stomdevice

you can
WANHAO

Duplicator 9 MARKI

INSTRUCTION MANUAL



stomdevice

Important

Getting the most out of your 3D Printer:



Read the manual carefully
It will help you get set up with ease



Visit www.wanhao3dprinter.com
We've gotten additional support videos and guides



Google Forum

Join Wanhao Technical Forum for online support
https://groups.google.com/forum/#!forum/wanhao-printer-3d



Email support@wanhao3dprinter.com
We're always happy to help

Warranty Note

If you experience any issues with this product, or it's performance is not what you had expected, please contact us at WANHAO before returning the item to the store.

It is likely that we can resolve any problems for you via phone or email.

We can be reached through your local distributor. Or on

Phone: +86-571-23290996(Mon-Sat; 8:30am-5:00pm)
Email: support@wanhao3dprinter.com
Web: www.wanhao3dprinter.com

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

stomdevice

WANHAO

3D Printer

Warranty Details

The product is guaranteed to be free from defects in workmanship and mother board for a period of 12 months from the date of purchase. The extruder is guaranteed to be free of defects for 90 days. Printing with materials other than WANHAO FILAMENTS which require temperatures above 250 °C or which are different from those recommended on the website www.wanhao3dprinter.com, will result in the immediate loss of the Warranty. Defects that occur within this warranty period, under normal use and care, will be repaired, replaced or refunded at our discretion. The benefits conferred by this warranty are in addition to all rights and remedies in respect of the product that the consumer has under the China laws. In case you purchase from WANHAO distributor, your warranty shall be covered by distributor's Warranty and honored by WANHAO distributor too.

Our goods come with guarantees that cannot be excluded under the Chinese Consumer Law.
You are entitled to a replacement or refund for a major failure and for compensation
for any other reasonably foreseeable loss or damage. You are also entitled to have the goods repaired or replaced if the goods fail to be of acceptable quality and the failure does not amount to a major failure.

AFTER SALES SUPPORT
☎ 86-571-23290996
support@wanhao3dprinter.com
MODEL: Duplicator 9

1
YEAR WARRANTY

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Safety Instructions

Before you get started, please read these important safety instructions.

CAUTION: The WANHAO 3D Printer generates high temperatures and includes moving parts that can cause injury. Never reach inside of the WANHAO 3D Printer while it is in operation, and allow time for the printer to cool down after operation.



HOT SURFACE



MOVING PARTS

Vapours/fumes may be irritating at operating temperatures. Always use the WANHAO 3D Printer in an open, well ventilated area.

Do not leave the WANHAO 3D Printer unattended during operation

Contact with extruded material may cause burns. Wait for printed objects to cool before removing them from the build platform.

Always take care when using the Scraper. Never direct the Scraper towards your fingers.

- The printer must not be exposed to water or rain, or damage will occur.
- The printer is designed to be used with ambient temperature ranging 15°C - 30°C, and humidity ranging 20% - 50%. Operating outside these limits may result in low quality models.
- In case of emergency, turn off the 3D printer at the power outlet.
- It is recommended that you use eye protection when cleaning/sanding printed models to avoid small particles contacting eyes.

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Contents

Warranty Details 4
Safety Instructions 5
Parts List 8
Technical Specifications 9
Product Overview 10
Printer Front View 10
Printer Rear View 11
Quick Guide 12
Unboxing and Assembly 12
Wiring instruction 15
Introduction 19
1.1 What is 3D Printing? 19
1.2 From 3D Design to Printed 3D Object: 19
1.3 Good 3D Printing Practices 21
Installation 22
1.4 Control Panel 22
1.4.1 Main Display 22
1.4.2 Menu Map 23
1.4.3 Default Configuration 24
Resuming after power failur 25
Setup 26
2. Levelling the Heated Print Bed 26
3. Filament 27
3.1 Filament Types 27
3.2 Inserting Filament: 28
3.3 Removing Filament: 29
3.4 Storage: 29
4. Using the Micro SD Card 29
4.1 Loading files from your Computer: 29
4.2 Printing from the Micro SD Card: 30
5. Removing 3D Objects from the Bed 30

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Using Cura 31
6. Creating a GCODE File using Cura 31
6.1 Installing Cura 31
6.2 Adding your WANHAO 3D Printer to Cura 32
6.3 Loading a 3D Design into Cura 32
6.4 Viewing and Positioning your 3D Model in Cura 32
6.5 Quick Print Settings 35
6.5.1 Exporting your GCODE File: 36
6.6 Advanced Print Settings 36
6.6.1 Basic Tab Options 36
6.6.2 Advanced Tab Options 39
6.6.3 Plugins Tab 42
6.6.4 Start/End GCODE Tab 42
6.6.5 Expert Settings 43
Servicing 48
7. Servicing your 3D Printer 48
7.1 Replacing the Print Mat 48
7.2 Nozzle Cleaning 48
7.3 Routine Component Check 49
7.4 Storing 49
Finishing 50
8. Finishing Techniques 50
8.1 Useful Tools/Equipment 50
8.2 Removing Support Material 50
8.3 Sanding 51
8.5 Surface Coating 51
Troubleshooting 52
9. Troubleshooting 52
9.1 Printing Problems 52
10.2 Connectivity Problems 53
10.3 Other Problems 54
10. Useful Terms 54
Appendix 56
References 56
Repair and Refurbished Goods or Parts Notice 58

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Parts List

1 x WANHAO 3D Printer
1 x Filament Spool Holder
1 x Filament Spool Stand
1 x Power Cable
1 x USB Cable
1 x 8GB SD Card
2 x Hex Screw Driver
1 x Hex Key Set
6 x Hex Bolts
2 x Plastic belts
1 x 1.75mm PLA Filament
1 x Instruction manual
1 x Getting Started Guide
1 x Needle for unclugging nozzle





AFTER SALES SUPPORT
86-571-23290996
support@wanhao3dprinter.com

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

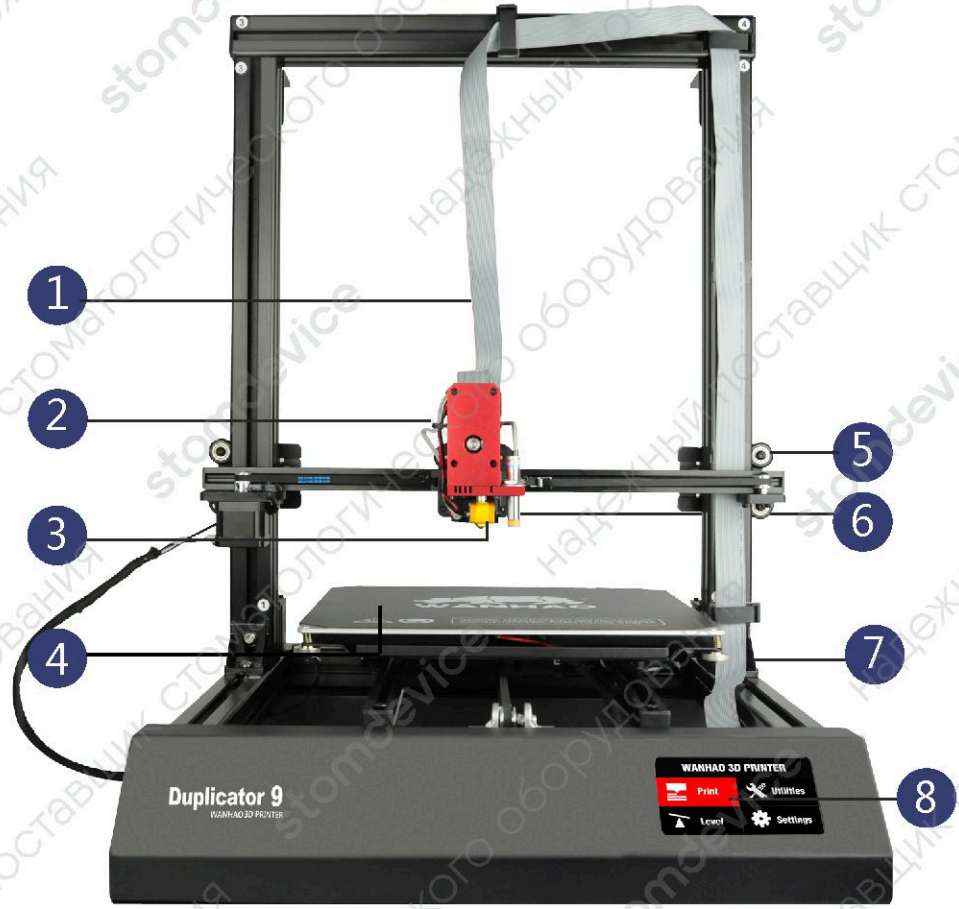
Technical Specifications

Primary Attributes of the Technology	PLA Printer, Entry Level Printer. Fully Assembled. Developed from Prusa i3.
Software	CURA, Repetier Host, Simplify 3D
Extruder System	MK10 Extruder with Full Metal Hot End
Material Support	PLA, PETG, PVA, Nylon, And Most Exotics Any Material Melting <=250℃
Max Printable Area	300*300*400mm/ 400*400*400mm / 500*500*500mm
Filament Diameter	1.75mm
Max Printing Speed	70mm/s
Case Material	Aluminum Case
With Insulated Cover?	Insulated Tent Optional.
Extruder Quantity	1
Motor Accuracy	X 0.012mm Y 0.012mm Z 0.004mm
Net Weight(kg)	20kg
Gross Weight(kg)	22kg
Material Cost	USD12-34
Bed Leveling	Auto leveling and manual leveling
Packing size(cm)	60*50*35cm
Plateform Holder	Steel Frame Holder
Rail Locking System	Steel Bolt Locking Frame
Layer Thickness	100 micron- 400 micron
Platform	Aluminium Platform w/Heating
Resume Printing?	Can resume printing after power failure
LCD Display	English/Chinese/ Can be customized for any language
Power Supply Unit	Inside 110 AND 250V(optional), 50/60Hz, 4.0A (input), and takes standard IEC cable
Typical Application	•A perfect printer for educators who intend to use the printers in technical educatioin. •Great printer for DIY hobbist.

