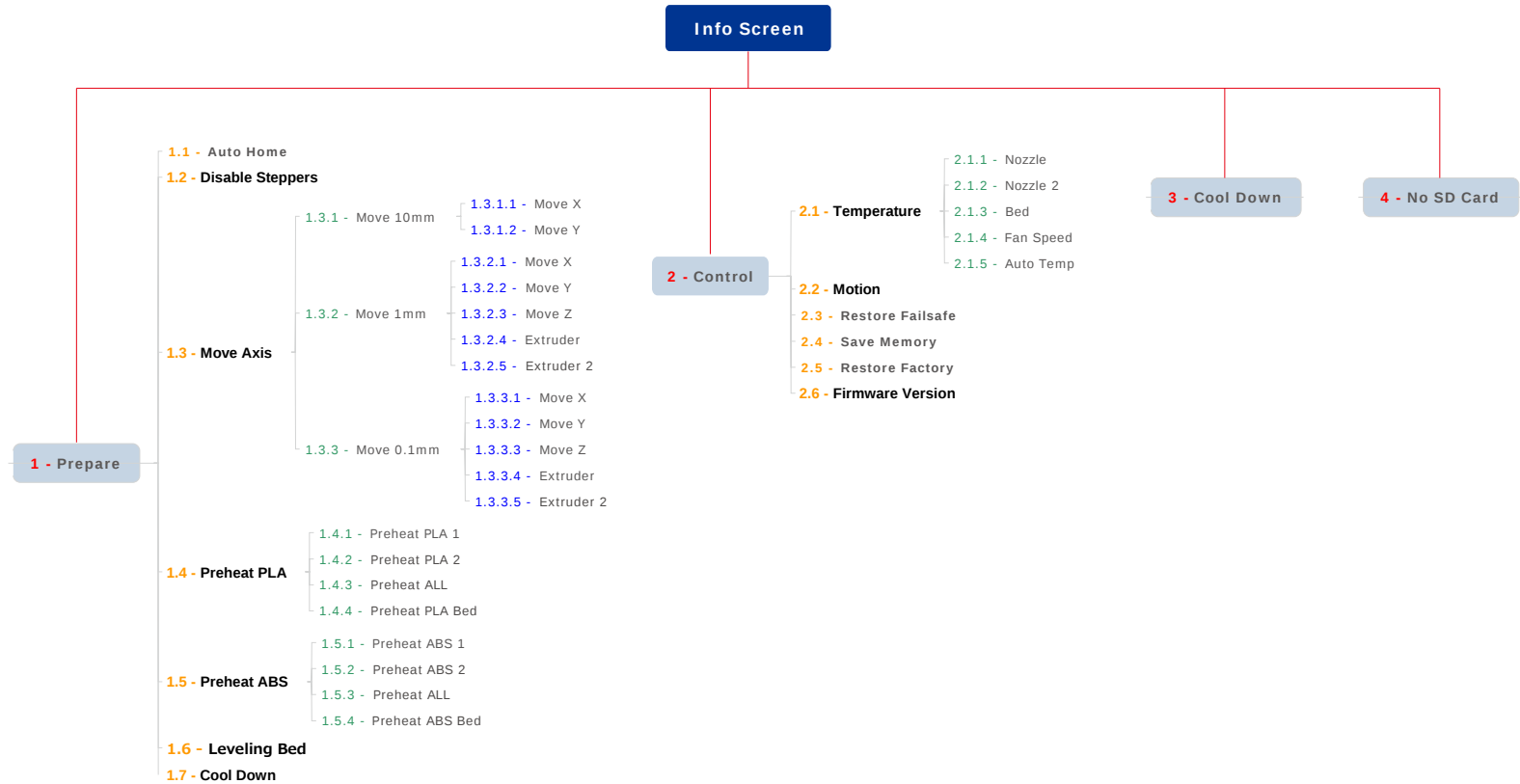
- 10. Power Socket
- 11. Machine label
- 12. Spool holder
- 13. Feeder motor
- 14. USB Socket

3

LCD INSTRUCTIONS

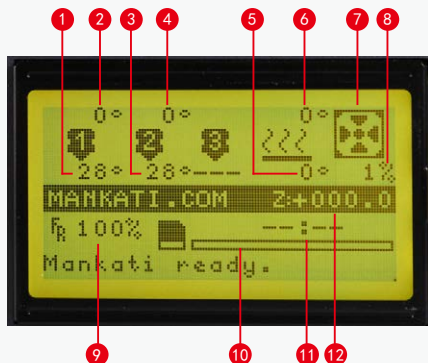
In this chapter you can learn about detailed instructions of the LCD menus.

LCD Menu Structure



LCD Menu

Info Screen



1. Actual Temperature of Nozzle 1;
2. Preset Temperature of Nozzle 1;
3. Actual Temperature of Nozzle 2;
4. Preset Temperature of Nozzle 2;
5. Actual Temperature of Build Plate;
6. Preset Temperature of Build Plate;
7. Cooling Fan;
8. Fan Speed Rate;
9. Filament Feed Rate;
10. Printing Process Bar;
11. Printing Time;
12. Build Plate Coordinate;

Press the knob to enter the menu:

1 - Prepare: The printer enters into ready state

1.1 - Auto home

The printer returns to original state (In X.Y direction, the extruder returns to the left bottom in top view; In Z direction, the build plate returns to top position)

1.3 - Move axis:

Move 10mm: Move 10mm each step

Move X: Move 10mm each step in X direction

Move Y: Move 10mm each step in Y direction

Move 1mm: Move 1mm each step

Move X: The extruder will move 10mm each step in X direction

Move Y: The extruder will move 10mm each step in Y direction

Move Z: The build plate will move 10mm each step in Y direction

Extruder: The feeding motor of left nozzle will extrude the filament in each step of 1mm

Extruder2: The feeding motor of right nozzle will extrude the filament in each step of 1mm

Move 0.1mm: Move 0.1mm each step

Move X: The extruder will move 0.1mm each step in X direction

Move Y: The extruder will move 0.1mm each step in Y direction

Move Z: The build plate will move 0.1mm each step in Y direction

Extruder: The feeding motor of left nozzle will extrude the filament in each step of 0.1mm

Extruder2: The feeding motor of right nozzle will extrude the filament in each step of 0.1mm

1.4 - Preheat PLA:

Preheat PLA1: Preheat extruder and build plate to the working temperature of PLA required (Default: Extruder, 210°C , Build plate, 45°C)

Preheat PLA2: Preheat extruder 2 and Build plate to the working temperature of PLA required (Default: Extruder, 210°C ; Build plate, 45°C)

Preheat PLA All: Preheat two extruders both and build plate to the working temperature of PLA required (Default: Extruder, 210°C ; Build plate, 45°C)

Preheat PLA Bed: Preheat build plate to the working temperature of PLA required (Default: Build plate, 45°C)

1.5 - Preheat ABS:

Preheat ABS1: Preheat extruder and build plate to the working temperature of ABS required (Default: Extruder, 235°C ; build plate, 70°C)

Preheat ABS2: Preheat extruder 2 and build plate to the working temperature of ABS required (Default: Extruder, 235°C ; Build plate, 70°C)

Preheat ABS All: Preheat two extruders both and build plate to the working temperature of

LCD Menu

ABS required (Default: Extruder, 230 C ; Build plate, 70 C)

Preheat ABS Bed: Preheat build plate to the working temperature of ABS required (Default: Build plate, 70 C).

1.6 - Leveling Bed: Build plate calibration assist-function. Rotate the button, the extruder will move to next position in clockwise direction from top view.

1.7 - Cool down: Cool down the extruder and build plate to normal temperature.

2 - Control: Adjust printer settings

2.1 - Temperature:

Nozzle: Left temperature setup

Nozzle 2: Right temperature setup

Bed: Build plate temperature setup

Fan speed: Cooling fan speed setup

Auto Temp: All the above data setup automatically

2.2 - Motion:

Please do NOT change any data in this menu if without specialist experience.

