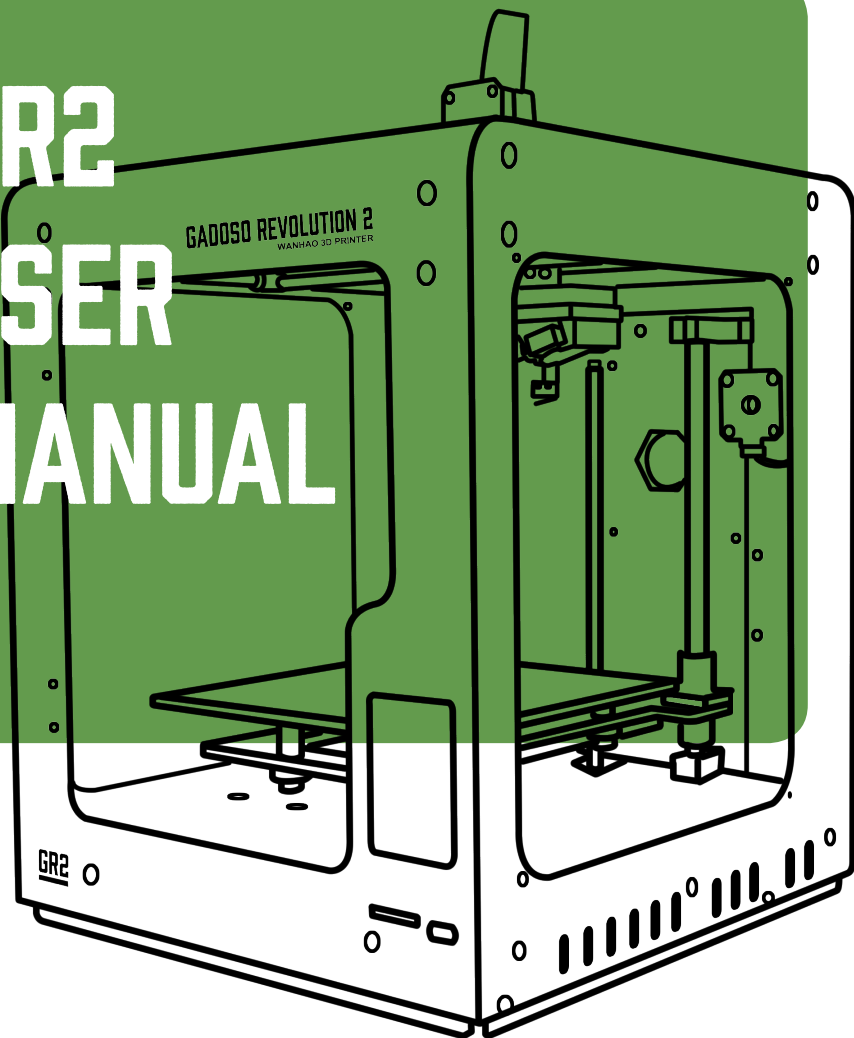


GR2 USER MANUAL

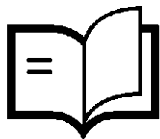


GADOSO

APP REMOTE OPERATION

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 pre-assembled WANHAO GR1 printer is covered by a 12-month warranty for USER over the world.

The GR1 provides a 12-month warranty on the functionality of individual parts for end-users over the world.

Wear and tear category components and consumables are excluded from this warranty. The warranty period begins with the day when the customer receives the goods.

WARNING: The exposure display and the FEP foil fall into the category of normal wear and tear components, which are not covered by our standard warranty. Instead, the warranty of the exposure display covers a maximum of 500 hours of printing - the SLA 3D printing process on the GR1 is considered the standard way of using and wearing the original display. The warranty of the FEP foil covers 10 cycles (print jobs). When used properly in accordance with the instructions in the official guides (such as this handbook), the life expectancy of the FEP foil is several times longer.

Neither the Seller's liability for defective products nor the quality warranty applies to printers or parts damaged by handling, which is in conflict with the instructions and recommendations in the official manuals and guides, or improper handling caused by unofficial modifications, such as modifications to hardware or software.

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: ZHEJIANG WEIBIN 3D TECHNOLOGY 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 allow resin or any liquids to get inside the chassis, wipe up any liquid spills immediately.
- Do not use solvents to clean the printer as they may damage the screen. Use a little detergent and warm water to clean any resin spills.
- 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.

SAFETY GUIDELINES

IRRITANT



- Keep the printer and resins out of the reach of children.
- The resin is an irritant to skin and eyes. Always wear gloves when coming into contact with the liquid resin. Always use in a well ventialed room.
- Resins are classified as irritants, not harmful, but in exceptional circumstances people can be sensitive to the resin and develop a skin irritation or rash. Avoid this possibility by always wearing gloves and avoid breathing fumes.
- The printer should be operated on a stable and level surface, preferably away from direct ambient light.

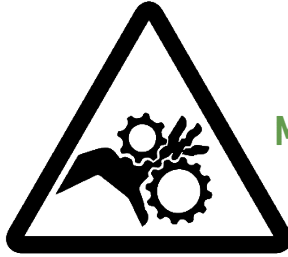
Please notice that the latest instructions will always be available from:

www.wanhao3dprinter.com

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.



Always wear gloves while handling resin or printed objects.



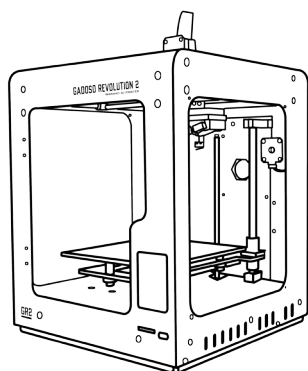
Always be careful when using the scraper. Never direct the scraper towards your fingers.

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

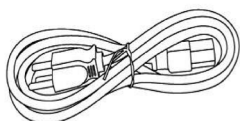
- The printer must not be exposed to water or rain, or the printer will be damaged.
- The printer is designed to be used in ambient temperatures, ranging from 15°C to 30°C.
- The printer is designed to be used in humidity levels ranging from 20% to 50%.
- Operating the printer outside the recommended temperatures and humidity levels may result in low quality print models.
- Eye protection during cleaning or sanding of the printed models is recommended to protect your eyes from small airborne particles.