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Product Overview

Printer Front View



1. Single Extruder Cable
2. Near End Filament Feeding System
3. Full Metal Hot End
4. Heating Bed
5. Anti-backlash System
6. MK10 Full Metal Extruder
7. Locking Level System
8. Touch Screen

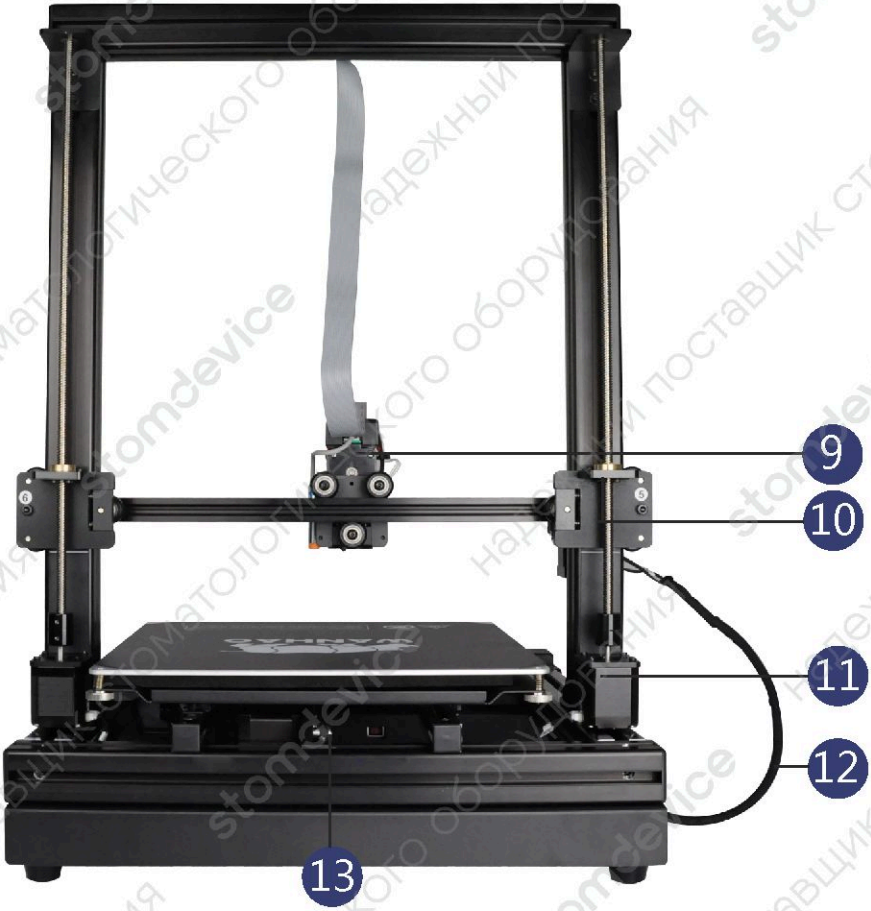
AFTER SALES SUPPORT

86-571-23290996 support@wanhao3dprinter.com

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Product Overview

Printer Rear View



9. Anti-backlash System
10. Z Axis Lead Screw
11. Super Precise Nozzle
12. Cable Chain
13.Y Axis Bearing

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Quick Guide

Unboxing and Assembly

1



Take out all components from the box and place them on the flat ground.

1-Base
2-Z axis Tower Pillar*2
3-X axis rail
4-X axis gantry
5-MK12 Extruder

2



Place the Base on it's side. Find the Z axis pillar with sticker No.2. Slide the No.2 pillar into the base vertical locker. Then tighten the bolt at bottom of base.

3



Tighten the locking bolt clockwise. Then install the other Z axis pillar onto the Base.

4



Loose the two locking bolt on No.3 and No.4 Z axis pillar by around 5mm. So there will be sufficient space for X axis pillar to slide in.

5



Slide the X axis pillar into the No.3 and No.4 bolts. If there's not enough space, you may need to re-loose the locking bolt.

6



To tighten the locking bolt through the hole on top of the X axis pillar. Lock both No.3 and No.4 pillar.

7



Insert the plastic cover onto the two ends of X axis pillar.

Note: Please double check the Z axis and X axis pillars. If any bolt not secured. You need to secure the screws to ensure the gantry are steady and tight.

8



Rotate the cylinder holder for a bit but not too close. Tight enough that can not slide into pillar freely. This is also same to the other side with stickers No.6.

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Quick Guide

9



Mount the X axis rail. Align the right and left side holders first. Please make sure the rail is located in the middle of the X axis. You may need to adjust the distance to both ends through the cylinder holders.

10



Then tighten the bolts on both end of No.5 and No.6.

11



Mount the MK12 extruder onto the X axis holder. Make sure the locking holes match to the holder holes.

12



A total of 4 bolts need to be tightened, but the first bolt should not be too tight until the other three bolts are mounted, then tighen all the bolts together.

13





Turn the couple to lower the two Z axis in order to level the two Z axis at same height.

Note: The ruler or height gauge(Horizontal line) would provide more accurate height for both two Z axis. The two axis leveling base need to be at same height to match the horizontal line marked on the both Z axis Pilar.

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Quick Guide

14



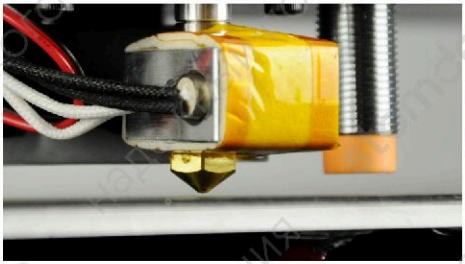
Please go to home page and select " **LEVEL---AUTO LEVEL**", this function would auto level the bed.

NOTE: Once you have done the auto leveling. The printer would memorize the leveling data for the next printing. You don't have to level the printer every time, unless you feel the printer is not not leveled. Auto leveling is for general printing purpose, if you need some special layer thickness or special purpose, you may need to adjust the 4 bolts under bed to achieve more precise distance.

Manual leveling. If the Auto leveling can not meet your needs, you may try manual leveling. Click **"LEVEL—MANUAL LEVEL"**, the instruction would guide you to check 4 bed points.

Click P1/P2/P3/P4 separately.
The distance between the Extruder and bed should be 0.1mm or roughly the thickness of an name card. The paper should have very Slight friction as you moves it under the Extruder but should still slide freely.

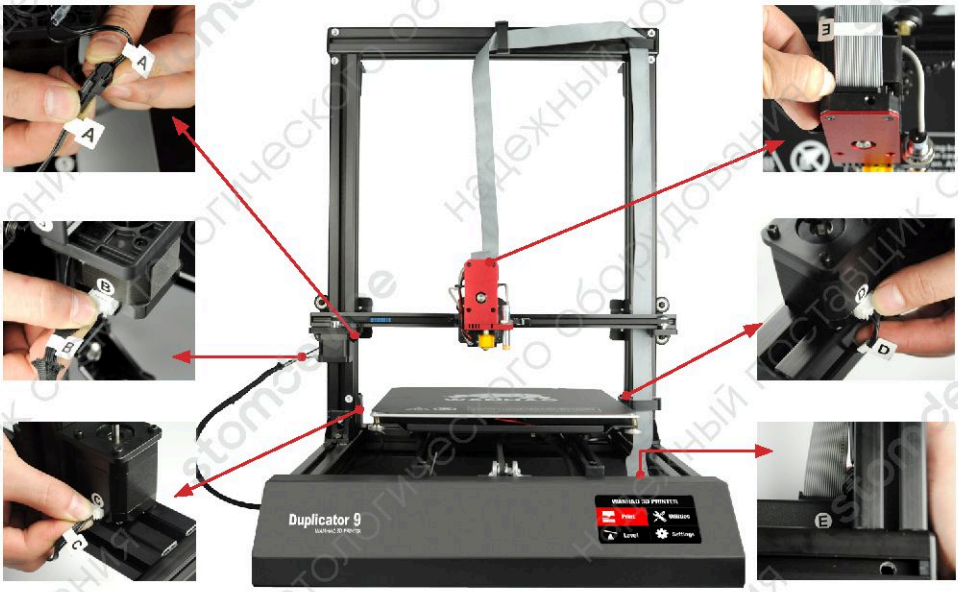




https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Quick Guide

Wiring instruction



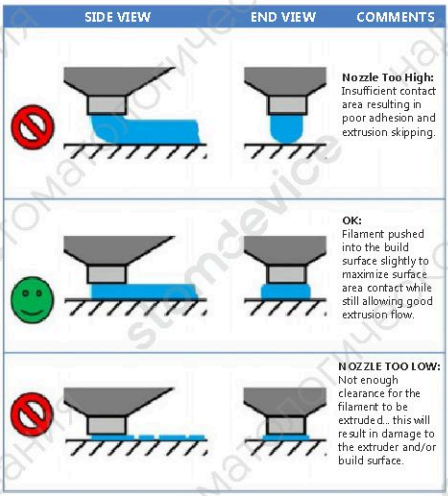
1. Remove the outer plastic nut from the Filament Spool Holder, then insert the threaded into the large hole of the Filament Spool Holder Stand. Tighten the plastic nuts on each side to secure.
2. Position the Filament Spool Holder Stand on top of the Extruder Tower, then insert the bolts and tighten with the Hex Wrench.
3. Ensure the power switch is in the OFF position, then insert the Power Supply Cable into the left of the Control Box. Turn on the power outlet, then turn on the 3D Printer, using the switch on the back of the Control Box .
4. Wait for the printer to initialise, with **Printer Ready**.

