

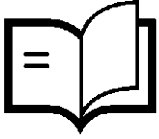
D12 230 USER MANUAL



DOUBLE EXTRUSION WIFI 3D PRINTER

IMPORTANT

HOW TO GET THE MOST OUT OF YOUR 3D PRINTER



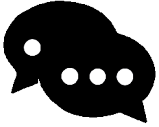
READ THIS MANUAL CAREFULLY

You will be able to set up with ease



VISIT OUR WEBSITE FOR TUTORIAL VIDEOS AND GUIDES

www.wanhao3dprinter.com



Google Forum

JOIN WANHAO GOOGLE FORUM FOR ONLINE SUPPORT

<https://groups.google.com/forum/#!forum/wanhao-printer-3d>



EMAIL US! WE'RE ALWAYS HAPPY TO HEAR FROM YOU

Technical support: support@wanhao3dprinter.com

Product feedback and partnerships: hello@wanhao3dprinter.com

Purchase orders: sales@wanhao3dprinter.com

WARRANTY NOTICE

If the product is not performing up to your expectations or you experience any issues, please contact us before returning the item.

We are experienced in resolving problems through phone and email, or you can reach us through your local distributor.

Phone: +86-571-6337-9578 (Hours: Mon to Sat from 8:30 AM to 5:00 PM China Time)

Email: support@wanhao3dprinter.com

Website: www.wanhao3dprinter.com



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

If at any time you are unsatisfied with how your repair is handled, you may escalate your complaint to WANHAO. Please contact us via:

Phone: +86-571-6337-9578

(Hours: Mon to Sat from 8:30 AM to 5:00 PM CST China Time)

Mail: HANGZHOU BEAN 3D PRINTER CO.,LTD

No. 18 Zhen Shan Road, Chang Kou Town, Fuyang District, Hangzhou City, Zhejiang Province, China

Email: support@wanhao3dprinter.com

Website: www.wanhao3dprinter.com



AFTER SALES SUPPORT



86-571-63379578



support@wanhao3dprinter.com



SAFETY INFORMATION

Read the instructions carefully. Keep this document for future reference. Follow all warnings and instructions marked on the product. Observe the following guidelines when connecting and disconnecting power to the external power supply unit:

- Install the power supply unit before connecting the power cord to the AC power outlet.
- Unplug the power cord before removing the power supply unit from the printer.
- Operate on a table or flat, stable surface, ensuring that the machine cannot fall and is level.
- Do not operate outdoors.
- Do not use solvents to clean the printer as they may damage the screen. Use a little detergent and warm water to clean.
- Disconnect the printer from AC before storage or when not used for a long period of time. This 3D printer is connected to the electrical network with an input of 110 or 240 V AC, 50/60 Hz, and has an operational voltage of 24V.
- Do not allow anything to rest on the power cord. Do not locate this product where people will walk on the cord.
- If an extension cord is used with this product, make sure that the total ampere rating of the equipment plugged into the extension cord does not exceed the extension cord ampere rating. Also, make sure that the total rating of all products plugged into the wall outlet does not exceed the fuse rating.
- Use the product only with the supplied power supply cord set. If you need to replace the power cord set, make sure that the new power cord meets the following requirements: detachable type, UL listed/CSA certified, VDE approved or its equivalent, 4.5 meters (15 feet) maximum.
- In case of malfunction, disconnect the printer immediately from network.
- Do not attempt to fix this product by yourself, as opening or removing covers may expose.

you to dangerous voltage points or other risks. Refer all repairs to qualified service personnel. Please send an email to your national supplier or contact technical service at:

support@wanhao3dprinter.com

Unplug this product from the wall outlet and refer servicing to a qualified service personnel:

- The power cord or plug is damaged, cut or frayed.
- Liquid has been spilled in the machine.
- The machine was exposed to rain or water.
- The machine has been dropped or the case has been damaged.
- The machine does not operate normally after following the operating instructions.

DISPOSAL INSTRUCTIONS



Do not throw this electronic device into the waste when discarding. To minimise pollution and ensure protection of the environment, please recycle or return to Photocentric for recycling.

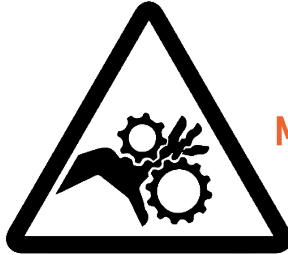
BEFORE YOU START



SAFETY INSTRUCTIONS

Before you get started, please read these safety instructions.

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



MOVING PARTS



Do not leave the WANHAO 3D Printer unattended during operation.



Do not point sharp tools at yourself when dealing with model stripping.



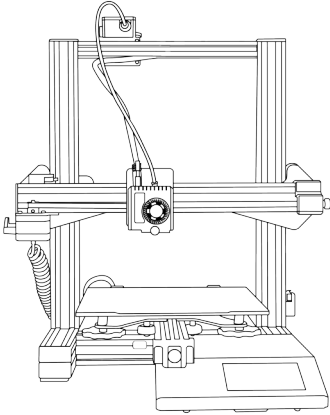
During the printing process, do not have a strong collision with the printer.

In case of emergency turn off the 3D printer by unplugging the power outlet.

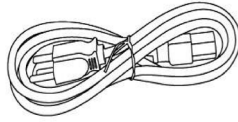
- The printer must not be exposed to water or rain, or the printer will be damaged.
- The printer is designed to be used in ambient temperatures, ranging from 15°C to 30°C.
- The printer is designed to be used in humidity levels ranging from 20% to 50%.
- Operating the printer outside the recommended temperatures and humidity levels may result in low quality print models.