CONTENTS

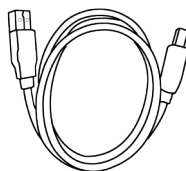
The box contains



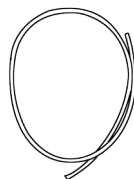
GR2 3D PRINTER



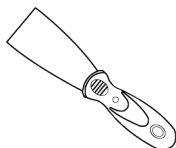
POWER CABLE



USB CABLE



WIRE FEEDING
TUBE



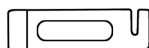
SHOVEL



4G U DISK



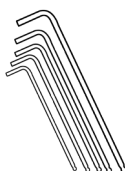
FILAMENT SHAFT



WIRE SUPPORT



DIGITAL PEN



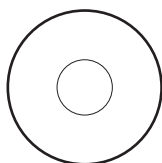
HEXAGON WRENCH



CLEANING HOOK



TWEEZERS



1KG FILAMENT



HOT BED
PRINT PAD

TECHNICAL SPECIFICATIONS

Software	WANHAO Cura
Extruder system	MK10
Material support	ABS, PLA
Max printable area	200 * 200 * 175 mm
Filament diameter/light	1.75MM
Max printing speed	30 mm/s-100 mm/s
Case material	Steel case
Extruder quantity	1
Net weight(kg)	13kg
Gross weight(kg)	14.8kg
Accuracy	X:0.05 mm Y:0.05 mm Z:0.05mm
Packing size(cm)	46CMx44CMx55.5cm
Layer thickness	100-400 um
Platform	Aluminium platform W/ heating
Certification	CE,CB, FCC
LCD display size	75*51 mm
Language	Chinese, English, Netherlands, French, German, Italian, Spanish
Power supply unit	Input 110 AND 250V(optional), 50/60Hz, 350W, 1.9A. Output 24V 4.5A

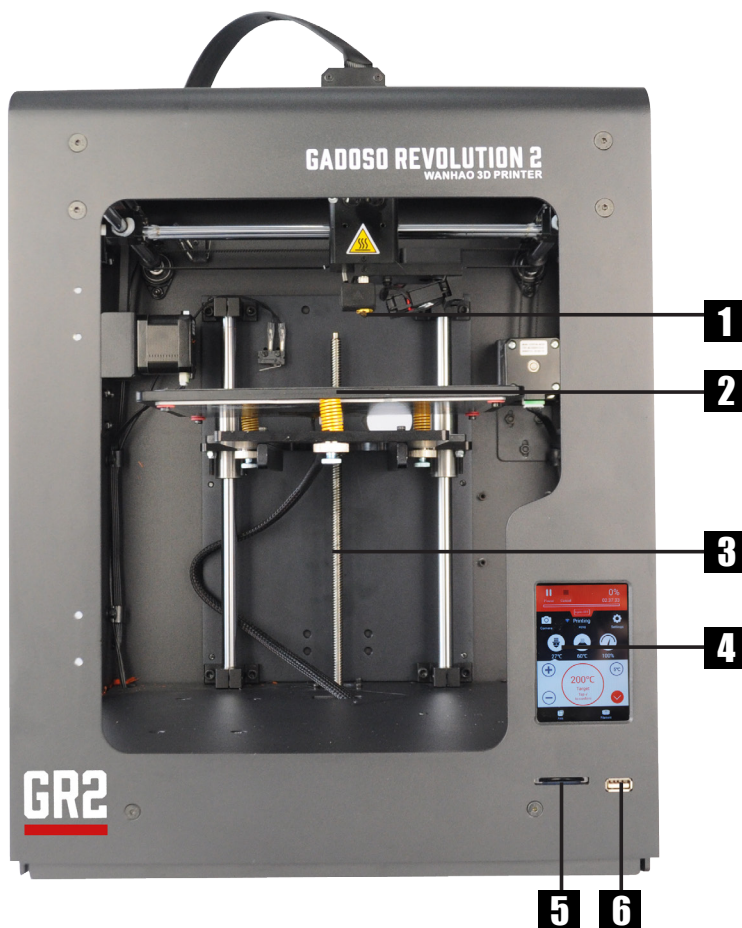
Read this manual carefully, it
will bring you great convenience