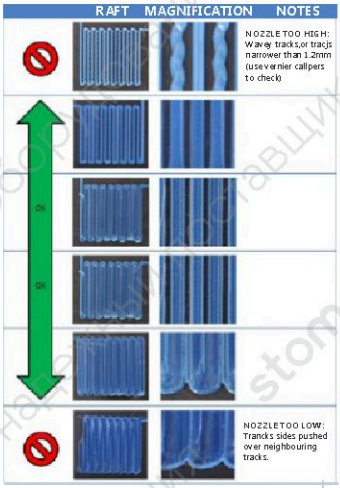
https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Quick Guide

5. Move the Extruder to each corners of the print bed then you can adjust the bolts under each corner. repeat the process. Move the Extruder to the centre of the Print Bed and measure the gap with a sheet of paper. Keep adjusting and testing until you find ensure the proper distance between nozzle and print bed.

CHECKING CHART





NOTE: Before proceed anything ,use a soft fabric to Wipe off the dusts from the building plate surface and clean it. Not leveled print bed may damage the Printing bed Mat during Auto Bed Leveling.

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Quick Guide

6. You can now select one of our pre-loaded files below by navigating to, and selecting the file name:

				
File Name:	Unicorn	M3 Thumbscrew	Vase	WANHAOBot
Size (LxWxH):	47x19x51mm	19x19x4mm	50x50x100mm	26x34x15mm
Infill:	100%	100%	0%	20%
Resolution:	0.2mm	0.1mm	0.2mm	0.1mm
Print Time:	50min	6min	1hr 43min	36min

7. Your printer will now commence heating the Extruder and Heated Print Bed to the correct printing range, before returning to the home position and commencing the print

8. Once your printer has completed printing the file, the nozzle and Heated Print Bed will commence cooling down. Once cooled, turn off your 3D printer and carefully remove the 3D printed object using the supplied Scraper and gently lifting the object with your hand



Always take care when using the Scraper. Never use excessive force or scrape towards your hand.

NOTE: Always keep the Scraper as level as possible. Scraper can damage the print mat if not used correctly

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

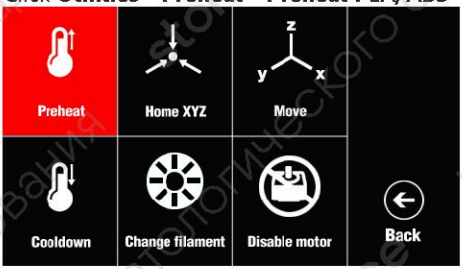
Quick Guide

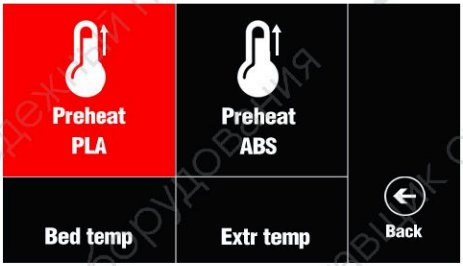
IMPORTANT NOTE: Levelling the bed is an extremely important step and if it is not done correctly every time, your prints will fail or be of poor quality.

PREHEAT EXTRUDER

To reduce the heating time before printing, you may preheat extruder while you prepare the printing file.The heating bed and extruder would reach the target setting and waiting for your further command.

Click **Utilities—Preheat—Preheat PLA/ABS**

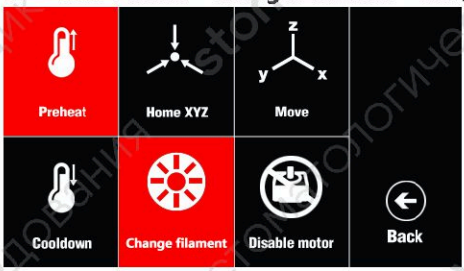


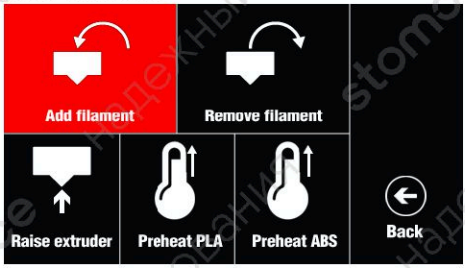


CHANGE FILAMENT

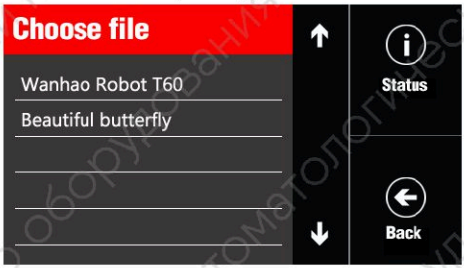
For your first time printing, you need to load the filament. Or in case you want to change the color of print, you would need to change the filament.

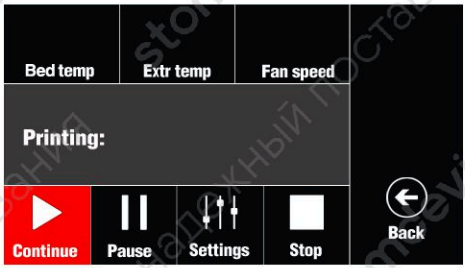
1. Click **Utilities—Change Filament— Add/Remove Filament**





2. You may need to raise the extruder, If the extruder is too near the bed which block the filament from extrusion. Then click **RAISE EXTRUDER**.
3. Or you may need to change the loading/unloading temperature. Here is option Preheat PLA which would heat extruder to 210'C. And Preheat ABS which would heat extruder to 230'C.
4. Turn on the 3D printer. Click on the **PRINT---CHOOSE FILE**. Then the printer would start printing.





https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Introduction

1.1 What is 3D Printing?

3D Printing is a process whereby a real object is created from a 3D Design. Fused Filament Fabrication (FFF) is one of the most common technologies used by 3D Printers to achieve this. FFF works on an 'additive' principle by laying down material, such as PLA or ABS filament, in layers to create a 3D object.

1.2 From 3D Design to Printed 3D Object:

3D Design → Slice to GCODE → Setup 3D Printer → 3D Printing Process → 3D Printed Object



3D Design

Having a 3D Design is the first step to creating a 3D Object. There are a number of free 3D Designs that can be downloaded and printed for personal use. Websites such as Thingiverse.com, Pinshape.com and YouMagine.com are a good place to start, but there are many more available!

Alternatively, you can create your own 3D Design using 3D Modelling Software. Programs such as SketchUp or Blender allow this, however this can take some practice before successfully modelling your 3D Design. After you've finished your 3D Design, you can export it as an .STL file, to be imported into a 'Slicer' program such as Cura.



Slice to GCODE

A 'Slicer' converts a 3D Design (usually in .STL or .AMF format) into individual layers. It then generates the machine code (such as GCODE) that the 3D Printer will use for printing.

We recommend Cura, as it uses a simple interface and includes a large amount of support material.

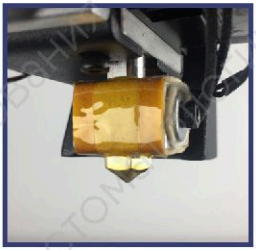
https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Introduction



Setup 3D Printer

It's important to set up your printer before every print. We've included some steps on page 20 in section 1.3 'Good 3D Printing Practices' for you to quickly refer to each time.



3D Printing Process

Now that you've loaded a GCODE file onto the printer, it will commence heating up the Extruder and Heated Print Bed, then start printing. While printing, the nozzle will move along the X and Y axis as each layer is printed. This can take some time, depending on the size of the 3D model, print speed of the 3D printer and resolution of each layer.

Here is an example of how size, infill, and resolution can effect printing time:

				
Dimensions:	62.5 x 25.75 x 67.75mm		70 x 70 x 140mm	
Infill:	20%	**0%**	20%	20%
Resolution:	2mm	2mm	2mm	**1mm**
Print Time:	53min	39min	6hr 40min	13hr 12min
Filament Used:	4m	2.54m	45m	45.67m

3D Printed Object

Once the printer has completed all of the layers, you will have a 3D Printed Object. The results are certainly worth it, with 3D models ranging from simple to complex.

The WANHAO 3D Printer is compatible with a broad range of filament types and is capable of printing 3D models ranging from small and simple to complex objects with moving parts. What you print is limited only by your imagination.

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

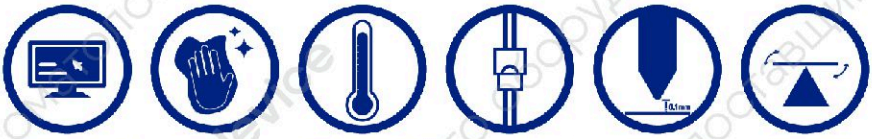
Introduction

1.3 Good 3D Printing Practices

Please refer to these steps during every print to ensure the best quality.

Before Printing

1. Ensure your GCODE file is correctly setup, filament diameter is set to to 1.75mm and correct temperature settings for the material you are using (refer to table 3.1, page 27)
2. Check all cables and connectors are connected
3. Check that Heated Print Bed is level and approximately 0.1mm from the nozzle
4. Check that Heated Print Bed is clean and dust free and nozzle does not have excess material stuck to it



Always check Gcode file settings match filament and printer

Always check Print Mat is clean and there is no residual plastic

Always check Extruder & Print Bed Temperatures are set for correct filament type

Always check all cords are connected

Always check Extruder distance is 0.1mm from Print Mat

Always check Print Bed is level

During Printing

1. Closely monitor the first layer being printed – Most problems will occur in the first layer. If problems occur, stop the 3D Printer at the power outlet
2. Do not leave the 3D Printer unattended while in operation
3. Keep hands clear while machine is operating

After Printing

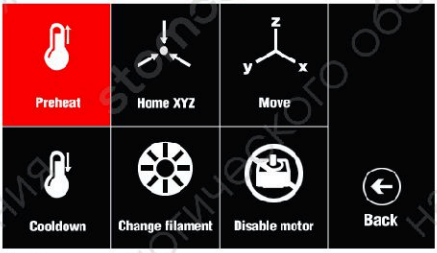
1. Wait for the Heated Print Bed and Extruder to cool down
2. Carefully remove 3D Object. Never use Scraper towards your body. Keep Scraper evenly flat when scraping to avoid damage to the Heated Print Bed. Damaged Heated Print Bed will affect adhesion for future prints
3. Store 3D printer in a clean, dry environment

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Installation

1.4 Control Panel

1.4.1 Main Display





System:

Home X Y Z: Returns Extruder and bed to home position using stop switches.