CON TENTS

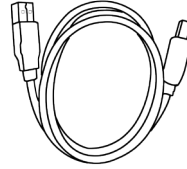
The box contains



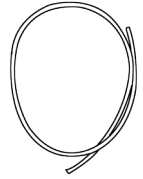
D12 230
3D PRINTER



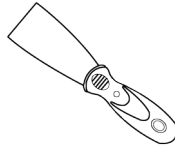
POWER CABLE



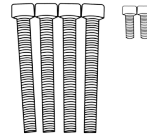
USB CABLE



FILAMENT
FEEDING TUBE



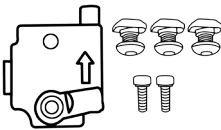
SHOVEL



SCREW



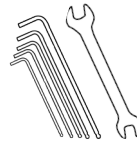
FILAMENT
SHAFT



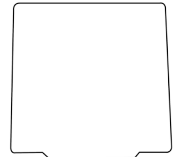
FILAMENT
DETECTOR



NOZZLE
CLEANER



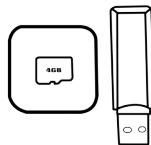
HEXAGON
WRENCH



MAGNETIC
HEATING PLATE



TWEEZERS



MICRO SD CARD
AND READER

TECHNICAL SPECIFICATIONS

Software	WANHAO Cura
Extruder system	MK12
Material support	PLA,PETG,SILK
Max printable area	230 * 230 * 250mm
Filament diameter/light	1.75MM
Max printing speed	30 mm/s-150mm/s
Case material	Steel case
Extruder quantity	1
Net weight(kg)	8.56kg
Gross weight(kg)	11kg
Accuracy	Motor Accuracy
Packing size(cm)	53.5*48*27.5cm
Layer thickness	100-400 um
Platform	Aluminium platform W/ heating
Certification	CE,CB, FCC
LCD display size	75*51 mm
Language	Chinese simplify, English, Russian, Spanish, French, Italian, OEM any other language
Power supply unit	Input 110 AND 250V(optional), 50/60Hz, 350W, 1.9A. Output 24V 4.5A