USING GR2 FOR THE FIRST TIME



PRODUCT OVERVIEW

Printer Front View

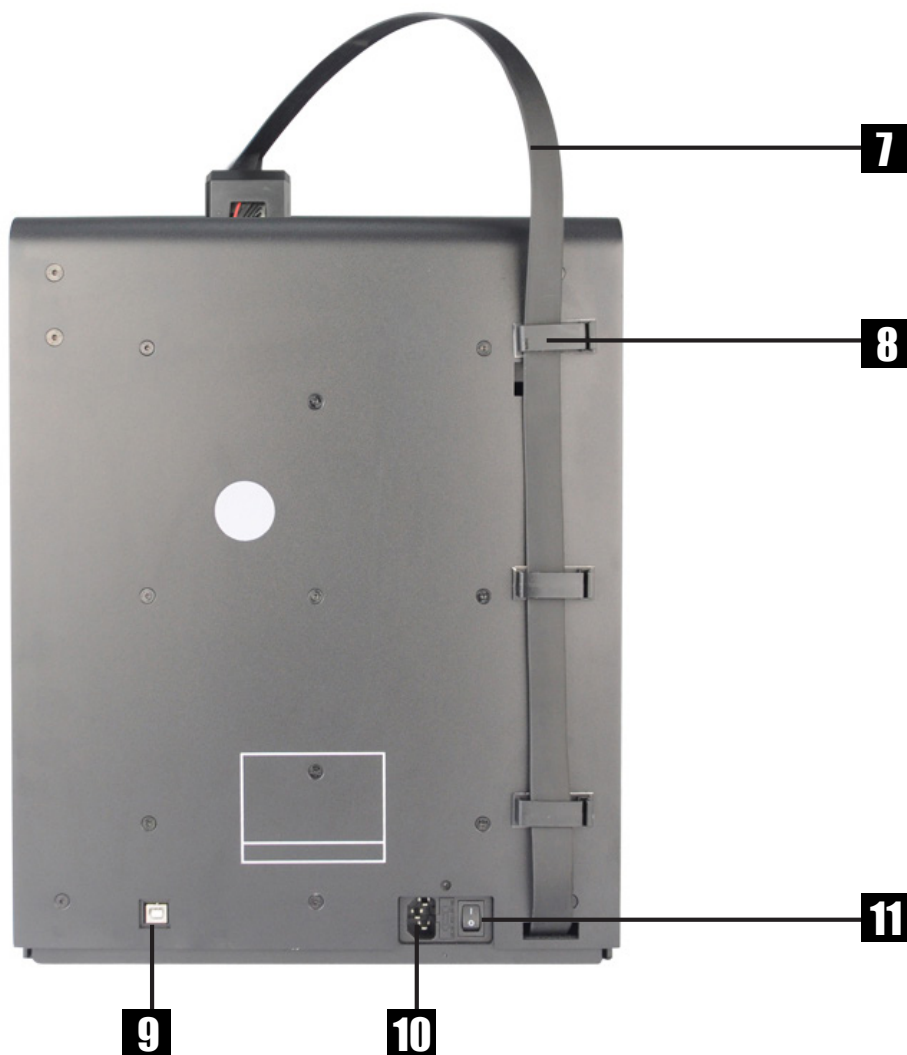


- 1. MK11 Extruderv
- 2. Printing Mat
- 3. Double locking
Z Axis screw

- 4. LCD Screen
- 5. SD Card slot
- 6. USB Card slot

PRODUCT OVERVIEW

Printer Rear View



7. Cable Quick Coupler

8. Cable Locker

9. USB Socket

10. Power Socket

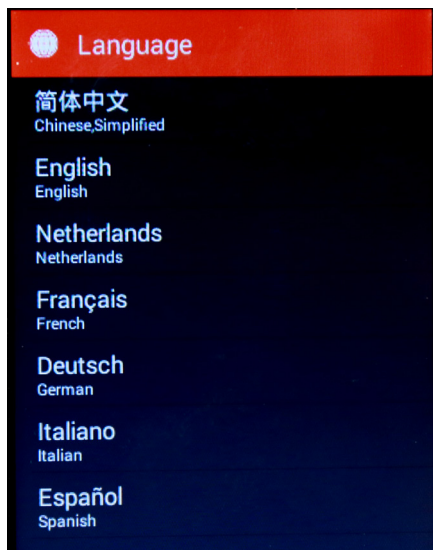
11. Power Switch

FIRST COMMISSIONING

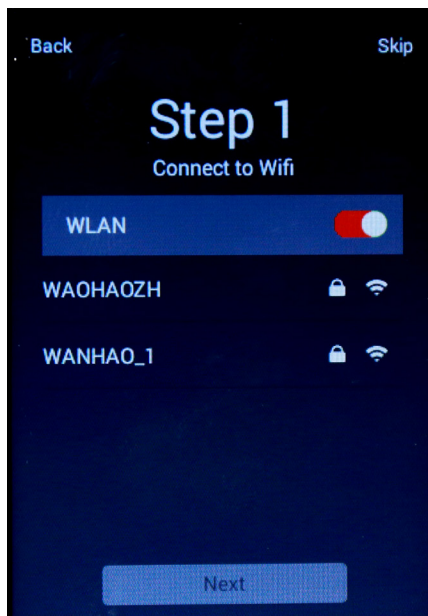
INITIAL SETTINGS



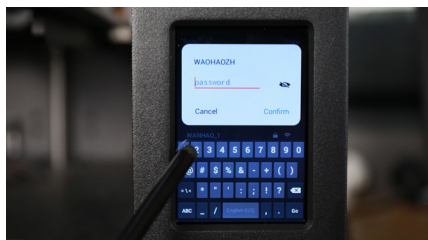
Plug in and turn on the power, When "I" is on, the printer is officially started.



Initialize the setting language. Please select the language that suits you in 7 languages. Go to the next step.

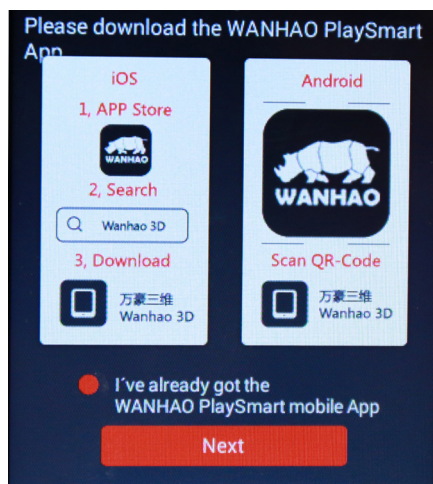


Step 1: Wifi connection options will appear on the current page. This step is critical and will be used later when your phone is connected to the printer. Please select the Wifi option closest to you, ready to enter your password.

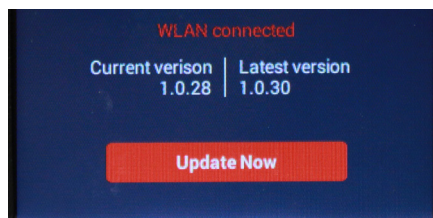


Tip: Please use a digital pen to operate more convenient for your chosen target.

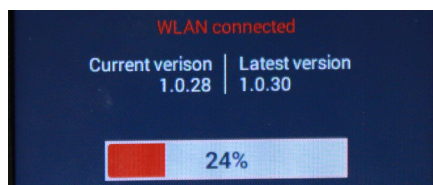
Go to the next step after connecting to Wifi.



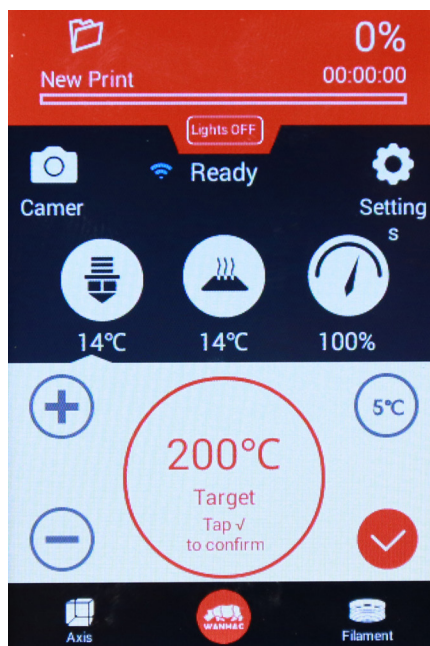
Step2: Here will prompt "Please download the WANHAO playSmart APP", which is an important reminder about how the mobile phone will operate the printer in the future. You can download this software through the mobile APP, then click on I've already got the WANHAO PlaySmart mobile APP.