PID: To Change PID value.

Motor: To move X, Y, Z, Extruder motor

X/Y/Z/E Position: Precisely adjust the location of a specific axis

Factory Settings: To return to default factory setting

Tool:

Preheat: Extruder and bed temperatures set and active
ABS: Sets Extruder to 245°C and Heated Print Bed to 90°C and active
PLA: Sets Extruder to 215°C and Heated Print Bed to 60°C and active
BED: (Current °C / Set °C)
E-T: (Current °C / Set °C)

Move:

Cool Down: To stop heating and cooling the extruder
Home All: Returns Extruder and bed to home position using stop switches.
Disable Drive: Motors are disabled and free to be moved by hand
Disable Power: Motherboard is disabled, no sensor and motor has power.

Print: To select the printing file and active printing
SD: Enter SD card file.
UP: Select the upper file
Down: Select the lower file
Printing: 3D Object is being printed, % displayed
Printing Speed: To change the printing speed. Default setting is 100%.
Ead Temp: Extruder set temperature
Bed Temp: Bed set temperature
FAN Speed: Fan set speed. Default setting is 100%

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Installation

1.4.2 Menu Map

Quick Settings:

- **Home All:** Returns Extruder and bed to home position using stop switches
- **Speed Mul.:** Default 100%. Motor speed multiplier. Speed of the motor is multiplied by this rate
- **Flow Mul.:** Default 100%. Printer flow multiplier. Amount of filament extruded is multiplied by this rate
- **Preheat PLA:** Sets Extruder to 215°C and Heated Print Bed to 60°C
- **Preheat ABS:** Sets Extruder to 245°C and Heated Print Bed to 90°C
- **Cooldown:** Sets Extruder and Heated Print Bed to 0°C
- **Disable Stepper:** Disables all motors so they are free to be moved by hand
- **Print File:** If an SD Card is mounted, files can be quickly accessed here.

Position:

- **Home All:** Returns Extruder and bed to home position using stop switches
- **Home X / Y / Z:** Returns specific axis to home position using stop switches
- **X / Y / Z Pos. Fast:** Quickly adjust the location of a specific axis
- **X / Y / Z Position:** Precisely adjust the location of a specific axis
- **Extr. Position:** Adjust the Extruder driving motor to feed, insert or remove filament

Extruder:

- **Bed Temp:** Manually set the bed temperature 30°C and 70°C
- **Temp. 1:** Manually set the Extruder temperature between 170°C and 240°C
- **Extruder 1 Off:** Quickly return the Extruder temperature to 0°C
- **Extr. Position:** Adjust the Extruder driving motor to feed, insert or remove filament
- **Set Origin:** Returns Extruder position value to 0mm

Fan Speed: Adjust the Extruder cooling fan to your desired percentage

SD Card:

- **Print File:** Select a file to be printed from the MicroSD Card
- **Mount Card:** If inserting the MicroSD Card after the 3D printer has initiated, card can be mounted using this setting
- **Unmount Card:** Safely unmount the MicroSD Card
- **Delete File:** Select a file to be deleted from the MicroSD Card

Debugging:

- **Echo:** Off. Echo GCODE steps to confirm what has been received
- **Info:** On. Show printing information on the main display
- **Errors:** On. Show errors on the main display
- **Dry Run:** Off. Perform the GCODE steps without extruding any filament

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Installation

1.4.3 Default Configuration

The following settings show the default configuration values of the WANHAO 3D Printer. We've set these to provide optimal quality for your 3D printed models, however, if you wish to adjust these settings, we recommend this to be done by only highly experienced users, as print quality and 3D printer durability may be affected.

• **General**
Baudrate: 115200
Stepper Inactive: 360s
Max Inactive: 0s
• **Acceleration**
Print X: 1000
Print Y: 1000
Print Z: 100
Move: X
Move: Y
Move: Z
Jerk: 20.0
Z-Jerk: 0.3
• **Feedrate**
Max X: 200
Max Y: 200
Max Z: 2
Home X: 40
Home Y: 40
Home Z: 2
• **Extruder**
Steps/mm: 96.0
Start FR: 20
Max FR: 50
Accel: 5000
Stab. Time: 1
Wait Units: 0mm
Wait Temp.: 150C
Control: Dead Time
DT/PID P: 7.00
PID I: 2.00
PID D: 40.00
Drive Min 40
Drive Max 230
PID Max: 255

- **Store to EEPROM:** Save configuration settings to the internal storage, to be re-loaded on reboot
- **Load f. EEPROM:** Return configuration settings to the last reboot

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Installation

Resuming after power failur

If you encounter a power failure, for any reasons such as power outage or accidentally cut the power cord during printing. Don't worry, D9 comes with printing resume feature. After power intrupption you can power on the printer and select [Continue] and you can proceed with the unfinished model.









NOTE: For resume printing Don't move the nozzle until the printer proceed for print, otherwise the printing postion would not be as accurate as it should be. Wait for the nozzle to be heated, and do not touch the nozzle with your hand, which may cause you to burn.

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Setup

2. Levelling the Heated Print Bed

It is important to ensure the Heated Print Bed is level and set to approximately 0.1mm distance prior to printing.

1. Press Tool → Move → Z Axis → + to Riaise the extruder.So that you can easily access the heated print bed.
2. Turn off your 3D Printer
3. Check the Z axis level. The X rail should be parallel to the Heated Print Bed. If it is not, manually rotate the right hand side Z axis stepper motor to level it
4. Squeeze each corner of the Heated Print Bed, and tighten each wing nut to reduce the distance between plates to a minimum
5. Wipe the top of the Heated Print Bed with a clean, lint-free cloth to ensure no dust particles or other objects are on the surface
6. Turn on the 3D Printer, using the switch on the back of the Control Box
7. With the printer initialised, press Tool → Home All
8. Navigate to and select . **Press Tool > Home All**. Your 3D printer will now try to align to the 'home' position, on the front left corner of the Heated Print Bed
9. Once aligned, turn off the 3D printer
10. With the printer powered off, move the Extruder to one corner of the Heated Print Bed to check the distance between these components
11. Rotate the wing nut to raise the Heated Print Bed. A good distance is 0.1mm, and will allow a sheet of paper to slide freely with slight friction between the Extruder and Heated Print Bed
12. Move the Extruder to each of the other corners and repeat the process. Move the Extruder to the centre of the Heated Print Bed and test the gap with a sheet of paper. Keep adjusting and testing until the proper gap is present at all points

IMPORTANT NOTE: Levelling the bed is an extremely important step and if it is not done correctly every time, your prints will fail or be of poor quality

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Setup

3. Filament

The WANHAO 3D Printer uses high quality PLA and ABS filaments. Only use quality WANHAO filaments as using generic filaments may damage your 3D Printer, affect the print quality and may void your warranty. Extra filament including multiple colours and types are available from the online store at **id.aliexpress.com/store/120824**

For beginners, we recommend using PLA materials as they are easy to print and does not produce fumes when printing. Before using other types of filaments, ensure you understand the advantages and disadvantages of other filaments and their proper usage.

3.1 Filament Types

Filament Type	PLA	ABS	PVA	HIPS
Extruder Temp:	195ºC	210ºC	185ºC	220ºC
Heat Bed Temp	60ºC	90ºC	50ºC	60ºC
Official Name:	Polyactic Acid	Acrylonitrile Butadiene Styrene	Polyvinylv Alcohol	High Impact Polystyrene
Printing Level:	Basic	Advanced	Advanced	Advanced
Advantages:	• Bioplastic • Non-Toxic • Less warping issues • Hard or soft/ flexible variants • High print speed	• Smooth finish • Durable • Ideal for mechanical parts	• Water soluble • Non-Toxic	• Lightweight • Cost effective • Dissolvable in Limonene
Disadvantages:	• Low heat resistance • Easier to break than ABS • Requires thicker walls than ABS	• Petroleumbased • Nonbiodegradable • Heated Print Bed required • Fumes when printing • Sunlight deterioration	• Sensitive to high heat. **Do not heat over 190**	• Only available in white
Finishing:	• Sanding possible • Limited gluing	• Easy sanding • Easily soluble in acetone	• Gluing Only	• Gluing Only

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Setup

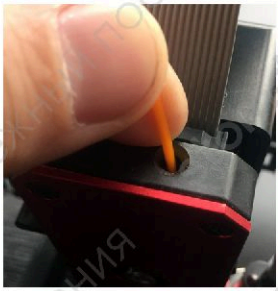
Alternative types of PLA

These PLA Alternatives require advanced knowledge of the 3D Printer. Please visit **www.wanhao3dprinter.com** for details of filament types and parameters.

NOTE: Only use WANHAO Filament with your 3D Printer, using other filaments may damage your 3D Printer and may void your warranty.

3.2 Inserting Filament:

1. With the printer initialised, press Tool → Preheat
2. Navigate to, and select Preheat PLA or Preheat ABS - depending on the material in use
3. Wait for the 3D printer to automatically return to the home screen, after approximately 15-30 seconds
4. Check the temperature of the Extruder and plate on the LCD. When the temperatures are within a few degrees of the target, you are ready to insert the filament



NOTE: Preheat PLA/ABS will still be shown, even after temperatures have been reached

5. Carefully insert the included filament sample into the hole on the Extruder as shown
6. While holding the filament in place with one hand, press the Dial to enter the menu system
7. Navigate to and select Tool → Move → E Axis → + Extruder > Extr. Position.
8. Gently push down on the filament while turning the Dial clockwise. You will feel the filament being pulled into the Extruder. Continue until a few centimetres have extruded out of the nozzle.
9. Carefully clean the extruded filament off the Heated Print Bed and nozzle

WARNING: Extruder and Heated Print Bed will still be hot

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Setup

3.3 Removing Filament:

1. With the printer initialised, press the Navigation Dial to enter the menu system.
2. Navigate to, and select **Tool** > **Preheat PLA** or **Preheat ABS** depending on the material.
3. Check the temperature of the Extruder and plate on the LCD. When the temperatures are within a few degrees of the target, you are ready to remove the filament.