Read this manual carefully, it
will bring you great convenience

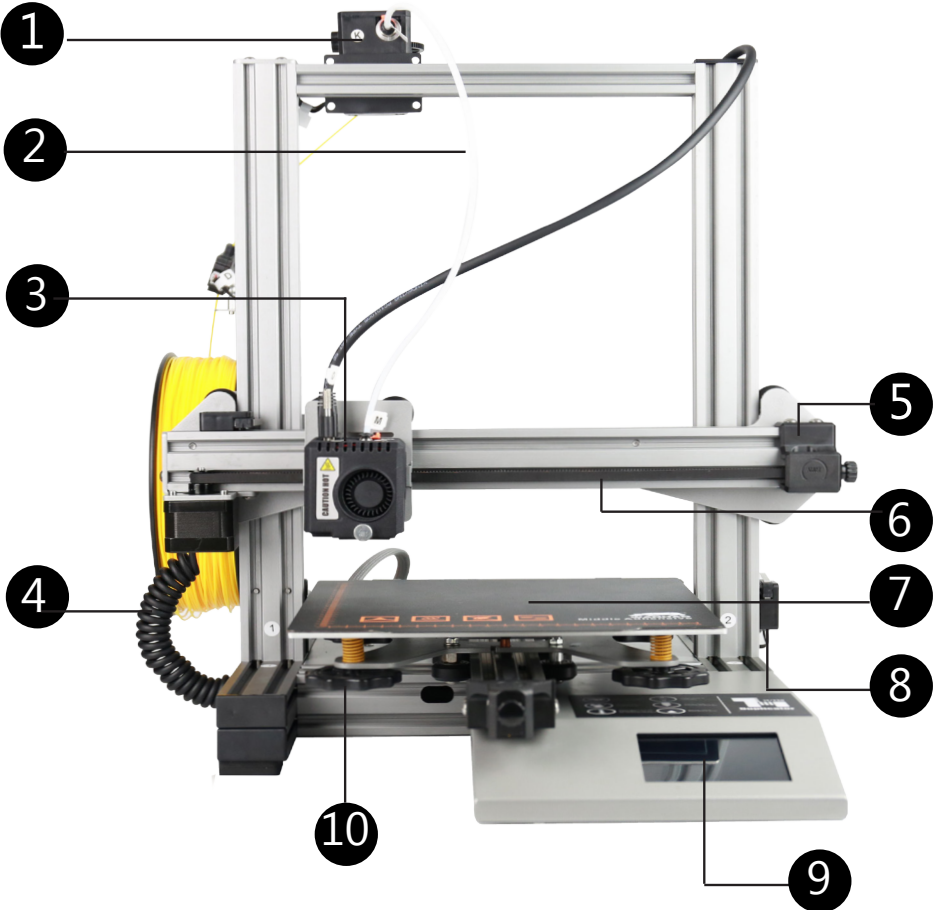
USING D12 230

THE FIRST TIME



PRODUCT OVERVIEW

PRINTER FRONT VIEW

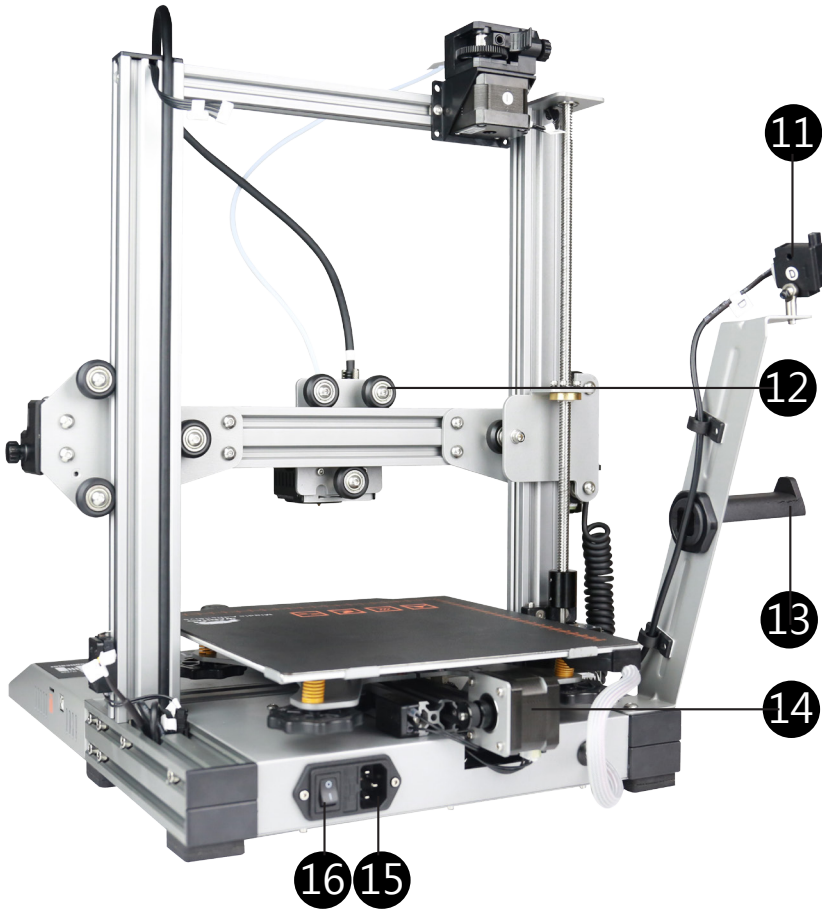


- 1. Extruder data cable
- 2. Filament tube
- 3. Extruder assembly
- 4. Z axis cable pack
- 5. X axis belt tension adjuster

- 6. X axis belt
- 7. Movable bed sheet
- 8. Z axis end stop switch
- 9. Touching screen
- 10. Leveling nut

PRODUCT OVERVIEW

PRINTER REAR VIEW



- 11. Filament detector
- 12. X axis roller
- 13. Filament holder

- 14. Y axis motor
- 15. Power socket
- 16. Power switch

FIRST COMMISSIONING

INITIAL SETTINGS

Thanks for choosing Wanhao 3D Printer D12/230. It's important that you follow these steps carefully to ensure correct setup of your 3D Printer.

You can also visit <https://www.wanhao3dprinter.com/Vedio/...> unboxing for step by step guide on unboxing and setup, levelling, filament installation and printing.



Package Contents:

6 sets components pack as left picture

1x Wanhao D12/230 printer

1x filament spool stand

(You may have 2 in case you have dual extruder)

1x power plug

1x filament feeding mechanism

(You may have 2 in case you have dual extruder)

1x printing head



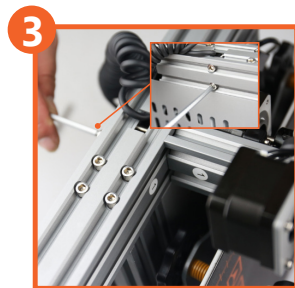
ASSEMBLING AND INSTALLATION



1 Place the printer components onto a flat table. Please handle with care.

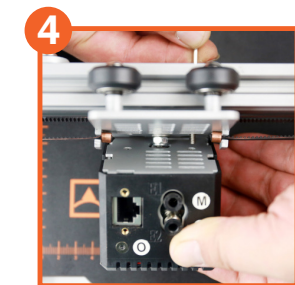


2 Remove the protection foam and tape from position.



3 Put the printer base on it's side. Install the printer tower onto the base. Then use 4pcs M5 x 40mm bolt to lock the tower onto the base.

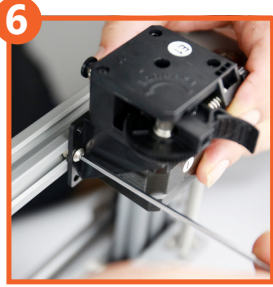
Note: On the base there's sticker No.1 and on the tower there's sticker No.1. So No.1 base match to the No.1 tower. No.2 base match to the No.2 tower.



4 Install the printing head onto the gantry. Then to lock the head with **2 M3 x 6mm** locking screw.



5 To push the printing head gently to ensure well secured. If not, re-lock the bolts.



Find the **E1** extruding mechanism. Install it onto the left side (near the Z axis screw) of tower by inserting the block into the aluminum slot and twist the bolt Mushroom head screw M4 x 5mm clockwise.



Install the filament plastic holder onto the left stand and twist the nut to lock it. Then install the **2 M3x 6mm** bolts with the blocks.



Inserting the block into the left side (near Z axis screw) aluminum slot and twist the bolt (Mushroom head screw M4 x 8mm) to lock it.



Install the filament sensor onto the cardan joint by one **M3 x 8mm** bolt. Then install the cardan joint onto the filament stand by one **M3 x 8mm** bolt.