Accel: Accelerated speed of extruder motion

Vmax x: Max speed in X direction

Vmax y: Max speed in Y direction

Vmax z: Max speed in Z direction

Vmax e: Max speed of feeding motor

Vmin: Min printing speed

VTrav min: Min travel speed

Amax x: Max accelerated speed in X direction

Amax y: Max accelerated speed in Y direction

Amax z: Max accelerated speed in Z direction

Amax e: Max accelerated speed of feeding motor

A-retract: Accelerated speed of retract

Xsteps/mm: Transmission ratio of X motor

Ysteps/mm: Transmission ratio of Y motor

Zsteps/mm: Transmission ratio of Z motor

Esteps/mm: Transmission ratio of feeding motor

2.3 - Restore Failsafe: Please do NOT change any data setting here in this menu.

2.4 - Save memory: Record the coordinate of printing stop

2.5 - Restore factory: Restore the manufacturer default settings

2.6 - Firmware version: Current firmware version

Menus in Printing Process

During the printing, you can press the knob to change some printing settings.

1. Tune:

Re-set some parameters

- **F/R:** the actual printing speed takes the percentage of the pre-set speed in the software. It can be adjusted while printing according to the actual situation.

- **Nozzle:** Temperature setup of left nozzle

- **Nozzle 2:** Temperature setup of right nozzle

- **Bed:** Temperature setup of build plate

- **Fan speed:** Fan speed setup

- **Flow:** Filament feeding flow rate

- **Flow 0:** Filament feeding flow rate of left nozzle

- **Flow 1:** Filament feeding flow rate of right nozzle

- **Auto Temp:** Set all the above temperature automatically

- **Change filament:** To stop the printing and change the filament. It will record the printing stop position so as to continue to print after filament changing.

2. Control:

Please do NOT change any data in this menu if without specialist experience.

3. Pause: To stop the printing, but it will record the printing stop position to continue the printing.

4. Stop Print: To finish the printing job

4

SOFTWARE

This chapter guide you how to download and install the software, and show you detailed introductions of parameters in MankatiUM.

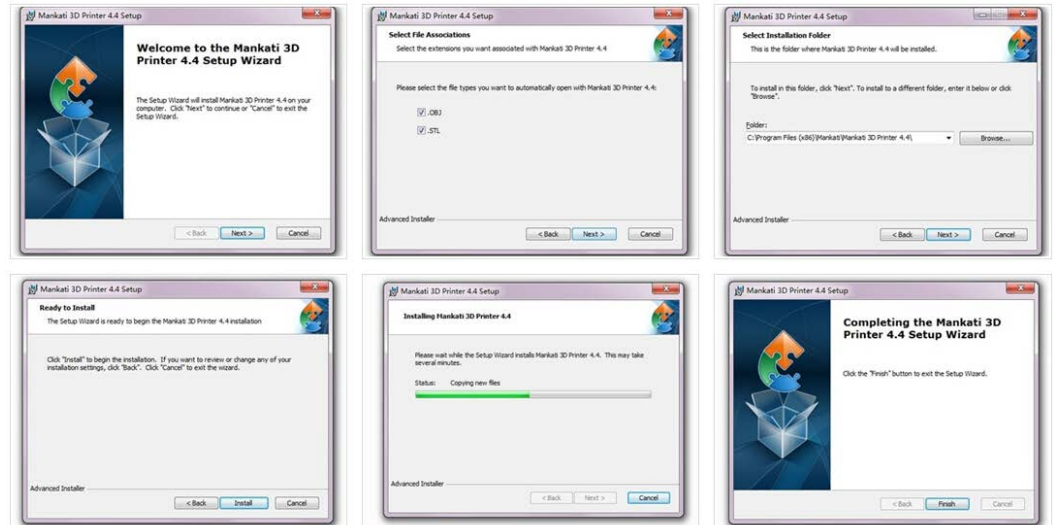
Download and Install

Enter Mankati official download website:

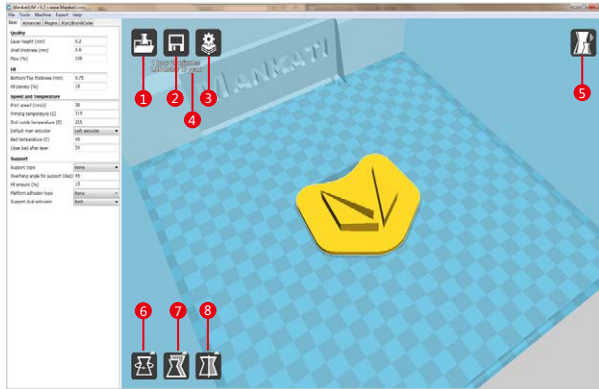
www.mankati.com/download

Choose the appropriate version of Mankati UM according to your computer operation system. Our software apply to Windows, MacOS, Ubuntu operating system.

Right click the software installation program, select "run as administrator" and install it following the prompts.



Interface



1. Load 3D model, supporting the files of STL/OBJ/DAE/AMF.

Note: MankatiUM also can recognize planar images, and transfer them to 3D model files. Supporting the files of JPG/BMP/PNG.

2. Save as Gcode file
3. If change any slicing setting, please click this "Slice" button to show the real printing time and filament requirement.
4. Printing time, the length and weight of filament consume.
5. Simulate the printing process under different modes.

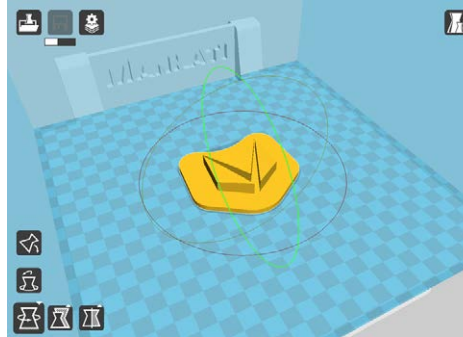
5.1 Normal: Default state

5.2 Overhang: Measure the angle of overhang, and judge if it is necessary to generate the support structure

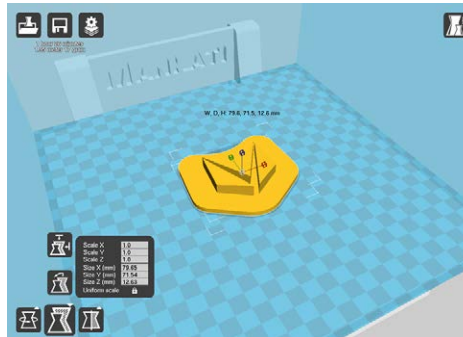
5.3 X-Ray: X-ray vision

5.4 Layers: Layer by layer to simulate the printing process

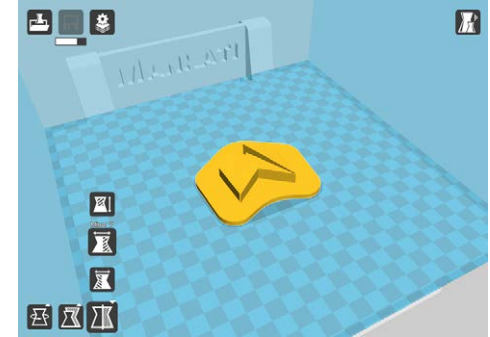
6. Rotate the model



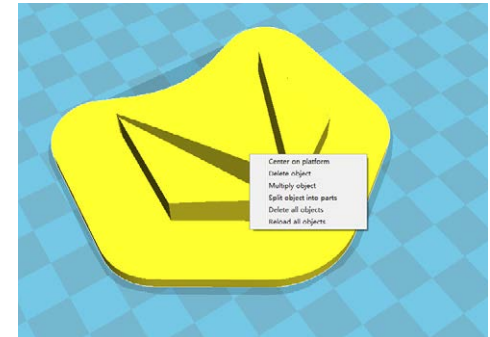
7. Change the size of the model



8. Mirror image of the model



9. Other operation



Five options for choice:

- Center on platform
- Delete object
- Multiply object

- Split object into parts (It will take too much time, so it is better to do this in modeling software)
- Delete all objects

Parameter Definition

Basic Setting

Note: Function prompt will be shown when you point the cursor on the tab.