Step 3: Firmware update interface. If you have already connected to Wifi, you need to pay attention to whether the version number is the latest. If not, you need to update the firmware before proceeding to the next step.



Tip: Do not turn off the power during firmware update.



After the firmware update, the system will restart automatically. After waiting a moment, you will get the GR2 UI main page. Congratulations on your successful initialization.

Tip: Because the APP usage permission of each machine is only bound to one mobile phone, if you want your family to share and use your printer, you need to enter the setting interface, select Reset, and restore the factory settings.

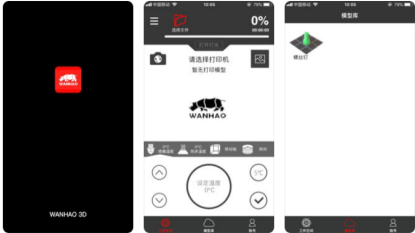
FIRST COMMISSIONING

APPLICATION INSTALLATION

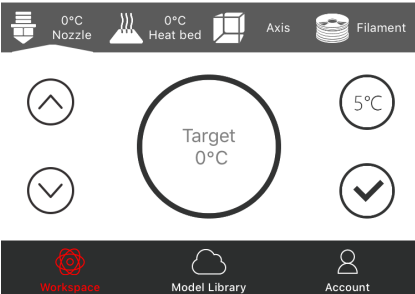
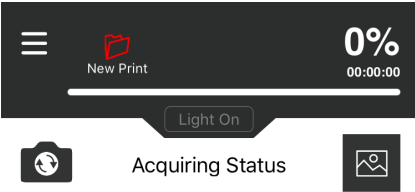


WANHAO Smart your...
Maker life so easy!

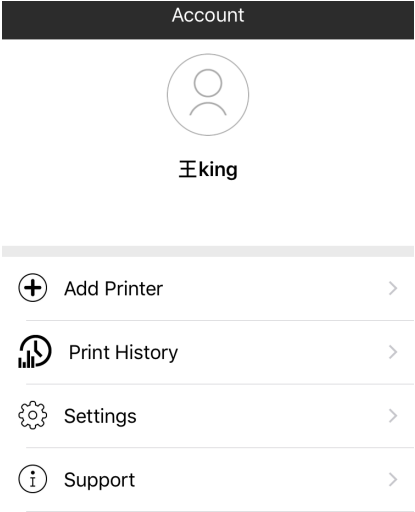
打开



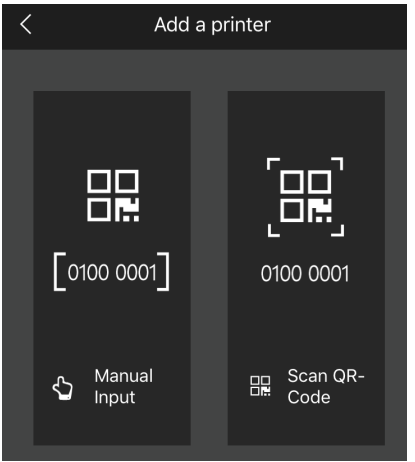
Open the mobile app store and enter WANHAO to search for the black logo download.



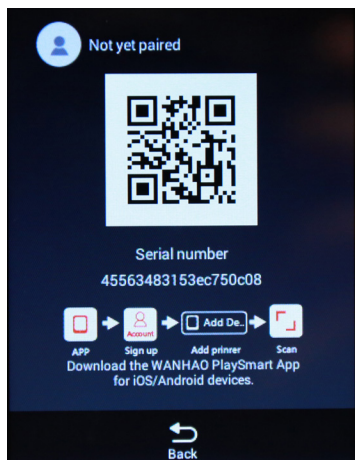
Open the mobile APP to present the WANHAO application interface.



Select Account login account password login interface.



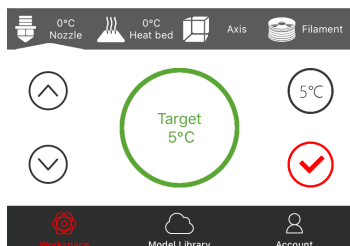
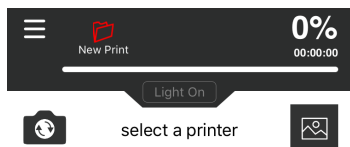
Select APP printer to load the scanning interface.



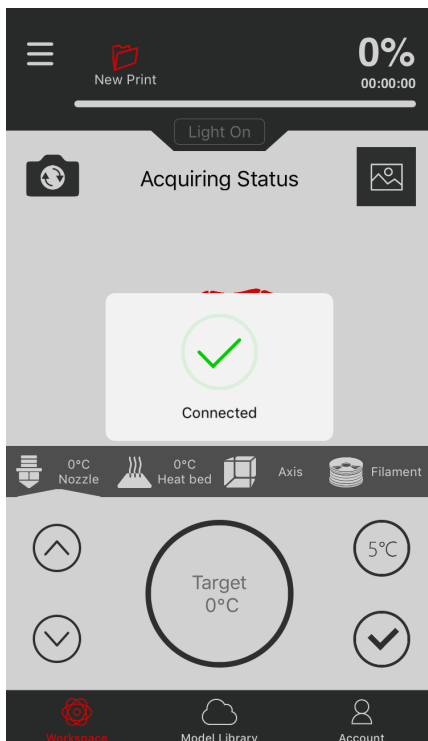
Find on GR2 【Settings - pair the printer】 to get the QR code interface and serial number.

How to connect a printer?