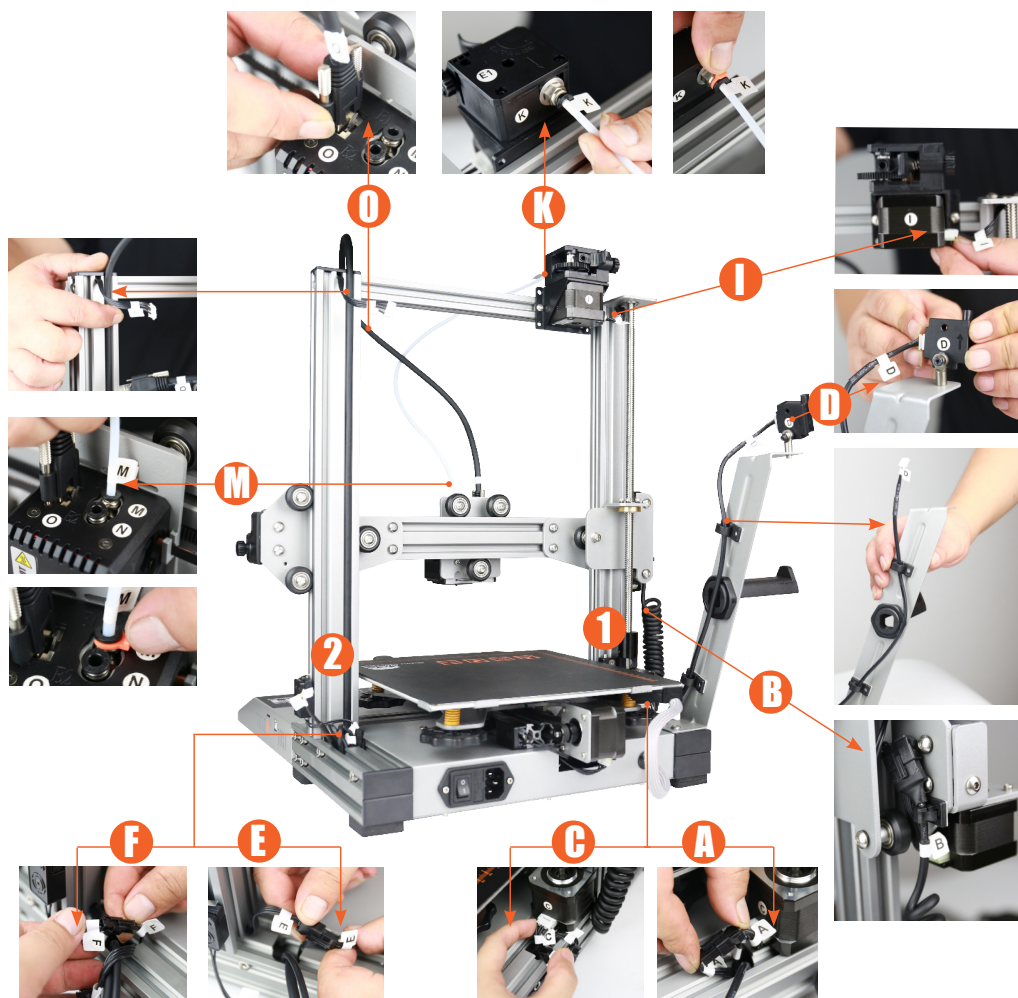


Twist the X axis belt tension adjusting knob clock wise to tighten the belt tension. You can check the tension by use one finger to push the belt. Either too tight or too loose would be perfect.



Twist the Y axis belt tension adjusting knob clock wise to tighten the belt tension. You can check the tension by use one finger to push the belt. Either too tight or too loose would be perfect.

WIRING INSTRUCTION

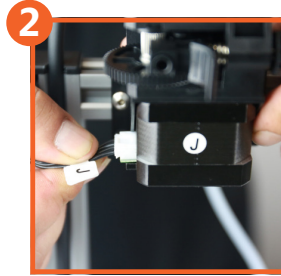


Note: Please connect every wire according to its Number which match to the corresponding connector.
Congratulation! You have complete the installation of D12/230. You are now ready to boot up and print.

INSTALLATION OF 2ND EXTRUDER.



Find the **E2** extruding mechanism. Install it onto the right side (on the other side of **E1**) of tower by inserting the block into the aluminum slot and twist the bolt clockwise.



Insert the **J** male connector onto the **J** female Motor connector.



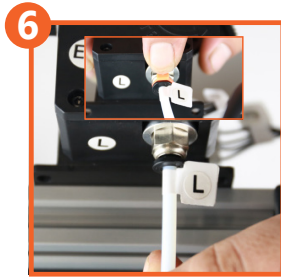
Install the filament plastic holder onto the left stand and twist the nut to lock it. Then install the **2 M3x 6mm** bolts with the blocks.



Inserting the block into the Right side aluminum slot and twist the bolt to lock it.



Install the filament sensor **H** onto the cardan joint by one **M3 x 6mm** bolt. Then install the cardan joint onto the filament stand by one **M3 x 6mm** bolt.



Insert the **L** filament tube into the **L** feeder. Then insert the orange locker to stop tube from falling out.



Insert the **N** tube into **N** connector.



Congratulation! You have install the 2nd extruder.

INTRODUCTION TO SLICING SOFTWARE

11 hours 7 minute
10.62 meter 3 g
15.39 meter 4 g

WANHAO CURA SOFTWARE

Our software is tailor-made for your D12 230 machines. Multiple language support. Using rapid iterations, we've included many popular requests from our community. The two-color writing model can be uploaded to the SD card connected to the printer through wifi after merging.