Quality

Layer height: The lower layer height, the higher printing accuracy within certain limits

Super accuracy printing: Recommended set as 0.1mm

High accuracy printing: Recommended set as 0.2mm

Giving consideration to both printing speed and accuracy printing: recommended 0.2mm

High speed printing: Recommended 0.3mm

Higher precision printing is available as 0.04mm, but because the filament extruded at 0.04mm is too soft and thin, which cannot print complicated model. Meanwhile, the speed will get low too much.

So here we don't recommend the printing less than 0.1mm unless it is inevitable.

Shell thickness: Thickness of the outside shell in the horizontal direction. Common setting as integral multiples of nozzle diameter to represent the outer wall print laps.

Flow: Flow compensation, the amount of material extruded is multiplied by this value.

Fill

Bottom/Top thickness: Used as the model need capping. Common setting as integral multiples of layer height, keep it near your wall thickness to make an evenly strong part.

Fill Density: Means the net support structure inside the model. The higher fill percentage, the higher density of net support. 0 means non-fill, and 100% means solid fill.

Speed and Temperature

Printing Speed: Means the extruder's moving speed. Printing speed decides the quality of printing model quality. Mankati Fullscale XT Plus's default speed is 30mm/s. High speed printing can be set as 50-70mm/s.

Higher speed printing needs a balance by increasing nozzle temperature, otherwise may easily cause shortage of filament's feeding. Printing speed also depends on the shape of model and many other factors. Recommended speed is ≤ 70 mm/s. ≥ 70 mm may cause the step motor losing steps, which will reduce the model printing accuracy.

Printing Temperature: Main extruder temperature(default as left extruder), setting according to the different melting temperatures of different filaments.

2nd extruder temperature: Vice extruder temperature(default as right extruder), setting according to the different melting temperatures of different filaments.

Default main extruder: Choose which one to be the main extruder. Default as left extruder. After finishing a model and you want to print another model in different filament(different colors and different materials). Changing vice extruder as main extruder can solve frequent changing filament problem. This function can come effective only when printing single color filament.

Bed temperature: Setting according to the different melting temperatures of different filaments.

Close bed after layer: Close bed temperature after certain layers. Not only energy, but also can prolong the life of the hot bed. 0 disable this feature.

Parameter Definition

Support

Support type: For the model with parts hanging in the air, support structure is a must when printing to ensure the parts hanging in the air will not collapse.

- None:** No support, if you print model with parts hanging in the air but choose structure as none, the parts hanging in the air may droop.
- Touching Build Plate:** Most commonly used support type, means the support will only be created when the support structure touches the build plate.
- Everywhere:** Create support wherever there is part hanging in the air.

Note: Support structure may affect the smoothness of model surface and printing speed.

If not essential, it's recommended to modify or split model to reduce the use of support.

Platform adhesion type: Different options that help in preventing corners from lifting due to warping.

- **None:** Default form of using outsider lining.
- **Brim:** Adds a single layer thick flat area around your object which is easy to cut off afterwards, and the recommended option.
- **Raft:** Adds a thick raster at below the object and a thin interface between this and your object.

Note: Enabling the brim or raft disables the skirt.

Support dual extrusion: Which extruder to use for support material. For easy break-away support you can use either extruder.

You can choose main extruder(left extruder) to print main structure, 2nd extruder(right extruder) to print soluble support material.

Advanced Setting

Basic	Advanced	Plugins	Start/End-GCode
Machine			
Nozzle size (mm)		0.4	
Retraction			
Speed (mm/s)		80.0	
Distance (mm)		10.0	
Dual extrusion switch amount (mm)		19.5	
Quality			
Initial layer thickness (mm)		0.25	
Initial layer line width (%)		100	
Cut off object bottom (mm)		0.0	
Dual extrusion overlap (mm)		0.1	
Speed			
Travel speed (mm/s)		70.0	
Bottom layer speed (mm/s)		30	
Infill speed (mm/s)		45	
Top/bottom speed (mm/s)		45	
Outer shell speed (mm/s)		45	
Inner shell speed (mm/s)		45	
Cool			
Minimal layer time (sec)		5	
Enable cooling fan		☒	

Note: Function prompt will be shown when you point the cursor on the tab.

Retraction

Enable retraction:

Retract the filament when the nozzle is moving over a non- printed area to avoid dropping and brushed. Details about the retraction can be configured in the advanced tab.

- Speed:** Speed at which the filament is retracted, a higher retraction speed works better. Commonly used as 80mm/s. A very high retraction speed can lead to filament grinding or breaking.
- Distance:** Amount of retraction. Set at 0 for no retraction at all. A value of 5 usually generates good result.
- **Dual extrusion switch amount:** Amount of retraction when switching nozzle with dual-extrusion.

Quality

-**Initial layer thickness:** Layer thickness of the bottom layer. A thicker bottom layer makes sticking to the bed easier. 100 means setting the bottom thickness the same as other layers.

-**Initial layer line width:** Extra width factor for the extrusion on the first layer.

-**Cut off object bottom:** Which sinks the object into platform. This can be used for objects that do not have a flat bottom or cutting off the already finished model height in order to stick or montage with other parts easily.

-**Dual extrusion overlap:** Adding a certain amount of overlapping extrusion on dual-extrusion prints.

Parameter Definition

Speed

-Travel Speed: Moving speed without printing. Usually setting at 70 is the best choice, Fullscale XT Plus can reach speeds of 250mm/s at most. But this may cause the motor missing steps problem.

-Bottom layer speed: The bottom print speed. Recommended 20, set as 0 means the same as other layer. Reducing the bottom layer speed can help model stick more closely to platform.

-Infill speed: Speed at which infill parts are printed. Recommended 40 with best performance. 0 means same print speed as other layer. Printing the infill faster can greatly reduce printing time, but this can negatively affect print quality.

Inner shell speed: Speed at which inner shells are printed. If set to 0 then the print speed is used. Printing the inner shell faster than the outer shell will reduce printing time. It is good to set this somewhere in between the outer shell speed and the infill printing speed.

-Top/bottom speed: Speed at which top/bottom parts are printed. Recommended 25 with best performance. 0 means printing speed is used for the infill. Printing the top/bottom faster can greatly reduce printing time, but this can negatively affect print quality.

-Outer shell speed: See function prompt in the software

-Inner shell speed: See function prompt in the software

-Enable cooling fan: enable cooling fan during printing to help cool filament down.

Cool

-Minimal layer time: The minimal time of printing layer to ensure enough cooling time before printing next layer. Print speed will adjust automatic to satisfy this feature.

Parameter Definition

Expert Setting

Expert config

Dual extrusion

Wipe&prime tower volume per layer (mm3) 15

Retraction

Minimum travel (mm) 2

Enable combing Off

Minimal extrusion before retracting (mm) 0

Z hop when retracting (mm) 3

Skirt

Line count 1

Start distance (mm) 3

Minimal length (mm) 260

Cool

Fan full on at height (mm) 0.5

Fan speed min (%) 100

Fan speed max (%) 100

Minimum speed (mm/s) 10

Cool head lift ☐

Infill

Solid infill top ☒

Solid infill bottom ☒

Infill overlap (%) 5

Support

Structure type Lines

Overhang angle for support (deg) 10

Fill amount (%) 15

Distance X/Y (mm) 1

Distance Z (mm) 0.1

Black Magic

Spiralize the outer contour ☐

Only follow mesh surface ☐

Brim

Brim line amount 5

Raft

Extra margin (mm) 5

Line spacing (mm) 2

Base thickness (mm) 0.25

Base line width (mm) 0.5

Interface thickness (mm) 0.2

Interface line width (mm) 0.4

Airgap 0.0

First Layer Airgap 0.2

Surface layers 2

Surface layer thickness (mm) 0.27

Surface layer line width (mm) 0.4

Fix horrible

Combine everything (Type-A) ☒

Combine everything (Type-B) ☐

Keep open faces ☐

Extensive stitching ☐

Ok

Dual Extrusion

-Wipe & prime tower volume per layer: The wipe-tower is a tower printed on every layer when switching between nozzles. The amount of material put in the wipe/p rime tower. This is done in volume because in general you want to extruder a certain amount of volume to get the extruder going, independent on the layer height. This means that with thinner layers, your tower gets bigger.