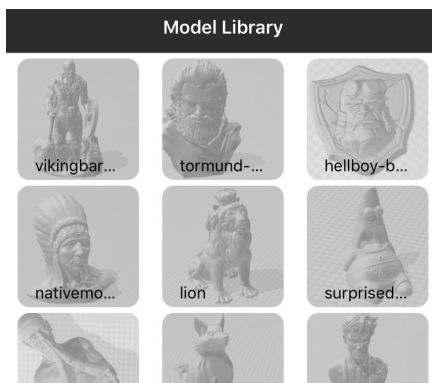
1. Scan QR Code with your phone.
2. Manual Input serial number that comes with the printer manually.



Sign in to the main page.

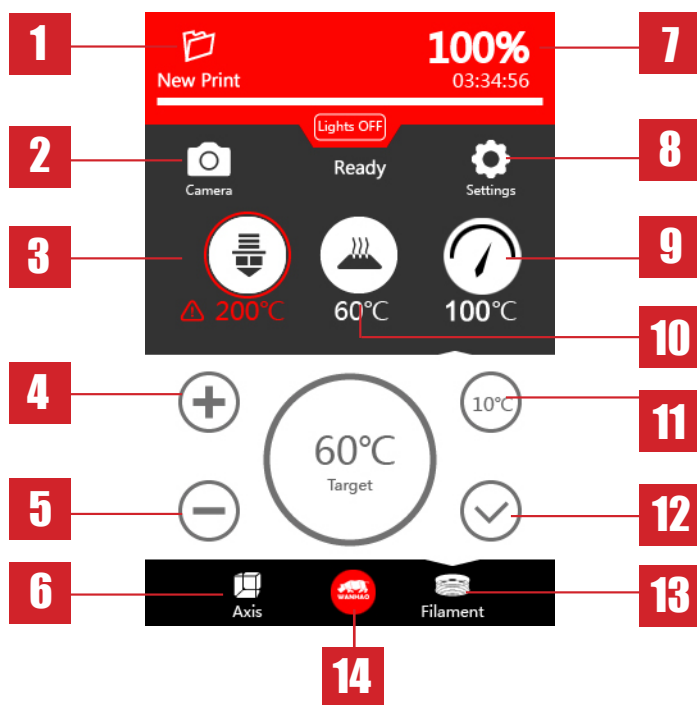


Open Workspace → select a printer and connect the name you noted. The system will directly match the printer to your mobile phone.



There are a variety of sliced models in the application to make your first print very convenient.

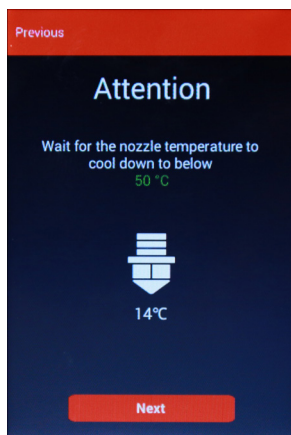
INTERFACE INTRODUCTION



1. File induction, click to select SD card import or USB import model file.
2. File induction, click to select SD card import or USB import model file.
3. Print head thermal control is always recorded, and the adjustment interface is switched.
4. Increase data key position.
5. Reduce data keys.
6. Adjust xyz axis movement data.
7. Display print time and print completion rate.
8. Setting interface is the main interface key.
9. Printing speed adjustment switch key position.
10. Printing hot bed adjustment switch key position.
11. Adjustment data, temperature display temperature, speed display percentage.
12. Determine the selected data and start execution.
13. Enter filament adjustment interface.
14. Real-time print interface, display print temperature, hot bed temperature and fan speed, you can view the progress and pictures of print files.

FIRST LEVELING

BED LEVELLING



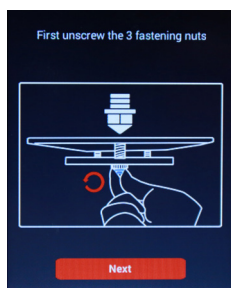
After leaving the factory, the GR2 is generally leveled and debugged by WANHAO inspection personnel, and there is a lock under the printing hot bed.

If you have used it for a long time in the future and need to fine-tune it, you can use the following methods.

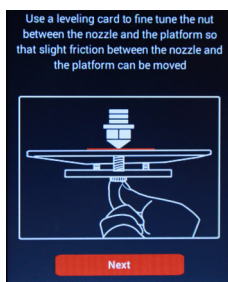
First open the **settings** - select **Bed Levelling**.

The picture shows that the thermal head has not yet started to be heated. If you feel you need to re-adjust your printing distance during printing, please follow the instructions.

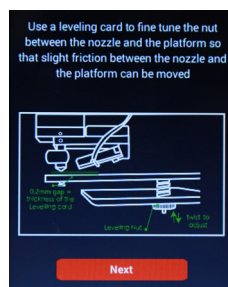
Note: While waiting for the print head to cool down, do not touch it to avoid burns.



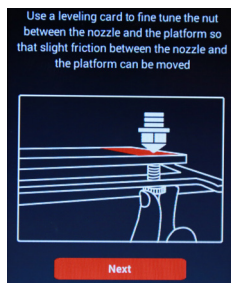
1. First unscrew the 3 fastening nuts.



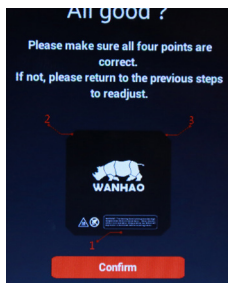
2. The print head will move to the middle. At this time you need to use a card and turn the nut to adjust the distance between the thermal head and the printer.



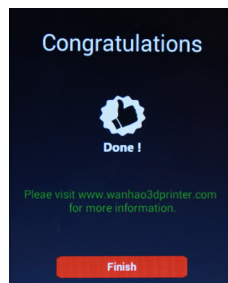
3. After the print head moves to the left rear, adjust the distance in the same way.



5. After the print head moves to the rear right, adjust the distance in the same way.



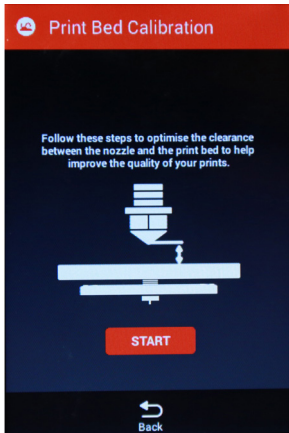
5. After you have adjusted the distance once, the system will remind you to confirm that you have adjusted the 3 corners to a level.