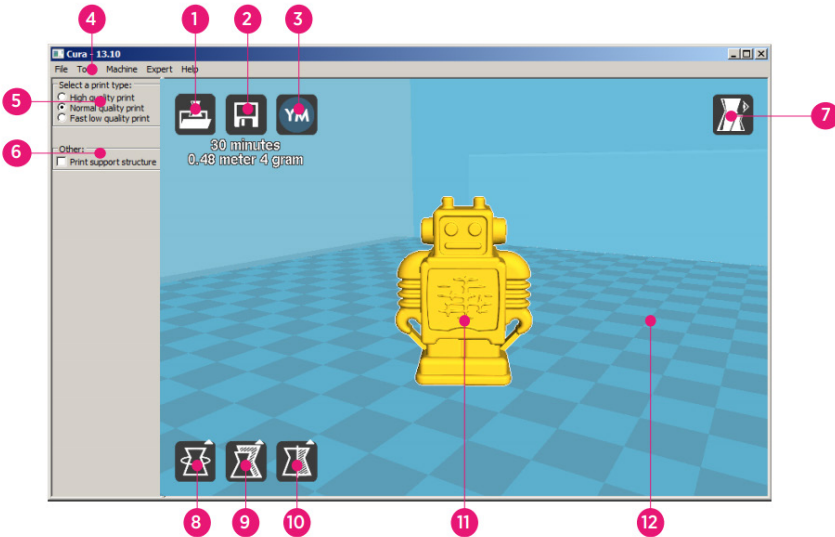
Info: Print one at a time mode disabled. Object to

温度
读取中...



CURA SOFTWARE INTERFACE

This is the main screen of Cura. Here you can choose which object or objects to print. And how you want them to be printed.



1. Load file

Click this button to load an object to your print area. You can add as many objects as you can fit on the plate.

2. Save toolpath (SD)

When you press this button the prepared toolpath will be saved on the SD card.

3. Share on

YouImagine.com
Through this button you can share your 3D files on YouImagine.

4. Menu bar

In this bar you can change to Full settings, preferences and more expert settings

5. Print type profiles

3 options to choose a print profile.

6. Print with support structure

The option to give your model some support material to make the print more successful.

7. View modes

After your model has been prepared for printing, you can use different view modes to analyse your design.

9. Scale model options

Option to change the scale of the object you like to print.

10. Mirror model options

Options to mirror the object you'd like to print.

11. Loaded 3D model

This is the object you have loaded through the load file button.

12. 3D print volume

This is a visualisation of the print area of your D6.

CURA SOFTWARE INTERFACE

The following mouse actions are used to work, navigate and view the 3D model:



Leftmouse button

Select objects. Hold and move the mouse to drag object on the 3D print area.



Scrollwheel button

Use the scroll wheel to zoom in or out.



Rightmouse button

Hold and move the mouse to rotate the viewpoint around the 3D model.



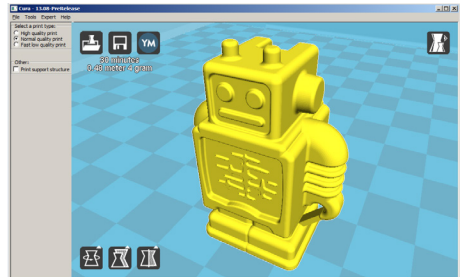
Rightmouse button + Shift

Hold and move the mouse to pan the 3D view.



Right and left mouse button

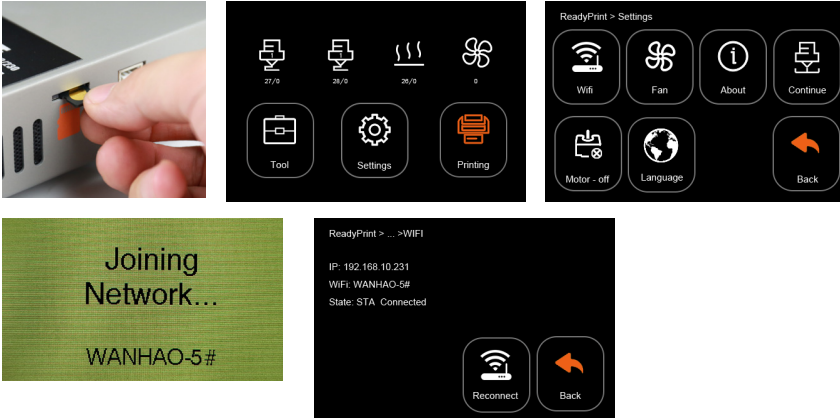
Hold and move the mouse to zoom.



Save the generated Gcode file to the SD card or USB flash drive, insert it into the printer, and start printing with the corresponding options.

WIFI CONNECTION CONTROL PRINT

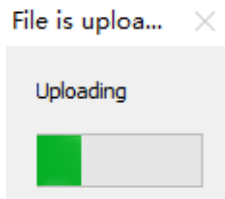
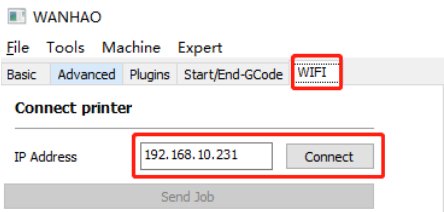
1. Insert the Micro SD card into printer. The file transferred from the computer would be stored here. Through your touching screen. Click SETTING-- WIFI--CHOOSE YOUR WIFI-- ENTER THE PIN--JOIN. Your printer would connect to your local router. Please record the printer IP address, 192.168.0.x



2. Connect your computer to your WIFI router. Please make sure the computer and the printer are connecting to the same WIFI router.



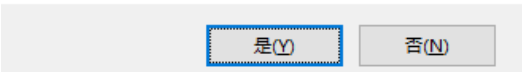
3. Open Cura WANHAO edition. Click Wifi. Enter the IP address which you record from your printer. Click Connect. Now your computer would connect to your printer. Once you finish slicing, you can click Send Job to send the printing file to your printer for printing.