Retraction

-Minimum travel: minimum amount of travel needed for a retraction to carry all. To make sure you do not get a lot of retraction in a small area, value = 2 is recommended as best performance, too small will lead to frequent retraction.

-Enable combing: combing is the act of avoiding holes in the print from the head to travel over. Enable combing extruder will always touch the surface of model but disable retraction. If combing is disabled the printer head moves straight from the start point to the end point and it will always retract. Default enable coming, but it will increase printing time.

-Minimal extrusion before retracting: The minimal extrusion that needs to be done before retracting again. If a retraction needs to happen before this minimal is reached the retraction is ignored. This avoids retraction a lot on the same piece of filament which flattens the filament and causes grinding issues. Recommended 0.02 at best performance, 0 means unlimited retraction frequency.

-Z hop when retracting: See function prompt in the software.

Skirt

-Line count: The skirt is a line drawn around the object at the first layer. This helps to prime your extruder, and to see if the object fits on your platform. Recommended setting is 1 with best performance. Setting this to 0 will disable the skirt. Multiple skirt lines can help priming your extruder better for smaller objects.

-Start distance: The distance between skirt line and model. Default distance is 3.

-Minimal length: The minimal length of the skirt, if this minimal length is not reached it will add more skirt lines to reach this minimal length. Recommended setting is 260 with best performance. If the line count is set to 0 this option is ignored.

Cool

-Fan full on at height: The height at which the fan is turned on completely. For the layers below this the fan speed is scaled linear with the fan off at layer 0.

-Fan speed min: Minimal fan speed during normal printing.

-Fan speed max: Maximal fan speed when automatic slowing down printing.

-Minimum speed: minimum printing speed of each layer. Recommended setting is 10 with best performance. Too slow will result in nozzle oozing.

-Cool head lift: lifting the head, if the minimal layer time is not enough, to extend the cooling time for model cooling.

Parameter Definition

Infill

-Solid infill top: Create a solid top surface, if set to false the top is filled with the fill percentage. Disable this feature and set fill density as 0 to print cups/vases that are without top model.

-Solid infill bottom: Create a solid bottom surface, if set to false the bottom is filled with the fill percentage. Disable this feature and set fill density as 0 to print buildings that are without bottom model.

-Infill overlap: Amount of overlap between the infill and the walls. There is a slight overlap with the walls and the infill so the walls connect firmly to the infill. Recommended setting as 10 with best performance. Too big will influence quality of model surface.

Support

-Structure type: See function prompt in the software

-Overhang angle for support: See function prompt in the software

-Fill amount: Support print density.

Recommended setting as 15 with best performance. More density makes the support not easy to remove. Less density influences print quality.

-Distance X/Y: Distance of the support material from the print, in the X/Y directions. 0.7mm gives a nice distance from the print so the support does not stick to the print. Too far will influence support performance.

-Distance Z: Distance from the top/bottom of the support to the print. A small gap here makes it easier to remove the support but makes the print a bit uglier. 0.15mm gives a good separation of the support material. Too far will influence support

performance.

Spiralize

-Spiralize the outer contour: Spiralize is smoothing out the Z move of the outer edge. This will create a steady Z increase over the whole print. This feature turns a solid object into a single walled print with a solid bottom.

When enabling this feature, the top will not be printed and automatically setting infill as 0.

Black Magic

-Spiralize the outer contour: New function in MankatiUM 6.5. Spiralize is smoothing out the Z move of the outer edge. This will create a steady Z increase over the whole print. This feature turns a solid object into a single walled print with a solid bottom. When enabling this feature, the top will not be printed and automatically setting infill as 0.

-Only follow mesh surface: new function in MankatiUM 6.5. See function prompt in the software.

Birm

-Birm line amount: Default as 5, more lines, more firmly the model sticking to platform.

Raft

-Extra margin: If the raft is enabled, this is the extra raft area around the object which is also rafted.

-Line spacing: See function prompt in the software.

-Base thickness: See function prompt in the software.

-Base line width: See function prompt in the software.

-Interface thickness: See function prompt in the software.

-Interface line width: See function prompt in the software.

-Airgap: See function prompt in the software.

-First Layer Airgap: See function prompt in the software.

-Surface layer: See function prompt in the software.

-Surface layer thickness: See function prompt in the software.

-Surface layer line width: See function prompt in the software.

Fix Horrible

Combing everything (Type-A)/(Type-B):

This expert option adds all parts of the model together. The result is usually that internal cavities disappear. Depending on the model this can be intended or not. Enabling this option is at your own risk. Type-A is depended on the model normals and tries to keep some internal holes intact. Type-B ignores all internal holes and only keeps the outside shape per layer.

Keep open faces:

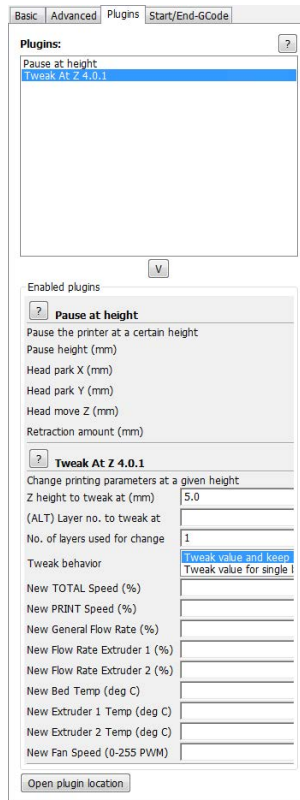
This expert option keeps all the open bits of the model intact. Normally MankatiUM tries to stitch up small holes and remove everything with big holes, but this option keeps bits that are not properly part of anything and just goes with whatever it is enable you to slice models otherwise failing to produce proper paths. As with all "Fix horrible" options, results may vary and use at your own risk.

Extensive stitching:

Extensive stitching tries to fix up open holes in the model by closing the hole with touching polygons. This algorithm is quite expensive and could introduce a lot of processing time. As with all "Fix horrible" options, results may vary and use at your own risk.

Parameter Definition

Plugins



In MankatiUM 6.5, we supply 2 types plugins for you. These 2 plugins will help to give stronger control feature. You can set different parameter with different print heights to have more accurate model.

Pause at height: You can pause printing at a certain height of the print, and you can precisely locate the print head position.

Tweak At Z: You can reset the printing parameters at a certain height of the print, this is a powerful option for experts.

You can reset printing speed, flow, bed temperature, nozzle temperature and fan speed.

5

FIRST RUN

This chapter guides you through steps to start your first run of the Mankati Fullscale XT Plus.

Stick the Kapton tape

Stick the kapton tape onto the heat bed.

It is a kind of high temperature resistance doubleside tape.



1. Clean the heat bed glass carefully to ensure it's clean and tidy. About 3 lines of tape is needed to cover the glass completely.

Take the tape out(15). If covering the plate completely, it will need 3 lines of the tape.



2. Uncover one side of the tape to stick it on the plate. Please ensure the tape is smooth enough without any bubbles.



3. You can use the yellow shovel in tool box to push and press it while sticking.



Stick the Kapton tape

4. Cut the tape off along the edge of the heat bed glass.



5. Cut off the parts on the screws when sticking the third line. If some big bubbles appear, just use a needle to stick into them. Then smooth them with yellow board.