6. Congratulations on your successful commissioning.

PRINT BED CALIBRATION

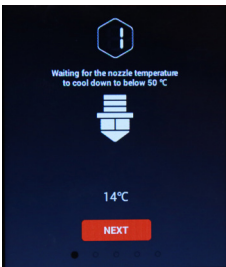
Z-AXIS COMPENSATION



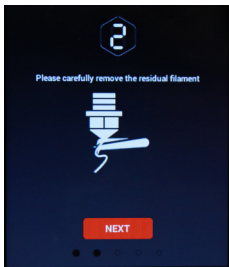
There may be some special situations during the printing process, but you don't want to level it over and over again. At this time, we can use Z-axis compensation, which greatly saves your time, and you can also fine-tune your leveling. Get more accurate distance.

Preferred Selection **Setting - Print Bed Calibration**.

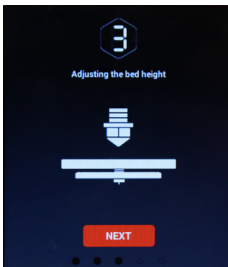
Click to start.



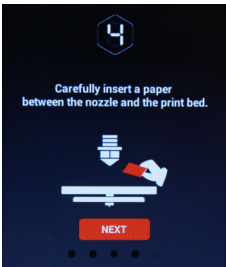
1. The picture shows that the thermal head has not yet started to be heated. If you feel you need to re-adjust your printing distance during printing, please follow the instructions.



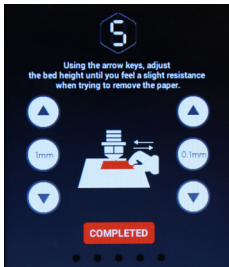
2. Please carefully remove the residual filament.



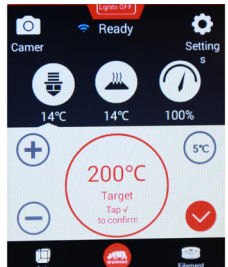
3. Adjusting the bed height



5. Carefully insert a paper between the nozzle and the print bed.



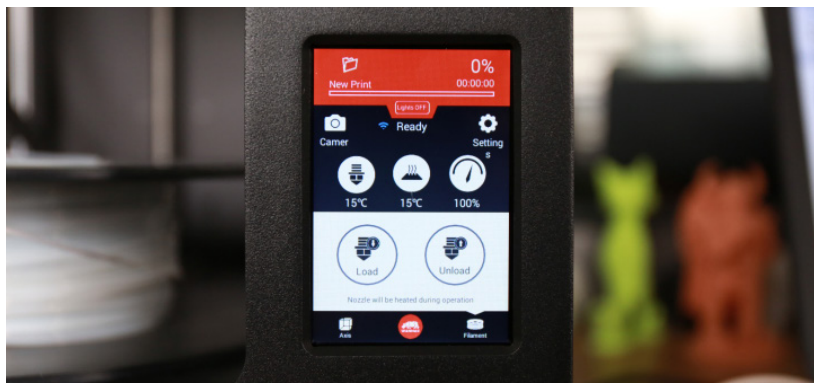
5. Using the arrow keys, adjust the bed height until you feel a slight resistance when trying to remove paper.



6. When everything is done, you will return to the main menu.

READY TO PRINT

INSTALL THE FILAMENT



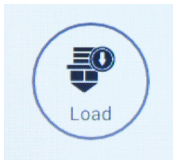
Click to enter the filament icon in the lower right corner of the main interface.



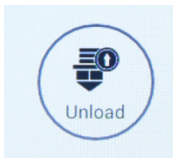
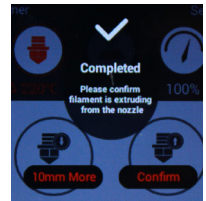
Please make sure you have a filament spool positioned over the spool holder, guided along the filament guide, with the filament in counter clockwise direction.



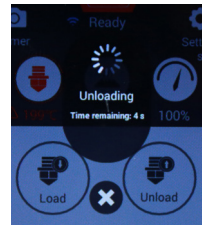
INSTALL THE FILAMENT



Click the "Load" button and a dialog box will appear asking you to wait for the thermal head temperature to rise to the specified temperature. After waiting for the double gears to feed, the system will prompt you if you need to continue loading 10mm.



Click the "Unload" button and a dialog box will appear asking you to wait for the thermal print head temperature to rise to the specified temperature. Wait for the double gear, the system will send the filament directly.



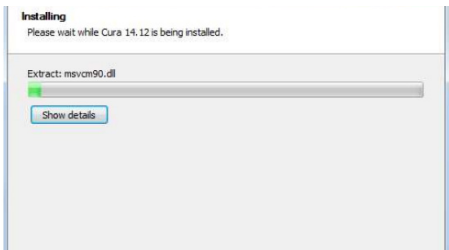
After your filament can be finished, then you can directly use the mobile phone application to start the first printing of Unicom printer, or you can use the file generated by CURA, use SD card or U disk to save and print.

INSTALL CURA

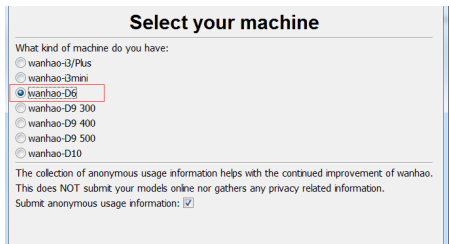
DOWNLOAD AND INSTALL



Log in to www.wanhao3dprinter.com WANHAO official website, find the download, find the interface of Duplicator 6, you can find the **CURA 18.04** version, although the time changes, you may need to update the software in the future.