Upload success

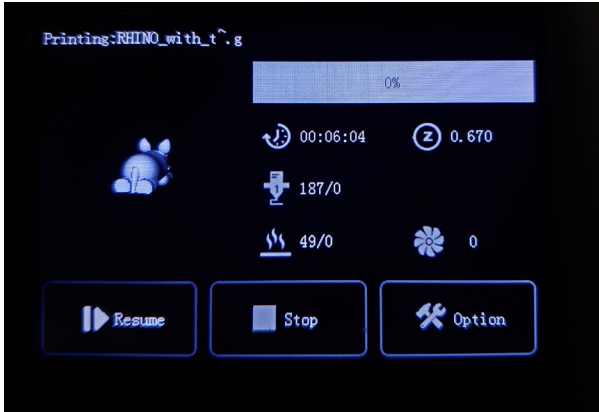
× **Note:** Please always Wifi transfer the file less than 30Mb. Too big file would take very long time to upload.

i Successful, do you want to print 11.gcode?



RESUME PRINTING AFTER POWER BREAK.

If you experience the Power break during printing. Once the power come back. The machine would initiate and back to resume page. You can just click RESUME to resume printing or you can stop to cancel the print.



INTRODUCTION

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.

FROM 3D DESIGN TO PRINTED 3D OBJECT:

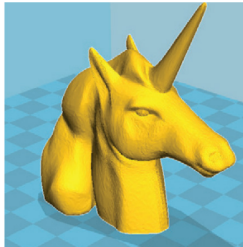
3D Design → Slice to GCODE → Setup 3D Printer → 3D Printing Process → 3D Printed Object



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.

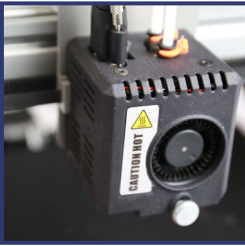
We recommend Cura, as it uses a simple interface and includes a large amount of support material.

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.

SETUP

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 Raise 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

SETUP

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](https://www.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.

FILAMENT TYPES

Filament Type	PLA	ABS	PVA	HIPS
Extruder Temp:	195°C	210°C	185°C	220°C
Heat Bed Temp	60°C	90°C	50°C	60°C
Official Name:	Polyactic Acid	Acrylonitrile Butadiene Styrene	Polyvinyl Alcohol	High Impact Polystyrene
Printing Level:	Basic	Advanced	Advanced	Advanced
Advantages:	<ul style="list-style-type: none">• Bioplastic• Non-Toxic• Less warping issues• Hard or soft/ flexible variants• High print speed	<ul style="list-style-type: none">• Smooth finish• Durable• Ideal for mechanical parts	<ul style="list-style-type: none">• Water soluble• Non-Toxic	<ul style="list-style-type: none">• Lightweight• Cost effective• Dissolvable in Limonene
Disadvantages:	<ul style="list-style-type: none">• Low heat resistance• Easier to break than ABS• Requires thicker walls than ABS	<ul style="list-style-type: none">• Petroleumbased• Nonbiodegradable• Heated Print Bed required• Fumes when printing• Sunlight deterioration	<ul style="list-style-type: none">• Sensitive to high heat. Do not heat over 190	<ul style="list-style-type: none">• Only available in white
Finishing:	<ul style="list-style-type: none">• Sanding possible• Limited gluing	<ul style="list-style-type: none">• Easy sanding• Easily soluble in acetone	<ul style="list-style-type: none">• Gluing Only	<ul style="list-style-type: none">• Gluing Only

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.

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

SETUP

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.

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

USING THE SD CARD

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.

SETUP

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, it is likely that 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

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 to keep 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://www.aliexpress.com/store/120824)

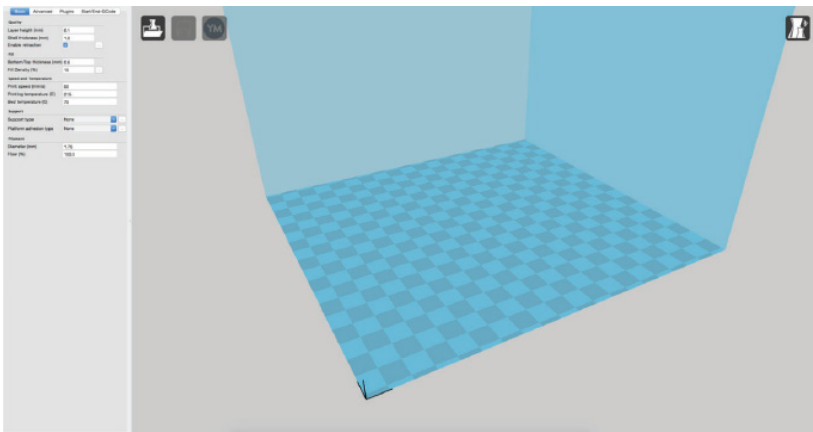
USING CURA

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.

INSTALLING CURA

1. Install the file located on your included MicroSD card



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

USING CURA

INSTALL CURA WANHAO EDITION ONTO YOUR COMPUTER

1. Double click the file: **Wanhao-Cura-18.061**
2. Follow the instruction and click install and next.
3. The software has already embedded all the basic settings for your Wanhao D12 230.



Download WINDOWS



Download MAC

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.