6. After sticking the tape on the plate, please uncover the other side of the tape.

Note: After several printing, the stickiness is not strong enough, please change the tape.



Leveling Build Plate

Notice:

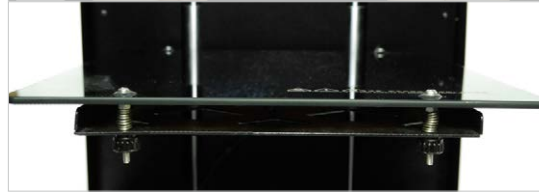
- Tightening the black screw (turning them to the right) to reduce plate height.
- Loosening the black screw (turning them to the left) to increase platform height.
- The gap between the nozzle and the build plate should be about 75% of the layer thickness you set in the software.

Why Leveling Build Plate is Important?

Frequent build plate leveling can keep Mankati Fullscale XT Plus best performance. Re-leveling cannot hurt the printer but without leveling will decrease your success rate of printing.

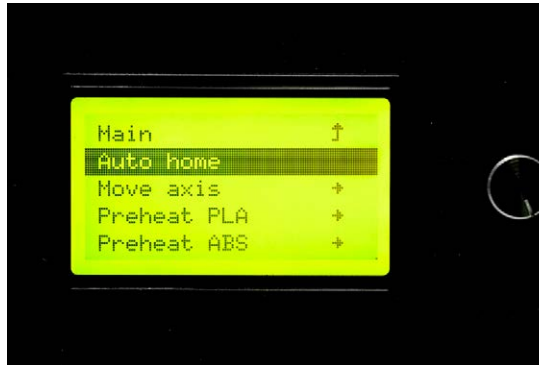
- If the build plate is too far from the extruder nozzle, or if one part of the build plate is further from the nozzle than another part, your object might not stick to the build plate.
- If the build plate is too close to the extruder nozzle, the build plate will block the extruding of filament from the nozzle. The nozzle will scratch the build plate
- Frequent build plate leveling will help to ensure that objects always stick well to the hot bed.

Plate Leveling Guide



1. Twisting the four black screws tight properly to reduce the build plate height and avoid collision between nozzle and plate.

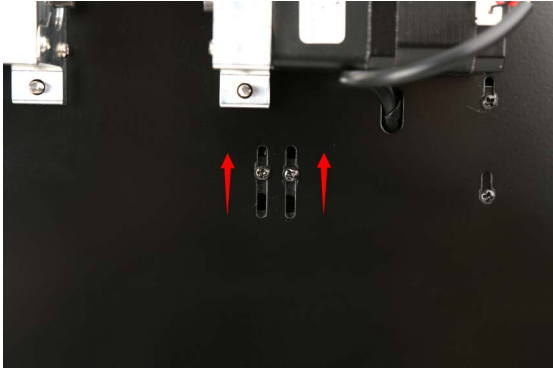
Note: Twisting screws too tight may cause the glass cracking!



2. Lifting the build plate to the top of the Z-axis. Go to menu "Prepare" - choose "Auto home" in the LCD, the extruder/print head will go to the original point, the build plate will go up to the top of Z-axis.

Leveling Build Plate

3. If the gap distance between nozzles and build plate is still too big (more than 4-5mm) after "Auto home", please adjust the limit stopper on the rear side of your machine. Loosen the 2 screws then move it upwards a few mm.



4. Now, you can start to adjust the four corners:

Insert a business card into the gap between the extruder and the hot bed, then move it back and forth. You should feel some friction on the card but still able to easily move the card between the gap without tearing or damaging the card.



5. In the same way, adjust the left 3 corners in anti-clockwise direction.



6. Repeat to adjust the 4 corners a little at least 3-4 times to ensure the best gap distance.



Feed the Filament

1. Preheat the nozzle:

Choose "Prepare">"preheat PLA (or ABS)" on LCD.
Choose the nozzle you want to preheat.



2. Feed the end of the spooled filament into the filament guide tube, where it attaches to the extruder.



3. Continue to feed the filament in until it emerges from the nozzle.

Note: Please DO push the filament GENTLY.

If filament attaches the nozzle, just keep them attached, stop pushing hardly. Otherwise, the nozzle connection could be broken.

After the nozzle is preheated enough, the filament will flow out from nozzle naturally.



4. Tighten the black screw to fix the filament with the clamp, making it not too loose or too tight.



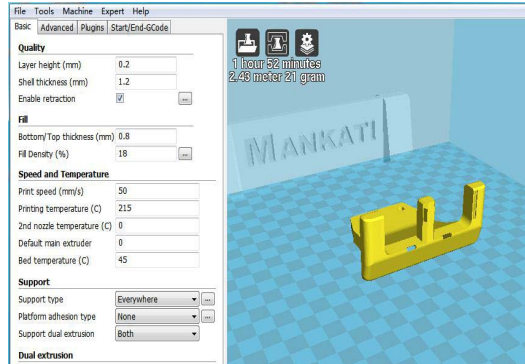
Caution:

1. Ensure the clamp is not too tight or too loose, otherwise, filament feeding will have problem causing failure prints.
2. After feeding successfully, check if filament has twining and knotting to ensure filament feeding smoothly.
3. Lubricate the XY axis and sliding blocks to avoid motion stuck.

Print from SD Card

1. Import the .stl format model files in our slicing software (Mankati UM), and output the Gcode files in the SD card.

Note: It is better to use the default settings for the first run.



2. Insert the SD card into Mankati Fullscale XT Plus.



3. Choose "Print from SD card", then select your Gcode file.



4. When your printing finished, the build plate will move down and be heated to 50°C automatically for taking your model off more easily.

Use shovel to take it out gently to avoid the damage of the heat bed glass and your model.



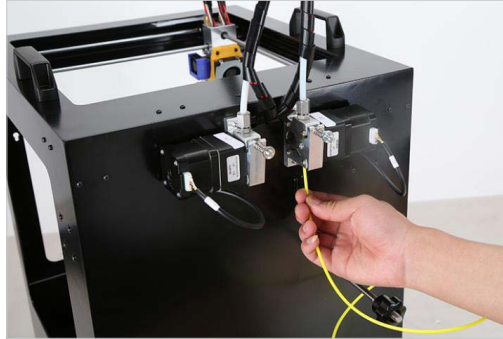
Note: If the model is difficult to be off, you can raise the build plate temperature to 60. Too much power pulling the model will cause damage to the glass surface! This kind of damage during use is NOT covered by the warranty of "free parts replacement service". Please do the removing carefully!!

Warning: Do not touch the nozzle while it is totally cooled down. Wait until the actual temperature on LCD is lower than 30 degrees.

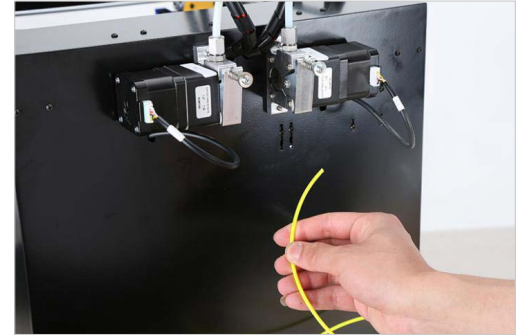
Unload the Filament



1. Heat the nozzle to the required temperature of filament. For example, PLA 210°C and ABS 235°C .



2. Pull the filament out in direction of perpendicular to the feeder motor hardly and quickly.



2. Done.

Note:

1. If the filament is stuck, please find solution in troubleshooting chapter.
 2. If your machine stops working for a long time, please switch off the printer to avoid the filament carbonized, which will cause the nozzle clogged.
- Clean the dust on the build plate, that will extend the lifetime of the kapton tape.