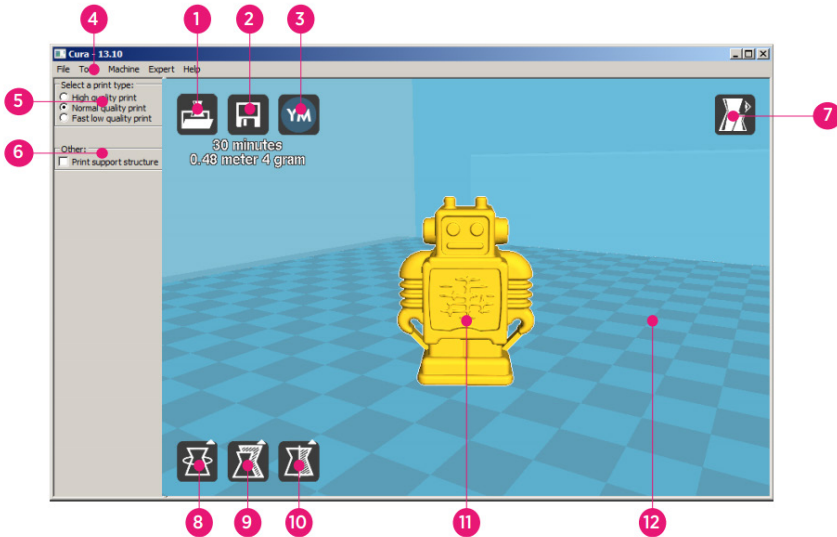
The downloaded software will be installed step by step, select the language you need, and click OK. Congratulations on your successful installation of Cura. You can make your favorite model file.



In the model settings, WANHAO recommends that you continue to use the D6's settings panel. Open the settings panel, select D6, the system will automatically jump to the page after you choose.

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

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

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.

8. Rotate model options

Option to change the rotation of the object you like to print

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:



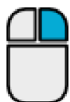
Leftmouse button

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.



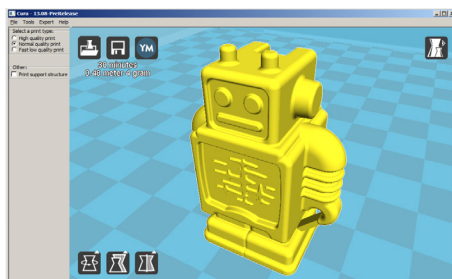
Rightmouse button + Shift

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.

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.

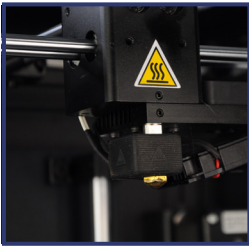
We recommend Cura, as it uses a simple interface and includes a large amount of support material.

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 it is 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, 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

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

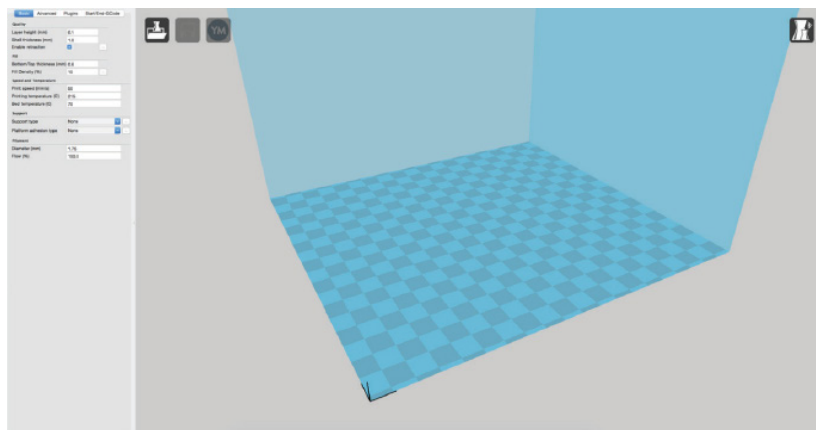
USING CURA

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 or download Cura from [http://www.wanhao3dprinter.com/Down load Software/Duplicator i3](http://www.wanhao3dprinter.com/Down%20load%20Software/Duplicator%20i3)



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: 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.

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.

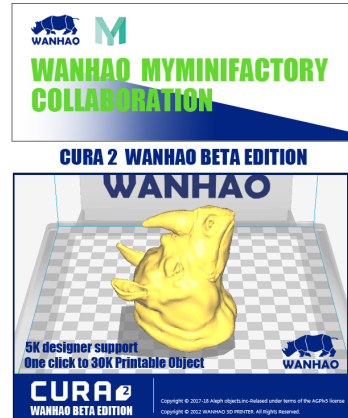
VIEWING AND POSITIONING YOUR 3D MODEL IN CURA

Changing the View:

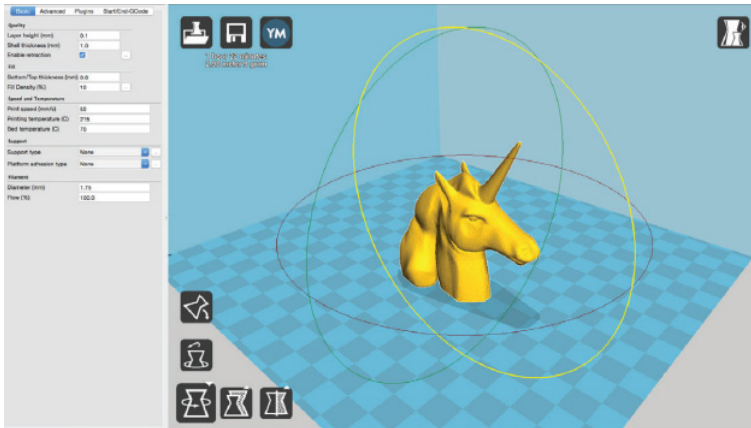
- **Zooming In/Out:** Use the mouse scroll wheel
- **Rotating the View:** Hold down the right mouse button and move the mouse

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

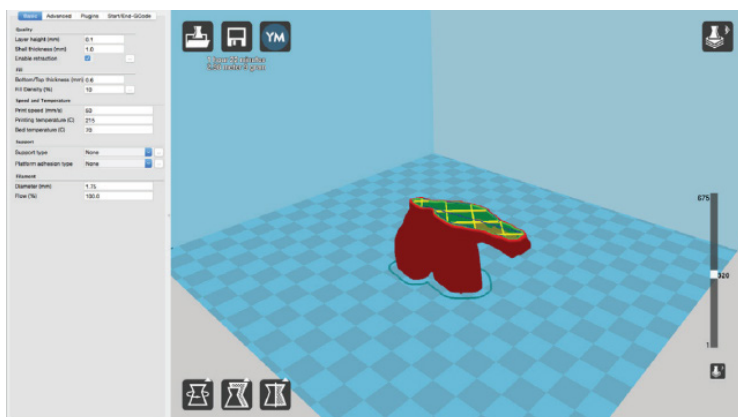
The Reset button will return your model to the original orientation as defined by the CAD program used to create the model.