VIEWING AND POSITIONING YOUR 3D MODEL IN CURA

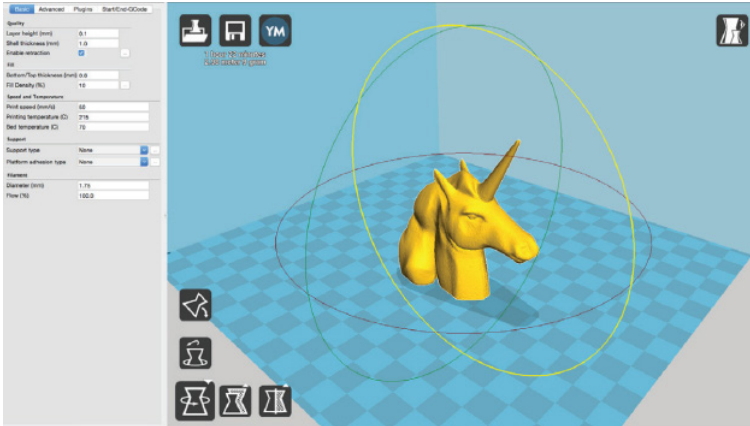
Changing the View:

- **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.

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

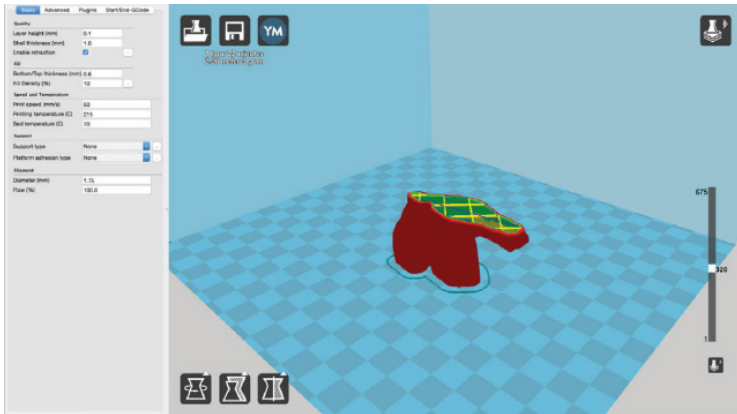
The Reset button will return your model to the original orientation as defined by the CAD program used to create the model.

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

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.

USING CURA

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](https://www.aliexpress.com/store/120824)

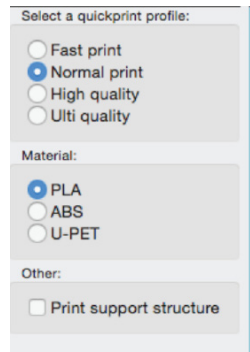
Diameter

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.




USING CURA




HOW TO REALIZE TWO-COLOR PRINTING

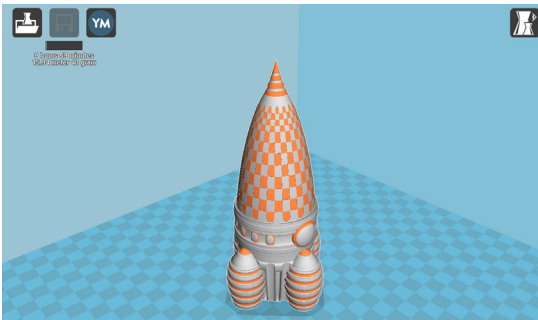
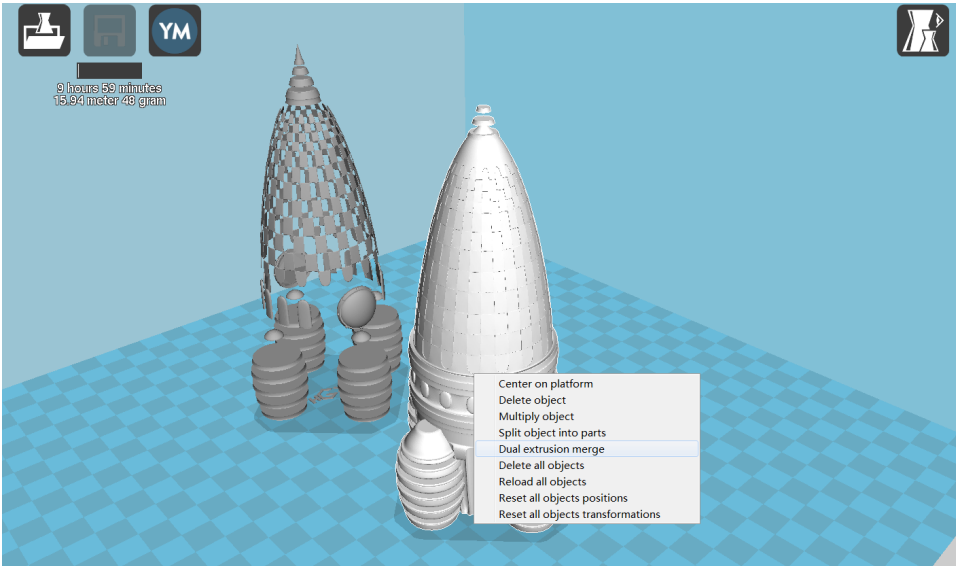
Open your CURa software and import the two-color print model file.

For example,

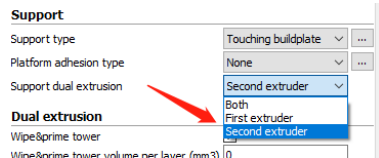
 Hex3D_DualRocket_Part1.stl

 Hex3D_DualRocket_Part2.stl

TIP: We can merge the 2 files by using automatic merging. Click on one file, left click and choose < **Dual Extrusion Merge**>. The 2 file would merge into one.