NOTE: Preheat PLA/Preheat ABS will still be shown, even after temperatures have been reached.

4. Depress the Extruder spring lever. Push down on the filament gently so that it starts to extrude from the nozzle.
5. When filament is extruding, quickly, but gently pull the filament upward. If it is done correctly, the end of the filament will pull out into a thin hair like strand.

NOTE: The aim is to sever the filament where is it soft (inside the nozzle). Swift upward action will ensure that the filament does not cool down before exiting the nozzle inlet. If the filament gets stuck, please see the troubleshooting section.

3.4 Storage:

Both PLA and ABS filaments are sensitive to moisture. If stored in a humid environment, the quality of the filament can deteriorate, with a tendency to bubble and spurt from the nozzle tip. This will affect print quality, so when storing PLA or ABS for an extended period of time, follow these tips.

- Store in a sealed, air tight bag/container. We recommend a vacuum seal bag
- Store with a silica gel (moisture absorber) pack. One is included with each filament spool

4. Using the SD Card

4.1 Loading files from your Computer:

1. Insert your SD Card into a card reader or memory card slot and connect to your computer (refer to your computer manual for further instructions if required).
2. The Micro SD Card will appear as a folder (refer to your computer manual for further instructions if required).
3. Copy your converted GCODE files to the SD card, then safely remove.

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Setup

4.2 Printing from the Micro SD Card:

1. Insert the included SD card into the slot on the side of the Control Box, ensuring that the metal contacts are facing upwards
2. Navigate to and select **SD Card** > **Print** > **Print File**
3. Navigate to your desired GCODE File, then press the Dial to commence printing
   Your WANHAO 3D Printer will now heat up the Extruder and Heated Print Bed before printing your 3D object

NOTE: If pressing print shows no files, chances are the SD Card has not been mounted. Remount your SD Card into the side of your Control Box

NOTE: If Extruder temperature is set at 0°C after attempting to print a file, the imported GCODE may have some compatibility issues with the printer. Manually change the Extruder temperature using **Extruder** > **Temp 1** or **Quick Settings** > **Preheat PLA** or **ABS** depending on material used

5. Removing 3D Objects from the Bed

When the WANHAO 3D Printer has completed printing your 3D printed object, it's time to remove the object from the bed.

1. Wait until the Extruder and Heated Print Bed have cooled down to room temperature. This will allow the plastic to contract, making it easier to remove
2. Using the Scraper, carefully 'wedge' under one of the edges of the 3D object. Take care tokeep the Scraper flat with the bed, as to not cause damage to the Print Mat
3. As one edge begins to lift, gently manoeuvre the Scraper forward to continue to 'wedge' between the 3D object and the Heated Print Bed
4. Once the part has fully separated from the bed, it is safe to remove

NOTE: Never scrape towards your body, or injury could occur

NOTE: Removing a 3D object, particularly one with a large surface size, can be a long process. Never rush when removing a 3D printed object

TIP: If you are persistently having trouble removing 3D printed objects, try the following:

- Use a layer of blue masking tape (designed to not leave behind residue) on the Heated Print Bed. Be sure to remove and replace after each print
- Check if the printing mat is damaged and replace if needed
  Replacement mats can be purchased at **id.aliexpress.com/store/120824**

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

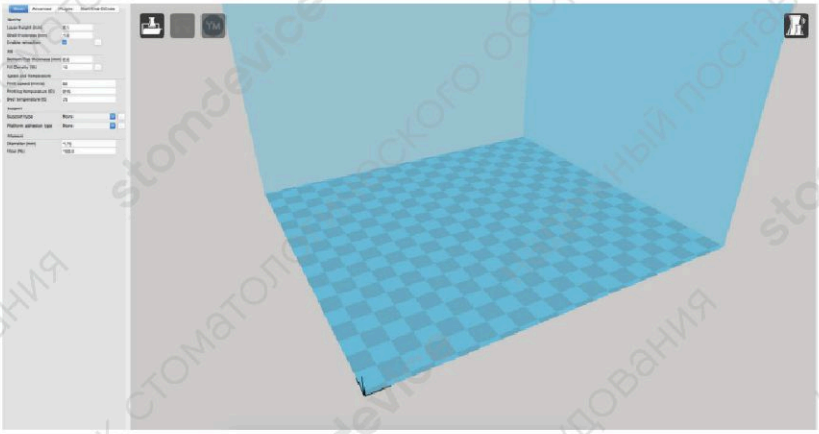
Using Cura

6. Creating a GCODE File using Cura

A GCODE file provides the essential temperature settings and movement information for your WANHAO 3D Printer. A number of 'Slicer' software exists, which convert a 3D model file into a series of 3D printable layers. Cura is a free Open Source slicer software released on the AGPLv3 license. It's available on MacOS X and Windows.

6.1 Installing Cura

1. Install the file located on your included MicroSD card or download Cura from **http://www.wanhao3dprinter.com/Down load Software/Duplicator i3**

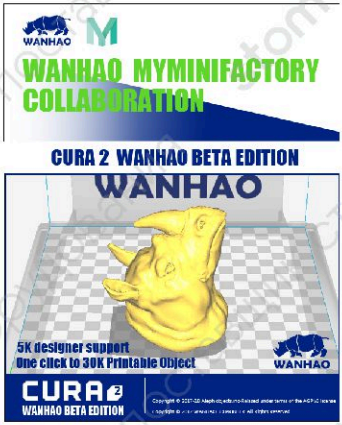


2. Select a destination folder for the install, then press 'Next'
3. Select the components to install, then press 'Install'
4. Once complete, press 'Finish'
5. Refer to your computer manual for further instructions on the above steps if required

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Using Cura

6.2 Install CURA WANHAO edition onto your computer



1. Double click the file: CURA_WANHAO_15.04.EXE
2. Follow the instruction and click install and next.
3. The software has already embedded all the basic settings for your Wanhao D9.

6.3 Loading a 3D Design into Cura

1. Clear the platform using **File > Clear platform...**
2. Then, load your 3D Design file onto the platform using **File > Load model file...**
3. Load any file with the following extensions: .STL, .OBJ, .DAE, .AMF

TIP: You can load more than one file onto the platform. Cura will automatically move the additional files to fit.

NOTE: Files in grey are outside of the print area and need to be moved or scaled to fit.

6.4 Viewing and Positioning your 3D Model in Cura

Changing the View:

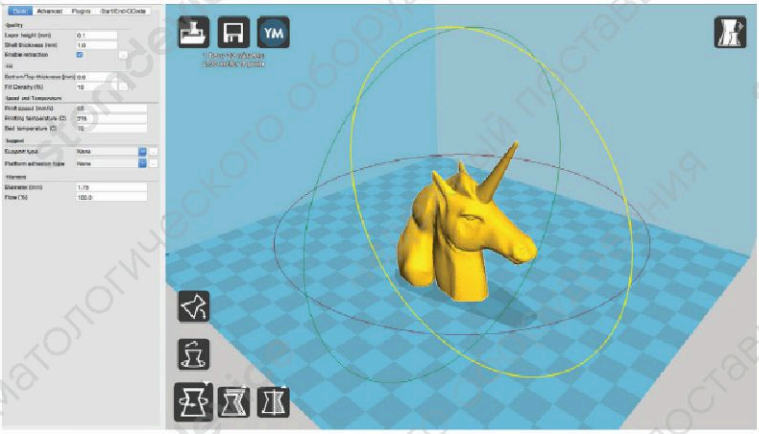
- **Zooming In/Out:** Use the mouse scroll wheel
- **Rotating the View:** Hold down the right mouse button and move the mouse

Move Position:

Move your model to change where it is printed on the build plate. Do this by left clicking on the model and dragging it to the desired location. The black outlined corner represents the lower left hand corner of the Heated Print Bed on your printer.

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Using Cura



Rotate

The Rotate button will give you the ability to orient your model in along all three axes. Once you click the rotate button, three circles will surround your model. The red circle will allow you to rotate around the Z axis. The Yellow circle will rotate around the Y axis. The Green circle will rotate around the X axis.

Lay Flat

The Lay Flat button will ensure that the flat portion of your print is securely attached to the bed. It is highly recommended to use this option after rotating your model in the Z direction, as it will help prevent adhesion issues during the print.

Reset

The Reset button will return your model to the original orientation as defined by the CAD program used to create the model.

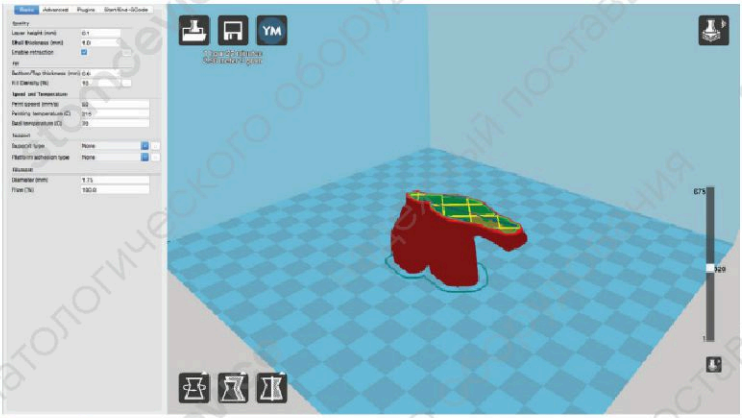
Scale

The Scale button displays the model dimensions, along with the ability to scale along the X Y or Z axes. Anything below the number 1.0 will reduce the objects size, while anything above the number 1.0 will increase the objects size. As a default, it will be set to uniform scaling. This will cause the X Y and Z axes to be scaled by the same amount when you make a change to any of them. To disable this, select the lock in the lower section of the scaling window.