6

MAINTENANCE

To keep your Mankati Fullscale XT Plus in perfect performance, some basic periodical maintenance is a must.

This chapter shows you some practical tips which will help you to take care of your printer well.

Maintenance

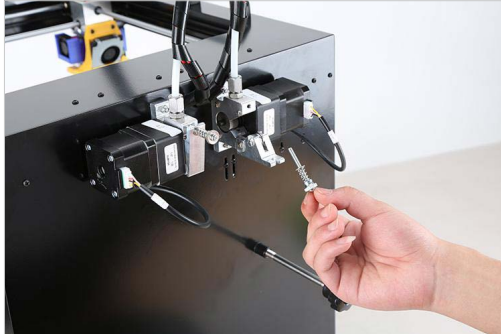
Build Plate

Ensure that the plate is clean. Bubbles, scratches, dust, and oil from your hands are avoided.

1. Use tweezer to remove filament residue.
2. Do not touch the platform glass with sharp things.
3. If not using the printer for a long time, please clean up the tape, clean the glass and fully release the four black screws under the build plate.

Cleaning the filament feeder

After a lot of printing, the wheel in the filament feeder might accumulate small plastic particles. You can clean this with a brush.



1. Remove the black screw on the feeder motor totally.



2. Select "Prepare" > "move axis" > "move 1mm" > "move extruder". The feeder gear will start to work.



3. Clean small plastic particles with a brush.

Maintenance

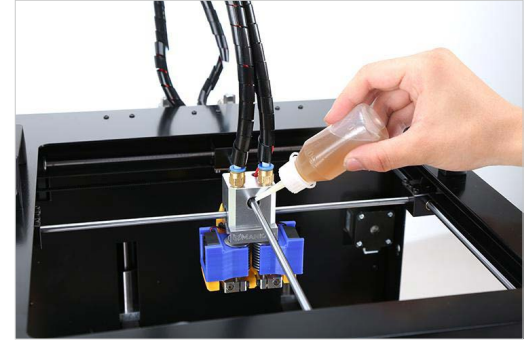
Lubricating the guide rails



1. Clean the X-Y axis and guides rails.



2. Lubricate the X-Y axis and rail guides periodically, no overflow is appropriate.



7

TROUBLESHOOTING AND SUPPORT

In case something goes wrong with your Mankati Fullscale XT Plus, the following chapter will help you. From diagnosing and fixing the problem yourself to contacting Mankati support.

Setting and Calibration Problems

Q: Temperature parameters setting of several commonly used filaments

A: PLA: Nozzle temperature: 200-210°C Bed temperature: 45°C

ABS: Nozzle temperature: 230-235°C Bed temperature: 80-95°C

PVA: Nozzle temperature: 190-220°C Bed temperature: 45°C

Q: What are the relationships among the layer thickness, print speed and nozzle temperature?

A: The purpose of parameters interaction is: to leave enough time for the filament heating and cooling. Normally, if the layer thickness goes up, please reduce the printing speed and turn up the printing temperature properly (If the printing speed is over 60, turn up by 5°C ; if over 90, turn up by 10°C); Conversely, if the layer thickness goes down, please increase the printing speed and turn down the printing temperature.

Q: While printing the model with sealed roof and bottom, how to set optimal parameters to get high quality printing surfaces of the roof and bottom?

A: Increase the printing thickness of the top or bottom in the basic parameters.

Q: When printing the model with single layer thickness, how to set parameters?

A: Usually please keep the layer thickness of the model over 1mm; Meanwhile, choose the "Spiralize" function in "Expert" setting.

Notes:

1. Layer thickness setting in the software can't be over 0.8 times of the nozzle diameter (For example, when the nozzle diameter is 0.4mm, the layer thickness setting can't be over 0.32mm).

2. The maximum temperature of the nozzle is 300°C , and the build plate is 110°C . The parameters of materials from different manufacturers could be different, so it is better to consult the relevant material manufacturers about the temperature setting.

Q: There is the cracking on the model, or the joint problem among the layers.

A: 1. Build plate is not properly leveled. Please level build plate once again.

2. Material is not adhering to the build plate. Please stick the kapton tape to your build plate.

Q: The idle nozzle scratches the printing model while single color printing.

A: 1. The height of the idle nozzle is too low, turn it up.

Please move the build platform to the top of Z-axis, adjust the height of two nozzles according to the gap between the print head and the platform, tighten the idle nozzle by wrench and make it a little bit higher than the other one.

2. The printing flow is too high, causing the actual layer thickness is higher than the original setting, after a while, the object is over high and touch the main nozzle.

Please turn down the printing flow.

Hardware Problems

Feeder and Extruder Problems

Q: Nozzle clogged

A: 1. There is the impurity in the filament, which is stacked in the nozzle.

Please clean the nozzle by the needle; or take the nozzle off and clean inside of the nozzle by needle and drill.

2. The nozzle is overheated causing the filament carbonized inside.

Please clean the nozzle by needle; or take the nozzle off and clean inside of the nozzle by needle and drill.

3. The deformation of the nozzle hole occurred by external force.

Please replace the nozzle.

Q: The nozzle heating failed

A: 1. The heating tube connection is too loose.

Please tighten the connection.

2. Heating tube/ temperature controller/ power tube damage

Please check if one of the nozzles works well, but the other one cannot be heated, please troubleshoot by the following methods:

1) Open the machine case, and find out the connectors of two heating tubes, exchange the two connectors, heat the nozzle again. If the failed one can work now, it means one heating tube is broken and needs to be replaced;

2) If after exchanging the two heating tube, the failed nozzle still can't work, please exchange the heating tubes back, and try to exchange the connection of the two temperature controller, heat the nozzle again, if the failed nozzle can work, it means one temperature controller is broken and needs to be replaced;

3) If the above cannot solve the problem, it means it should be the problem of the power tube on the main board, please replace it.

Q: MAXTEMP/ MINTEMP

A: 1. Nozzle temperature is too high.

Please reduce the nozzle preset temperature.

2. Temperature controller damage.

Please replace the temperature controller.

3. LCD cable loose / LCD damaged.

Q: Print Head Stuck/Cannot move

A: 1. The axes lack of lubricating oil.

Please clean the axes and smear the lubricating oil evenly.

2. The deformation of machine framework caused by external force during the transportation.

3. The cable connection of X.Y motor is loose.

Please open the machine base, check and re-connect the cable.

4. The fixing screw of the timing pulley is loose.

Please check the fixing screw of XY axis, find out the loose one, and tighten it with crewdriver.

5. Synchronous belt drops or breaks.

Please change a new synchronous belt.

Q: Print head hit the framework

A: The connecting wires of limit switch get loose or the limit switch is broken.

Please check if the wires on the hit side, if the wires connecting limit switch and main board are loose. If yes, you need to pull out the wires and reinsert. If no, that means the limit switch is broken and need to be replaced.

Q: Filament Feeder Motor make intermittent noise

A: 1. The clamp of the feeding device is too tight..

Loosen the screw of the clamp..

2. Nozzle clogging, causing the filament not to be fed smoothly.

Please clean the nozzle with drill and needle..

Hardware Problems

Build Plate Problems

Q: Even the screws under build plate have been loose totally, the build plate still is far away from the nozzle (the gap between build plate and nozzle is too big to be adjusted).

A: Adjust Z axis limit switch on the back the printer, raise it a little bit; after adjusting, if problem still exist, the main cause is deformation is not enough of the hot bed's corners, it can be removed by tensile processing.

Q: The Z-axis movement of the build plate is abnormal.

A: 1. The top of the Z-axis screw is loose.

Please reinforced screw at the top of the Z-axis screw.

2. Z-axis motor loose connections.

Please open the base to re-plug the motor cables.

3. Z-axis screw need lubrication.

Please add oil to the Z-axis screw.

Printing Problems

Q: Problem with first layer printing

Q: The filament doesn't stick onto the build plate.