To print one file with another support material. You need to use double extrusion. Select < Support Dual Extrusion> --Second Extruder. The Extruder 2 would provide support material to the extruder 1.



USING CURA

GENERAL SETUP DETAILS

【SUPPORT】

Support type:

Type of support structure build.

"Touching buildplate" is the most commonly used support setting.

Support	
Support type	None
Platform adhesion type	Raft
Support dual extrusion	Both
Dual extrusion	
Wipe&prime tower	<input checked="" type="checkbox"/>
Wipe&prime tower volume per layer (mm ³)	50
Ooze shield	<input type="checkbox"/>

None:

None does not do any support.

Touching buildplate:

only creates support where the support structure will touch the build platform.

Everywhere:

Creates support even on top of part of the model.

Platform adhesion type:

Different options that help in preventing corners from lifting due to warping.

None:

None does not do any support.

Brim:

Brim adds a single layer thick flat area around your object which is easy to cut off afterwards, and it is the recommended option.

Raft:

Raft adds a thick raster below the object and a thin interface between this and your object. (**Note** that enabling the brim or raft disables the skirt)

Support dual extrusion:

Which extruder to use for support material, for break-away support you can use both extruders. But if one of the materials is more expensive than the other you could select an extruder to use for support material. This causes more extruder switches. You can also use the 2nd extruder for soluble support materials.

【DUAL EXTRUSION】

Wipe&prime tower:

The wipe-tower is a tower printed on every layer when switching between nozzles. The old nozzle is wiped off on the tower before the new nozzle is used to print the 2nd color.

Wipe&prime tower volume per layer (mm³):

The amount of material put in the wipe/prime tower.

This is done in volume because in general you want to extrude 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.

Ooze shield:

The ooze shield is a 1 line thick shell around the object which stands a few mm from the object. This shield catches any oozing from the unused nozzle in dual-extrusion.

USING CURA

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

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.

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.

Section	Parameter	Value
Quality	Layer height (mm)	0.2
	Shell thickness (mm)	1.0
	Enable retraction	<input checked="" type="checkbox"/>
Fill	Bottom/Top thickness (mm)	0.6
	Fill Density (%)	10
Speed and Temperature	Print speed (mm/s)	50
	Printing temperature (C)	245
	Bed temperature (C)	90
Support	Support type	None
	Platform adhesion type	Raft
Filament	Diameter (mm)	1.75
	Flow (%)	100.0

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.

USING CURA

SUPPORT

Support Type

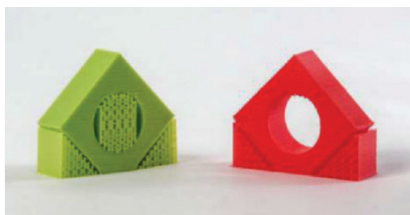
Some models will require support material in order to print properly. This will usually occur when an object has an angle in relation to the Heated Print Bed between 0 to 45 degrees. It is highly recommended to orient your object so that it minimizes or eliminates the need for support.

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.

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.

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.

The screenshot shows the Cura software interface with the 'Advanced' tab selected. The settings are organized into several sections:

- Machine**: Nozzle size (mm) is set to 0.4.
- Retraction**: Speed (mm/s) is 40.0, and Distance (mm) is 4.5.
- Quality**: Initial layer thickness (mm) is 0.3, Initial layer line width (%) is 100, Cut off object bottom (mm) is 0.0, and Dual extrusion overlap (mm) is 0.15.
- Speed**: Travel speed (mm/s) is 150, Bottom layer speed (mm/s) is 20, Infill speed (mm/s) is 0.0, Top/bottom speed (mm/s) is 0.0, Outer shell speed (mm/s) is 0.0, and Inner shell speed (mm/s) is 0.0.
- Cool**: Minimal layer time (sec) is 5, and Enable cooling fan is checked.

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.

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..

USING CURA

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

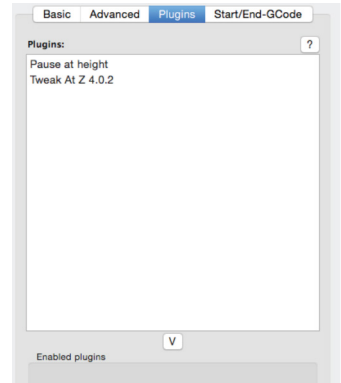
Pause your print at a specified height. You can also specify where to move the print head and how much filament to retract. This will prevent 'blobs' from accumulating on your print while paused. This setting is most commonly used when switching colours of filaments in the middle of a print.

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.



USING CURA

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 oozing, and strings from being deposited on your print.

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.

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.

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.

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.

SOME 3D MODELS AND OTHER MATERIALS ARE REFERENCED FROM THE FOLLOWING WEBSITE

Human Jaw by gebrin
Published on Aug 14, 2015
www.thingiverse.com/thing:968975



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/thing:18672



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



Creative Commons - Attribution - Share Alike

Twisted bracelet by Gepetto

Published on January 25, 2015
www.thingiverse.com/thing:651677





VIDEO GUIDE



DISPOSAL INSTRUCTIONS

Do not throw this electronic device into the waste when discarding. To minimize pollution and ensure utmost protection of the global environment, please recycle or return to Photocentric for recycling.

1.6

Phone: +86-571-6337-9578

(Hours: Mon to Sat from 8:30 AM to 5:00 PM CST China Time)

Mail: HANGZHOU BEAN 3D PRINTER CO.,LTD

No. 18 Zhen Shan Road, Chang Kou Town, Fuyang District,
Hangzhou City, Zhejiang Province, China

Email: support@wanhao3dprinter.com

Website: www.wanhao3dprinter.com