TIP: Ensure your model is selected by left clicking on it. When selected, the model will have a white outline

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Using Cura



View Mode 

This mode allows you to view your model in a variety of different ways.
This can be helpful for spotting issues before the print even starts.

Normal
This is the standard view and shows the solid outer surfaces of the model.

Overhang
Overhang mode shows where your model may need support material. Red highlighted areas show overhangs and more severe angles and areas where support material is recommended. Refer to page 38 for information on how to add support material to your model.

Ghost
Ghost view mode makes the model translucent to allow you to see what is behind it.

Xray
Xray is very similar to Ghost mode. It will allow you to see into objects, ensuring that inner details are correct.

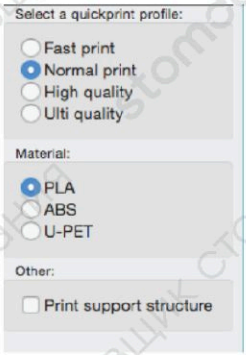
Layers

To view the toolpath of your print head and to ensure no skipped layers or gaps, use this option. Use the slide bar on the right hand side of the window to move up and down through the toolpath layers.

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Using Cura

6.5 Quick Print Settings



Quick Print mode allows you to quickly set the quality, material, diameter and support structure of your intended GCODE file.

Enter quick print mode by selecting **Expert > Switch to Quickprint...**

High Quality

Designed to give greater detail and finer objects. This will have a smaller layer height, which will make each layer thinner, so that curves seem more natural and walls seem less noticeable.

Medium Quality

Designed to give a medium resolution, by increasing the layer height and print speeds. This will make the organic curves slightly more step-like than the fine setting, but will reduce printing time.

Fast Quality

Designed for the fast prints, where overall model finish is not of concern. Most commonly used for quick iteration of designs found in rapid prototyping.

Material Selection

Choose your desired filament. The WANHAO 3D Printer includes a 10 metre sample of PLA to be used in your first print.

Other filaments are available for purchase at **id.aliexpress.com/store/120824**

Diameter

Ensure this is always set to match the filament diameter. Your 3D Printer only accepts WANHAO 1.75mm filaments.

NOTE: Advanced users can fine tune the diameter value to allow for tolerance if needed

Printing Support Material

Your WANHAO 3D printer is able to print models that have angles and overhangs, some of these models will require support material depending on the overhang distance and angle. Turn this option on if your model could benefit from support material.

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Using Cura

6.5.1 Exporting your GCODE File:

Take note of the time and length required to print your 3D object. From the top menu, select **File > Save GCODE**. You can now save your GCODE to your desired directory.

NOTE: Always reopen the GCODE file again in CURA to confirm that all slices of the model have been included. This will open in the 'layers' view, if you want to see your object, select 'normal' view

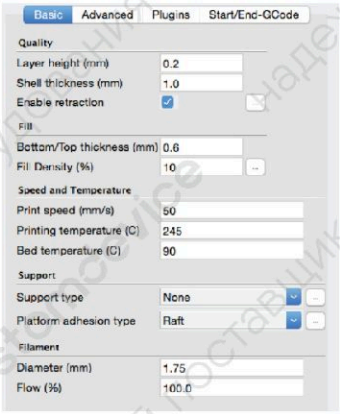
6.6 Advanced Print Settings

Advanced print mode should not be used until you have gained sufficient experience with 3D printing. In order to have more control of your slicing and GCODE generation, switch to advanced print mode.

Enter advanced print mode by selecting **Expert > Switch to full settings...**

Loading a Profile

When you first switch to Full Settings, Cura will revert to very generic settings. We recommend using our tested profiles that are available at **www.wanhao3dprinter.com.** You will want to choose the profile that matches your filament and quality needs. Once downloaded, you can load the file into Cura by selecting File > Open Profile. Choose your desired profile.
This will automatically update all of your settings for use with your printer.



6.6.1 Basic Tab Options

QUALITY

Layer Height (mm)

The thickness of each printed layer is known as the Layer Height. The smaller the layer height, the smoother curves will appear. Larger layer heights are better for bridging and overhangs. Smaller layer heights will also increase print time, as it will take more layers to complete the object.

Shell Thickness (mm)

This defines the number of vertical walls that comprise the outside of your model. We recommend keeping this set to multiples of your nozzle width. Your WANHAO 3D Printer is equipped with a 0.4 mm nozzle.

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Using Cura

Enable Retraction

Retraction tells your printer to pull filament out of the Extruder upon travel moves. Travel moves are when your print head moves from one area of the print, to another without laying down filament. We recommend keeping this on for all filament types, and adjusting the retraction length and speed for the specific filament.

FILL
Bottom/Top Thickness (mm)

Also known as Surface Layers; this will determine how thick the top and bottom layers are. A larger number here will create a thicker top and bottom which can be helpful for strength, bridging, and quality purposes. We recommend using a number which is a multiple of your layer height.
Eg: If using a layer height of 0.2mm, set the thickness to 0.4, 0.6 or 0.8mm

Fill Density (%)

This number is expressed as a percentage. 0% will give a completely hollow print, while 100% will give you a completely solid object. We recommend 20% to 40% fill density as it is functional for most prints.

SPEED AND TEMPERATURE
Print Speed (mm/s)

Your overall printing speed can be adjusted here. If no other speeds are determined in the later sections your printer will automatically default to this speed. This speed will be different, depending on what type of filament you are using. We recommend setting speed to 30mm/s.

Printing Bed Temperature (C)

When using different filament materials, you'll need to update the desired Extruder and Heated Print Bed temperature. Any temperatures specified here will be used to automatically set both the
Extruder and Heated Print Bed. Your print will not begin until these temperatures are met. As a guide, refer to Table 3.1 on page 27 for temperature settings.

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Using Cura

SUPPORT

Support Type

Some models will require support material in order to print properly. This will usually occur when an object has an angle in relation to the Heated Print Bede between 0 to 45 degrees. It is highly recommended to orient your object so that it minimizes or eliminates the need for support.

Touching Buildplate

This causes the support material to build up between the Heated Print Bed and the object. The red example is 'Touching Buildplate'

Everywhere

This prints support material between the Heated Print Bed and object as well as between the object and itself. The green example is 'Support Everywhere'.



Platform Adhesion Type

Some models have a small surface area contacting the plate. This can create adhesion issues causing your part to pop off at some point during the print. To fix this, use either 'Brim' or 'Raft'. Raft is better used when a model has small contact points with the Heated Print Bed and overhangs.

Brim

Brim will create a single layer of filament, contacting and surrounding your model. This will increase the surface area of the part contacting the build platform thereby preventing it from popping off the Heated Print Bed. Brim will also help in situations where you are seeing corner lift.

Brim settings can be adjusted by selecting '...' next to the tab.

Raft

Raft will generate a layer of material underneath your object. Raft was more often used before the addition of heated plates to increase surface area. Raft settings can be adjusted by selecting '...' next to the tab.

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Using Cura

FILAMENT

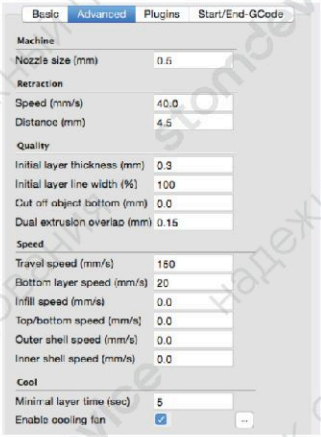
Diameter (mm)

The filament diameter setting is one of the more important settings. Make sure that you update this value periodically with your average filament diameter. While your filament may be referred to as 1.75mm, it is more likely going to be near 1.7mm +/- 0.1mm. You will want this to be an accurate average, as it will allow your printer to correctly calculate how much filament it is pulling into the Extruder.

Flow (%)

This controls how much filament your printer is extruding in relation to speed. This setting is mainly used to adjust for filament density variations. Leave this value at 100% as changing it can lead to surface quality issues.

6.6.2 Advanced Tab Options



MACHINE

Nozzle Size (mm)

This defines your nozzle size. The slicing engine uses this value combined with your other settings to determine how quickly to feed filament into your Extruder. The WANHAO 3D Printer uses a 0.4mm nozzle.

RETRACTION

Speed (mm/s)

Retraction Speed determines the speed at which your filament is reversed out of the Extruder for travel moves and when changing direction during printing. We recommend keeping this set to 40mm/s.

Distance (mm)

Retraction Distance determines how much filament is pulled out of your Extruder on travel moves and when changing direction. You will want to adjust this depending on temperature settings and filament type. Higher thermal retaining filaments such as PLA behave better with a longer retraction distance. Anywhere from 1mm to 3mm is a good starting range.

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Using Cura

QUALITY

Initial Layer Thickness (mm)

This will control how thick your first printed layer height is printed onto the Heated Print Bed. Having a larger initial layer height will help prevent your part from popping off the plate. Your WANHAO 3D Printer auto levelling system could be affected if you change this from the standard profiles. **Adjust at your own risk.**

Initial Layer Line Width (%)

This will control how wide your first extruded filament path is for the initial layer. A wider line width will help with bed adhesion. We recommend 125% as a good starting place. For models with moving printed in place parts, a smaller initial layer line width is recommended.

Dual Extrusion Overlap (mm)

This will determine how far your Dual Extruders will overlap when laying down material. This will help adhesion between the two different colours or types of filament. This setting is not applicable to the WANHAO 3D Printer, it is only for printers with dual Extruders.

SPEED

Travel Speed (mm/s)

This setting will determine how fast your print head moves while not extruding filament. A normal travel speed of 125 - 150mm/s is recommended.

Bottom Layer Speed (mm/s)

This will control your initial layer speed. In general, a slower initial layer speed will help with first layer adhesion.

Infill Speed (mm/s)

This is how fast your print head speed will be while laying down the interior portion of your model. Faster speeds are usually tolerable here, as none of the infill will be visible from the outside of your object. However, if you go too fast compared to your inner and outer shells, you can have adhesion issues or globs of filament left behind from the print head.