- A1: The gap between nozzle and build plate is too big.
- A2: The build plate has not been leveled yet.
- A3: Kapton tape has not been stuck on the build plate yet; or the non-sticky top layer of the Kapton tape is not removed.
- A4: The first layer's height is too small, recommended setting as ≥ 0.2

Q: Little filament sticks onto the build plate.

A: The gap between nozzle and build plate too small which may cause the nozzle clogged.

Q: No filament output from the nozzle.

A: The filament has not entered into the end of the nozzle yet when you feed the filament.

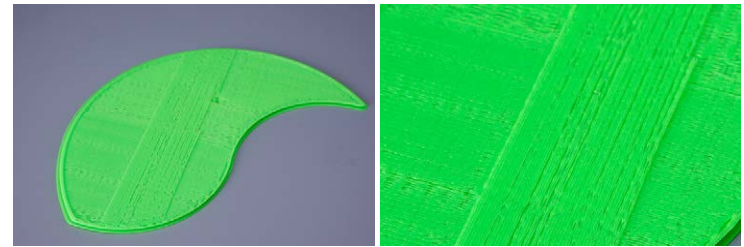
Q: The edge is lifted or warped, or the print cracked while printing with ABS filament

Please keep the build plate heated continuously; Change a new Kapton



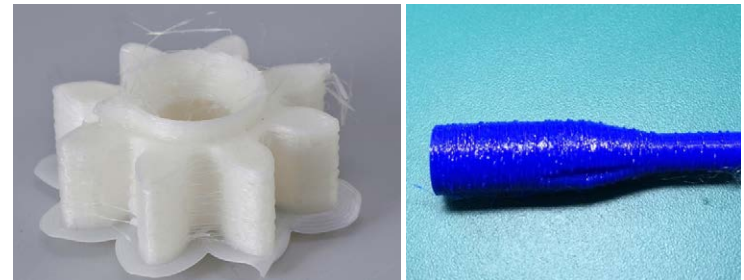
tape; Slow down the printing speed of first layer; Use the enclosure plate to keep the constant temperature while printing; Avoid air condition's blowing directly; Inside ventilation should not be so strong.

Q: Model surface is loose with crack



A: The layer thickness is too big; Or printing speed is too high (the nozzle temperature needs to do responded balance to have a faster printing speed); Or the temperature is too low; Or the wall thickness is too think; Or 3D model is not completely closed solid; Or the feeding motor's fixing screw is too loose; Or wrong choice of filament diameter; Or filament quality is poor; Or the filament gets stuck and could not be fed smoothly.

Q: Model has flow line



Printing Problems

A: Temperature is too high; Or the filament is carbonized or liquefied(PLA will get liquefied if over-high temperature, ABS will get carbonized if over-high temperature)
If still cannot solve the problem, please contact with our technical support team at support@mankati.com

Q: Model surface is unsmooth



A: Reduce the retraction travel; Or reduce minimal extrusion before retracting;

Note: Please reduce the retraction amount if retracting frequently.

Q: Model is easy to fall off

A: Having not used the Kapton tape or the Kapton tape loses stickiness; Or the model bottom is too small, you can use a hot melting glue gun to help; Adding support structure to support model; Using the feature in MankatiUM of Advance-Quality-Cut off object bottom.

Q: There is the joint dislocation between the parts while dual color printing.

A: Adjusting the feature of the relative position coordinate in MankatiUM: "Machine-Machine settings-Extruder 2".

Q: Hard to remove the support structure

A: Turn the direction of model properly in MankatiUM; The support density should not be set too high; Divide the model into different parts to print if necessary.

Q: Failed prototyping of small model

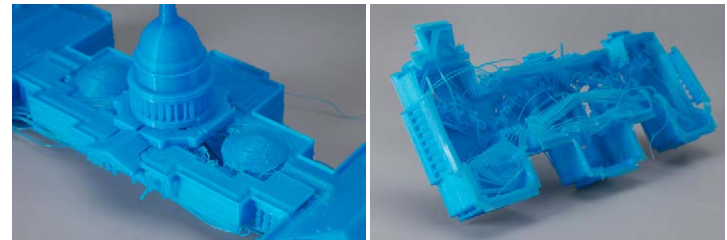
A: If printing a small model, the extruder will always move in a small space, so the heat will be concentrated in the model, and hard to be distributed.

That's why it is hard to print the very small model.

The solution is to print 3-4 pieces of the small model together. If so, the extruder will move among different models, leaving time for heat dissipation.

Q: Some part of the object unable to be modeling.

A: Alter the support functions in the MankatiUM software.

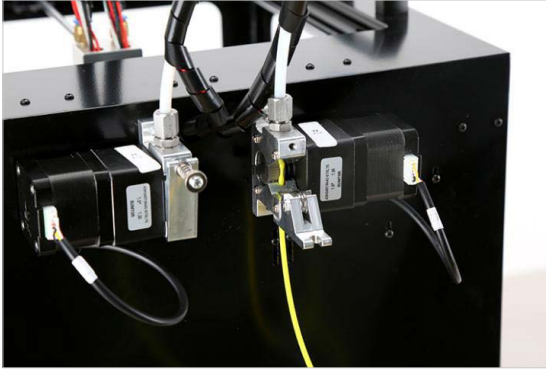


How To

How to deal with filament stuck in the feeder motor?

1. Sometimes the filament sticks at the feeder motor because of the slow filament unloading operation.

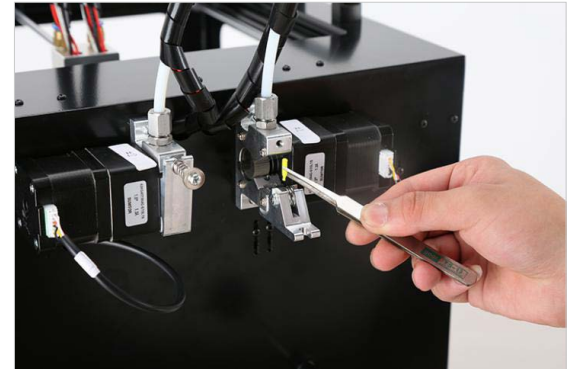
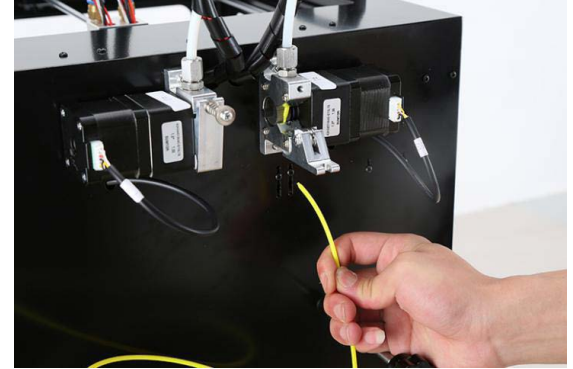
First: loose the black screw totally.



2. Cut off the thicker part.



3. Take the filament off with a tweezer.



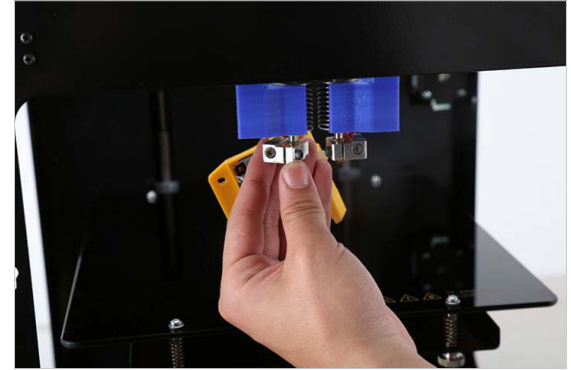
How To

How to disassemble the nozzle?

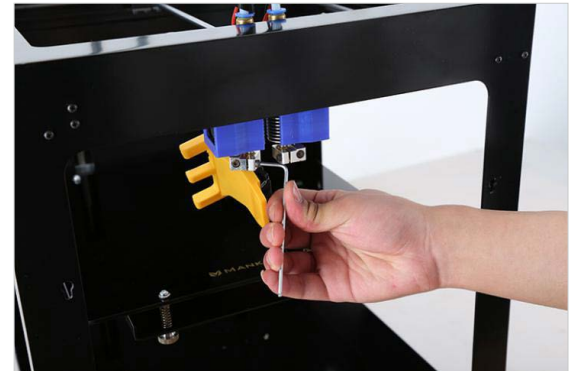
1. Heat the nozzle to 210 degrees, making residual filament inside the nozzle automatically flow out. Afterwards, lower the temperature below 50 degrees, then wait until the nozzle is totally cooled down.



2. Tweak the nozzle to the point as picture shows

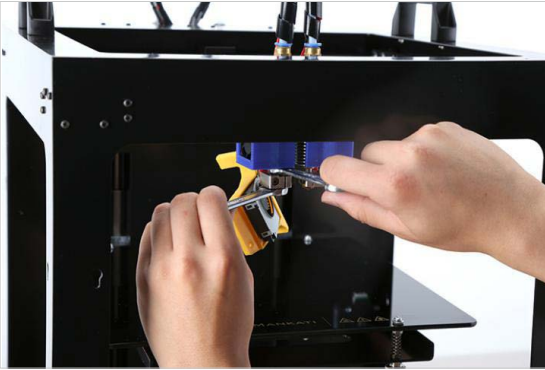


3. Loosen the screw with a L-shape wrench.



How To

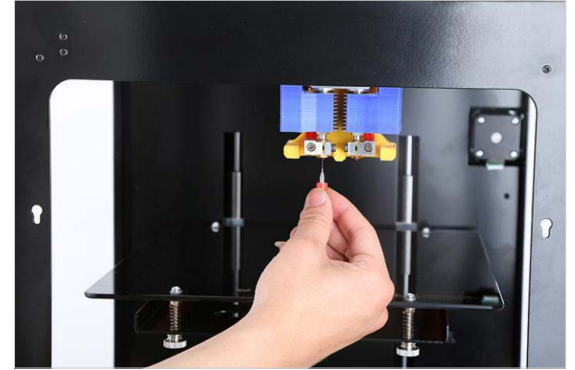
5. Use 2 spanners: one for fixing at the upper location, the other for loosen the nozzle under, then take it off.



How to clean the nozzle?

1. Heat the nozzle to 210 degrees, making residual filament in the nozzle automatically flow out.

2. After heating and filament extruding, clean nozzle with the drill in tool box. Pay attention not to break off it into the nozzle.



How To

How to change the fuse?

1. Find the new fuse(9) in the tool box.



2. Disconnect the power, turn the machine on the base, remove the power jack.



3. Remove the broken fuse, put in the new one.



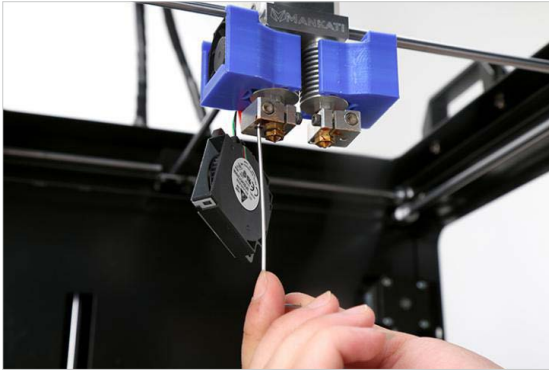
4. Reinstall the power jack into the machine. Done.



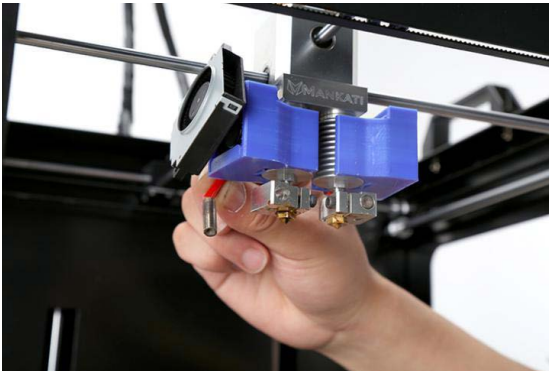
How To

How to change the thermostat (temperature controller) / heating tube?

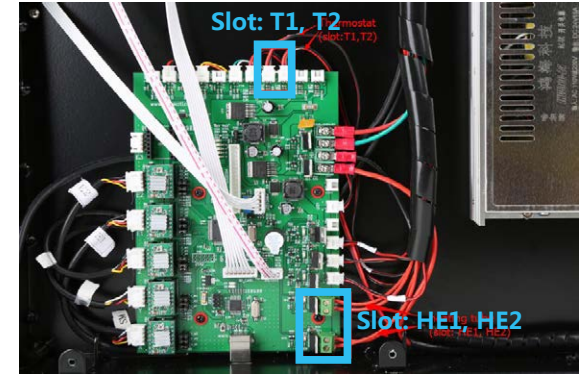
1. Loose the screws on heating blocks.



2. Push the thermostat /heating tube out.



3. For each wire of thermostat /heating tube, find their correct sockets (thermostat: T1,T2; heating tube:HE1, HE2) on the motherboard.



4. Replace the faulty one with the spare one in your tool box.

Note: The black&red wire is thermostat, the red wire is heating tube.



How To

How to tighten the belts?

1. Loosen the screw on the synchromesh gear which drives the corresponding belt you want to tighten.



2. Hold the belt's end reaching out from the black sliding block with a clamp.



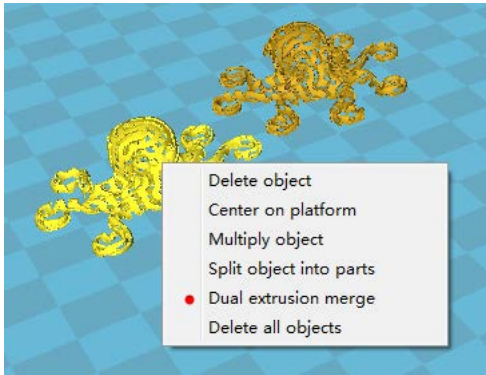
3. Loosen the screws on the sliding block with the L-shape wrench, then use a clamp to tighten the belt properly. Finally, lock the screws back on sliding block and synchromesh gear.



How To

How to print with double colors in one object?

1. Load the model into MankatiUM, and then right-click model, choose "Dual extrusion merge"



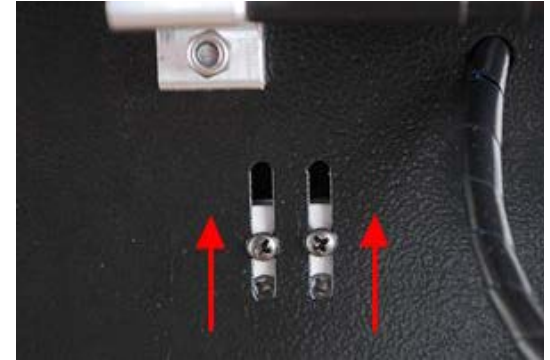
2. Then you can set parameters for the main nozzle and vice nozzle.



How to deal with issues of "faulty distance between nozzle & plate"?

How to deal with issues of "build plate still inclines or faulty distance between nozzle&plate after plate leveling"?

Loosen the screws and raise the Z limit stopper on rear side a little higher..



CONTACT

Support

Support@mankati.com

Skype ID: Support_mankati

Tel: +8621-67618178

(9:30-18:00 Beijing Time, Monday - Friday)

Further more, feel free to share your 3D printing experience and discuss with makers all over the world on Mankati Forum:

Forum: <http://forum.mankati.com>

For more videos of printing operation tutorials and sharings, please subscribe to our

Youtube Channel: Mankati 3D Printer