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Using Cura

Outer Shell Speed (mm/s)

This will be the outermost surface of the model. This is the most important setting, as it controls the speed of your print head on the visible layers. As a general rule of thumb, the slower you go the better looking print you will get.

Inner Shell Speed (mm/s)

This affects vertical walls that are in between the outer shell and infill. This will not be visible but will help support the outer shell and the infill. We recommend keeping this speed setting between your infill speed and your outer shell speed.

COOL

Minimal Layer Time

This will determine a minimum amount of time your printer will spend laying down each layer. If your layer print time falls below this your printer will automatically slow down to reach this time before moving onto the next layer. Tweaking this can help get cleaner, crisper prints.

Enable Cooling Fan

Enables operation of your Extruder's active cooling fan. The fan settings can be adjusted by selecting '...' next to the tab..

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Using Cura

6.6.3 Plugins Tab



Plugins are custom settings which will alter your print at specific points. The two that come pre-loaded with Cura are Tweak at Z, and Pause at Height. More plugins and information can be found here:

http://wiki.ultimaker.com/Category:CuraPlugin

To activate one of these highlight the desired plugin and click the drop-down arrow directly below the Plugins box.

Tweak at Z

Make basic changes at specified Z heights. You can determine the Z height or layer count at which you want to make a change. Then choose how you would like to change your settings. You can alter temperatures, fan speeds, and print speeds. Fine tuning these for specific STL files, can produce cleaner prints.

Pause at Z Height

Pause your print at a specified height. You can also specify where to move the print head and how much filament to retract. This will prevent 'blobs' from accumulating on your print while paused. This setting is most commonly used when switching colours of filaments in the middle of a print.

6.6.4 Start/End GCODE Tab



Custom Gcode allows for complex automatic printer movements and operations. By adding custom Gcode into the start or end of your file, you can alter how it prints. A comprehensive list of Gcode commands can be found here:

http://reprap.org/wiki/G-code

We recommend new users to leave this as provided.

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Using Cura

6.6.5 Expert Settings

Expert settings will give you more specific options for your retraction, skirt, active cooling, infill, support, brim, raft, and special settings. To gain access to this section you go to **Expert > Open Expert Settings** or on your keyboard press **Control + E.**

Retraction

Retraction pulls filament out of your nozzle when it is not extruding to prevent your print head from dripping on your object. This section is where you will control how your Extruder retracts its filament.

Minimum Travel (mm)

This sets the minimum travel distance of your print head in order to retract. If your print head is not moving this far during travel moves, it will not retract.

Enable Combing

This option prevents your print head from travelling over holes in the X/Y plane when printing.

This will slightly increase print time, but will prevent strings from getting caught on the holes during travel moves. We recommend keeping this setting on.

Minimal Extrusion before Retracting (mm)

This will control the distance at which retraction occurs if the printing movement exceeds the minimum extrusion amount. This will prevent a retraction move, if your Extruder has not put out Xmm of filament since its last retraction.

Z Hop When Retracting (mm)

This will raise your print head Xmm while retracting. This setting helps prevent ooze, and strings from being deposited on your print.

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Using Cura

Skirt

Skirt creates a line around the outside of your object. This is most commonly used to prime the Extruder. In order to prevent missed filament at the beginning of a print, we recommend you leave this setting on.

Line Count

This will define the number of loops the Skirt creates around the outside of your object. Smaller models will require more loops to properly prime the Extruder.

Start Distance (mm)

This will define the distance away from your model that the skirt will be created. If using as an envelope to prevent drafts, it is recommended to be closer to your object.

Minimal Length (mm)

This will define the minimum extruded line length for the skirt. This will over ride your line count, producing as many lines as required to reach the minimal length.

Cool

This section will define how your Extruder cooling fan will operate during the print. Your fan will not start until it has reached 25% or higher for speed settings. If your print speeds are slowed down due to minimal layer time, the fan will run between minimum and maximum speed based upon how much the layer is slowed down.

Fan on at Full Height (mm)

The fan will not turn on for the first layer. The fan will then be scaled between 0% and your minimum fan speed based on layer height. It is especially helpful with high temperature retaining filaments such as PLA.

Fan Speed Min (%)

Once the Z height is reached for Fan on at Full Height setting, this will be the speed your fan runs at.

Fan Speed Max (%)

This is the fastest speed at which your fan will ever run. When your print speed is slowed down due to Minimal Layer Time settings, your fan will run between minimum and maximum speed. The maximum fan speed is reached when your printer speed is slowed by 50% or greater.

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Using Cura

Support

You define how your support material is generated here. You must have a support type in the 'Basic Settings; in order for these support settings to have an effect.

Structure Type

You can choose between a 'Grid' or a 'Line' pattern for your support material. The grid will be a checkerboard pattern in the X and Y direction. The line option will produce lines along the Y-axis for support. The grid will provide stronger support than the line option, but will be harder to remove.

Overhang Angle for Support (deg)

This will determine where support material is generated. In general you will be able to print a model with 45 to 90 degree angles in relation to the Heated Print Bed without support. We recommend leaving this setting at 45 degrees.

Fill Amount (%)

This will determine how dense your support material is printed, similar to Infill Percentage. The higher percentage the better support, but it will be harder to remove the support material and will use more filament.

Distance X/Y(mm)

This will determine how far away from your object in the X/Y plane that the support material is being placed.

Distance Z (mm)

This will determine how far away your support material is from your object in the vertical direction. A smaller number here is better for support, but makes it harder to remove.

Black Magic

This section allows you to transform your model into a hollow shell, a single layer thick

Spiralize the Outer Contour

This causes your Z-axis to be constantly moving upward as you are printing your single outer wall shell. This results in no layer change lines, giving a much smoother surface. This setting is typically only used for artistic objects as they will be fragile.

Only Follow Mesh Surface

This will cause your print to follow the outside of your model, building it completely hollow with a single wall outer shell. The only difference between this and Spiralize, is that the Z-axis moves regularly. That is, it prints a layer and then moves up to the next one.

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Using Cura

Brim

Brim circles the base of the print while making contact, helping adhere the print to the Heated Print Bed. This is only one layer thick, and can be easily removed post-print. This section defines how the brim is formed when brim is activated in 'Basic Settings'.

Brim Line Amount

This will determine the distance the brim will cover around the outside of your object. The more brim used, the better your part will adhere to the plate.

Raft

Raft is a platform built underneath your object, designed to help adhesion and prevent warping. It will lay down support material, and then a platform on top of the supports. Your model will be built on top of this platform. The bottom surface of your printed object will not be as clean or as even when using this option. Raft is typically not recommended.

Extra Margin (mm)

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

Line Spacing (mm)

This will determine the spacing between support lines for the raft. A small spacing makes the support structures closer together improving strength of the raft, but uses more filament.

Base Thickness (mm)

This defines how thick your raft will be.

Base Line Width (mm)

This will define how wide your support material is for the raft. This setting will determine how well the surface layers of the raft print.

Interface Thickness (mm)

This will determine how thick the surface layers of the raft are. The surface layers are the platform that is built upon the supports.

Interface Line Width (mm)

This will determine how wide the top layers of the platform will be. In general, you can keep this set to your nozzle size, as surface quality of the removable raft is not important.

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Using Cura

Airgap

This will define the distance between your raft and your print. A larger gap will make your part easier to remove, but will make the bottom of your print look worse.

Surface Layers

This will determine the number of layers that create the 'platform' of your raft. If you have a wide line spacing, you may want to increase this number to ensure a solid platform.

Fix Horrible

These are some of the more advanced and experimental options. They are designed to help repair models with errors to make them suitable for 3D printing. However they may not always work. Please be cautious when using these options as they can have unintended effects on your print quality.

Combine Everything (Type-A)

This will attempt to fix all external mesh errors, while keeping internal holes intact. This can accidentally fill in intentional internal holes.

Combine Everything (Type-B)

This will ignore all internal holes of the model and only focus on the external holes. This is helpful when only the outside finish of the model is important.

Keep Open Faces

This will ignore all manifold errors in the object. It can create issues generating the Gcode as Cura does not know how to interpret the open holes. This option should only be used if you are sure that the holes in the mesh are intended. In general, you should not use this option.

Extensive Stitching

This causes Cura to automatically add triangle mesh in an attempt to fix manifold errors. This algorithm will greatly increase Gcode generation time and may end up adding in un-intended mesh. It is recommended that you repair your model through your CAD program before attempting this option.

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Servicing

7. Servicing your 3D Printer

7.1 Replacing the Print Mat



The WANHAO 3D Printer includes 2 x Print Mats. Additional Print Mats are available for purchase from **id.aliexpress.com/store/120824**

1. Remove the current Print Mat by slowly peeling from one corner
2. Using rubbing alcohol and a microfiber towel, carefully remove any residual adhesive left on the Heated Print Bed
3. Replace the new printing pad,you must making sure to align with each edge.

TIP: Warming the bed slightly may make removal easier. You can also use masking tape in lieu of a Print Mat. Blue painters tape is recommended.

7.2 Nozzle Cleaning





If your Extruder nozzle becomes blocked, follow these steps to clean the nozzle.

1. Remove the existing filament using steps from section 3.3 'Removing Filament'
2. Raise the Extruder using Position > Z Pos. Fast, then rotating the Dial clockwise
3. Heat the Extruder to 215°C
4. Using a clean 0.4mm drill bit, insert the drill bit into the nozzle by hand, twisting while inserting
5. Remove the drill bit from the nozzle, twisting while removing
6. Repeat steps '4' and '5' three times, then follow the steps in section 3.2 'Inserting Filament' to re-insert the filament into the nozzle

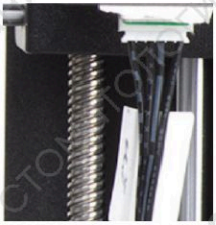
https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Servicing

7.3 Routine Component Check

Motor Connectors

Loose motor connections could cause a 3D print to fail. It's a good habit to routinely check for any damage to the connectors or the black cables.



Axis Lubrication

Regular lubrication of the X and Y Axis rods is important for the longevity and smooth running of your printer. The rods can be lubricated with oil or grease. We recommend using bearing grease, and to apply generously to both the X Axis and Y Axis rods.



Nuts and Bolts

Loose nuts and bolts could affect the geometry of the 3D Printer. Check that all nuts and bolts remain firmly tightened. If required, re-tighten using the hex wrench.

7.4 Storing

Always store your 3D Printer in a cool, dry location that is free from dust.

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Finishing

8. Finishing Techniques

8.1 Useful Tools/Equipment

- Safety Goggles
- Mask
- Long-nose Pliers
- Wire cutters
- Sandpaper (100,200,300,400,500,600 Grit)
- Medium, fine and extra fine sanding sponges
- Acrylic based primer spray paint
- Acrylic based spray paint (colour of your choice)
- Acrylic based varnish spray paint

NOTE: Different paint types are also compatible with PLA and ABS. Consult your local paint supplier for the best compatibility for your intended purpose.

8.2 Removing Support Material

Even after support material is removed by hand, residual material will remain.

1. Ensure that safety goggles and a mask are worn for protection
2. Remove most of the support material by hand, taking care not to damage the 3D object
3. Using the long-nose pliers, carefully grasp some of the remaining material, then twist to remove. Repeat until no more material can be removed.
4. Using the wire cutters, cut away any excess filament.



https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Finishing

8.3 Sanding

Though Fused Filament Fabrication (FFF) is capable of producing high quality objects straight out of the machine, the layer lines will be visible. Sanding will reduce these 'step' marks in the model.

1. Ensure that safety goggles and a mask are worn for protection
2. Starting with 100 grit sandpaper, carefully sand your 3D object using small, circular motions

NOTE: PLA is less resistant to high temperatures caused by sanding. Take your time sanding your 3D object to avoid damage.

3. Using the medium sanding sponge, sand difficult to reach areas of your 3D object
4. Gradually increase the sandpaper grid until the 3D print is smooth

TIP: Spraying a primer filler will also help to reduce the layer lines on the 3D Object. This can then be sanded away and repeated, until the desired smoothness is achieved

8.5 Surface Coating

Finally, to make your models more presentable, follow these steps;

1. Ensure that safety goggles and a mask are worn for protection
2. Ensure that your 3D object is clean and free from dust
3. Spray your model using an acrylic based primer, following the directions of your primer
4. Allow appropriate time to dry before applying additional coats
5. Spray your model using an acrylic based paint of your colour. Allow appropriate time to dry
6. Spray your model using an acrylic based varnish, following the directions of your varnish. Allow appropriate time to dry before applying additional coats.

NOTE: Different paint types are also compatible with PLA and ABS. Consult your local paint supplier for the best compatibility for your intended purpose.

Congratulations! Your 3D object now looks like a professionally finished product!

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Troubleshooting

9. Troubleshooting

If having any troubles when using the 3D printer, please refer to the troubleshooting guide below, or call customer support on **86-571-23290996**

You can also visit Please visit **www.wanhao3dprinter.com/Video/D9 video/wanhao D9 unboxing** for support videos and advanced tutorials.

9.1 Printing Problems

Clicking sound from one of the stepper motors

One of your stepper motors may not be connected properly. Check each connection and then inspect the cable routing for any faults

File not printing

Remove the Micro SD Card and insert into your Computer. Open the GCODE file using a text editor (eg. Notepad), and inspect if GCODE is readable or not. If file consists of multiple 'ÿÿÿÿ' symbol, then file has been corrupted. Try reformatting your Micro SD Card and reloading the GCODE file.

NOTE: Reformatting will delete all files on your MicroSD card

Not extruding when starting to print

Ensure that the nozzle temperature has been set to match your material

Not sticking to the bed

- Ensure that the bed is clean and free from dust
- Check that the bed is level and set to a distance of approximately 0.1mm
- If there are defects / deep scratches in the printing mat, it may need to be changed. Change the mat or use blue painters masking tape
- Add a brim or raft to your file
- Check that the bed temperature matches your material
- Ensure nozzle to bed gap is not bigger than 0.1mm. Material will solidify if gap is large

Models Warping/Curling

- Check that the bed temperature matches your material
- Check the infill % of your GCODE. The higher the infill, the more likely to warp
- Add a brim or raft to your file

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Troubleshooting

Model appears very 'stringy'

- Ensure that the filament diameter is set to match your filament spool (1.75mm), then try creating a new GCODE before printing again
- Check you are using the correct temperature for the filament used

Print stopped halfway

- Check that the GCODE file is complete and not corrupt
- Check the original model file

The motor does not stop at the end of axis

Check that the stop micro switches are aligned with the motor mount and registering.

The print head/bed does not move

- Check the part can be moved by hand when stepper motor is disabled. Clear anything that is blocking the path
- Check that the motor is turning (using position menu). If motor turns but belt does not, you will need to tighten the gear motor gear nut. Please visit **www.wanhao3dprinter.com/Video/d9 video/wanhao d9 unboxing**

10.2 Connectivity Problems

My 3D Printer isn't turning on

Check that you have correctly inserted the power cable into the rear of the Control Box, the power cord is plugged into the mains outlet and that the Control Box power switch is on.

Micro SD Card not shown

Navigate to SD Card, select Unmount SD Card, then select Mount SD Card. Your Micro SD card should now appear in the menu.

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Troubleshooting

10.3 Other Problems

Filament stuck when removing

Please visit **www.wanhao3dprinter.com/Video/d9 video/wanhao d9 unboxing** for a video on how to clear the nozzle with stuck filament.

Where can I buy more filament?

If you require more filament, visit our online store, **id.aliexpress.com/store/120824V** to purchase various colours and types.

What brand of filament should I buy?

We recommend using only WANHAO branded filament. Lower quality generic filaments can damage your 3D Printer and potentially void your warranty.

What types of filament can my 3D Printer Print?

The WANHAO 3D Printer accepts various types of filaments including PLA, ABS, PVA, HIPS and specialty filaments such as wood, flexible, luminous and conductive. For our full range of filaments visit **id.aliexpress.com/store/120824**

10. Useful Terms

ABS Filament

Commonly used plastic filament with good strength and industrial properties

Display

Provides the main interface for operating the 3D Printer

Firmware

Software that is programmed into the electronic board

GCODE

Code that is produced using the slicing process. Describes the movements and temperature settings of your 3D printer

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Troubleshooting

Heated Print Bed
Surface that your 3D objects are printed onto

Nozzle
Opening at the bottom of the print head where filament is pushed through

Micro SD Card
Secure Digital memory card. This is where your GCODE files can be stored and accessed by the 3D Printer

PLA Filament
Commonly used plastic filament with low environmental impact

Slicing
Process that turns a 3D Design into a code that can be used by 3D Printers

Spool
Filaments of PLA and ABS are wrapped, creating a spool

Stepper Motor
DC motors that move in discrete steps. This allows for precise positioning and/or speed control

.stl
A common file format for 3D Designs/Models

USB Cable
This cable allows communication between the WANHAO 3D Printer and a computer, using the USB port of the computer

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

Appendix

References

Some of the 3D Models used in this manual and other included materials were sourced from the below:

Unicorn Head by kellesabelle
Published on February 5, 2012
www.thingiverse.com/thing:17090

Creative Commons - Attribution - Share Alike

Voronoi lamp by Markellov
Published on December 9, 2014
www.thingiverse.com/thing:584714

Creative Commons - Attribution - Share Alike

Twisted 6-sided Vase Basic by MaakMijnIdee
Published on March 7, 2012
www.thingiverse.com/things18672

Creative Commons - Attribution - Share Alike

General purpose hook by Erikjuh
Published on October 20, 2015
www.thingiverse.com/thing:1081862

Creative Commons - Attribution

Cable Holder (Cable Clip) by bardiaesm
Published on April 6, 2013
www.thingiverse.com/thing:70549

Creative Commons - Attribution

Twisted bracelet by Gepetto
Published on January 25, 2015
www.thingiverse.com/thing:651677

Creative Commons - Attribution - Share Alike

Geoflower by EMRL
Published on August 25, 2015
www.thingiverse.com/thing:982849

Creative Commons - Attribution - Share Alike

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ

stomdevice

WANHAO

3D Printer

Repair and Refurbished Goods or Parts Notice

Unfortunately, from time to time, faulty products are manufactured which need to be returned to the Supplier for repair.

Please be aware that if your product is capable of retaining user-generated data (such as files stored on a computer hard drive, telephone numbers stored on a mobile telephone, songs stored on a portable media player, games saved on a games console or files stored on a USB memory stick) during the process of repair, some or all of your stored data may be lost. **We recommend you save this data elsewhere prior to sending the product for repair.** You should also be aware that rather than repairing goods, we may replace them with refurbished goods of the same type or use refurbished parts in the repair process.

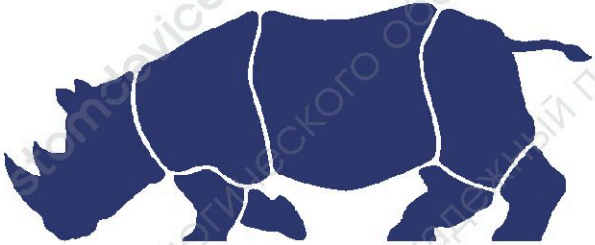
Please be assured though, refurbished parts or replacements are only used where they meet WANHAO's stringent quality specifications.

If at any time you feel your repair is being handled unsatisfactorily, you may escalate your complaint. Please telephone us on 86-571-23290996 or write to us at:

WANHAO Precision Casting Co.,Ltd
77, RENMING ROAD, JINHUA, CHINA
WANHAO Help Desk 86-571-23290996
(Operating Hours: Mon - Sat; 8:00am - 5:00pm)
support@wanhao3dprinter.com
www.wanhao3dprinter.com

1 YEAR WARRANTY

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ



WANHAO

https://stomdevice.ru - стоматологическое и зуботехническое оборудование с доставкой по России и СНГ