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)

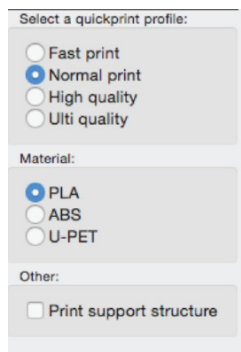
Diameter

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.

A screenshot of the Cura software's 'Quickprint' settings window. The window has a light gray background and a thin blue border. At the top, it says 'Select a quickprint profile:'. Below this are four radio button options: 'Fast print', 'Normal print' (which is selected with a blue dot), 'High quality', and 'Ulti quality'. Below these is a section labeled 'Material:' with three radio button options: 'PLA' (selected with a blue dot), 'ABS', and 'U-PET'. At the bottom, there is a checkbox labeled 'Print support structure' which is currently unchecked.

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

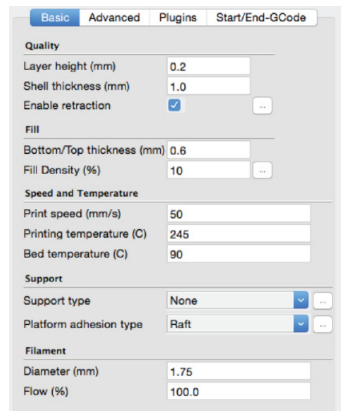
QUALITY

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.



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

- Quality**
 - Layer height (mm): 0.2
 - Shell thickness (mm): 1.0
 - Enable retraction: ☒
- 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

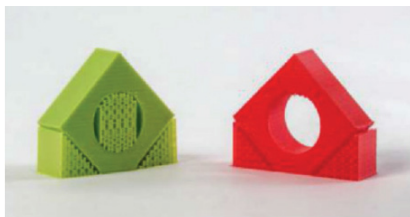
Some models will require support material in order to print properly. This will usually occur when an object has an angle in relation to the Heated Print Bed between 0 to 45 degrees. It is highly recommended to orient your object so that it minimizes or eliminates the need for support.

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.

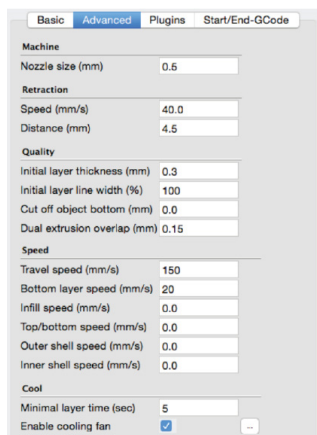
RETRACTION

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 'Advanced' tab in the Cura software interface. It contains various settings for the printer, organized into sections: Machine, Retraction, Quality, Speed, and Cool. Each section has several parameters with input fields and dropdown menus. The 'Machine' section includes 'Nozzle size (mm)' set to 0.5. The 'Retraction' section includes 'Speed (mm/s)' set to 40.0 and 'Distance (mm)' set to 4.5. The 'Quality' section includes 'Initial layer thickness (mm)' set to 0.3, 'Initial layer line width (%)' set to 100, 'Cut off object bottom (mm)' set to 0.0, and 'Dual extrusion overlap (mm)' set to 0.15. The 'Speed' section includes 'Travel speed (mm/s)' set to 150, 'Bottom layer speed (mm/s)' set to 20, 'Infill speed (mm/s)' set to 0.0, 'Top/bottom speed (mm/s)' set to 0.0, 'Outer shell speed (mm/s)' set to 0.0, and 'Inner shell speed (mm/s)' set to 0.0. The 'Cool' section includes 'Minimal layer time (sec)' set to 5 and 'Enable cooling fan' checked.

Section	Parameter	Value
Machine	Nozzle size (mm)	0.5
Retraction	Speed (mm/s)	40.0
	Distance (mm)	4.5
Quality	Initial layer thickness (mm)	0.3
	Initial layer line width (%)	100
	Cut off object bottom (mm)	0.0
	Dual extrusion overlap (mm)	0.15
Speed	Travel speed (mm/s)	150
	Bottom layer speed (mm/s)	20
	Infill speed (mm/s)	0.0
	Top/bottom speed (mm/s)	0.0
	Outer shell speed (mm/s)	0.0
	Inner shell speed (mm/s)	0.0
Cool	Minimal layer time (sec)	5
	Enable cooling fan	<input checked="" type="checkbox"/>

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

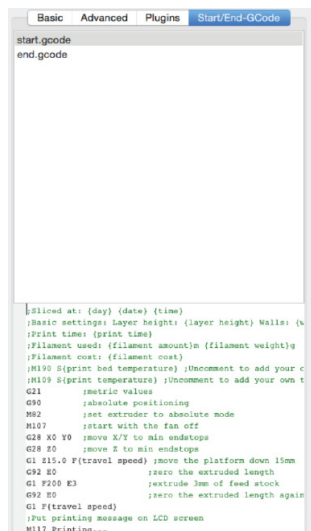
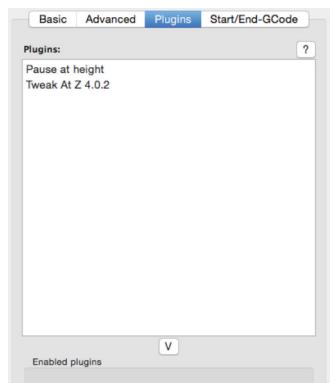
Pause your print at a specified height. You can also specify where to move the print head and how much filament to retract. This will prevent 'blobs' from accumulating on your print while paused. This setting is most commonly used when switching colours of filaments in the middle of a print.

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 ooze, 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

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



Twisted bracelet by Gepetto

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



Creative Commons - Attribution - Share
Alike



MAKE THE MOST SINCERE GOOD PRODUCT



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.

1.0

Phone: +86-571-6337-9578

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

Mail: ZHEJIANG WEIBIN 3D TECHNOLOGY CO.,LTD

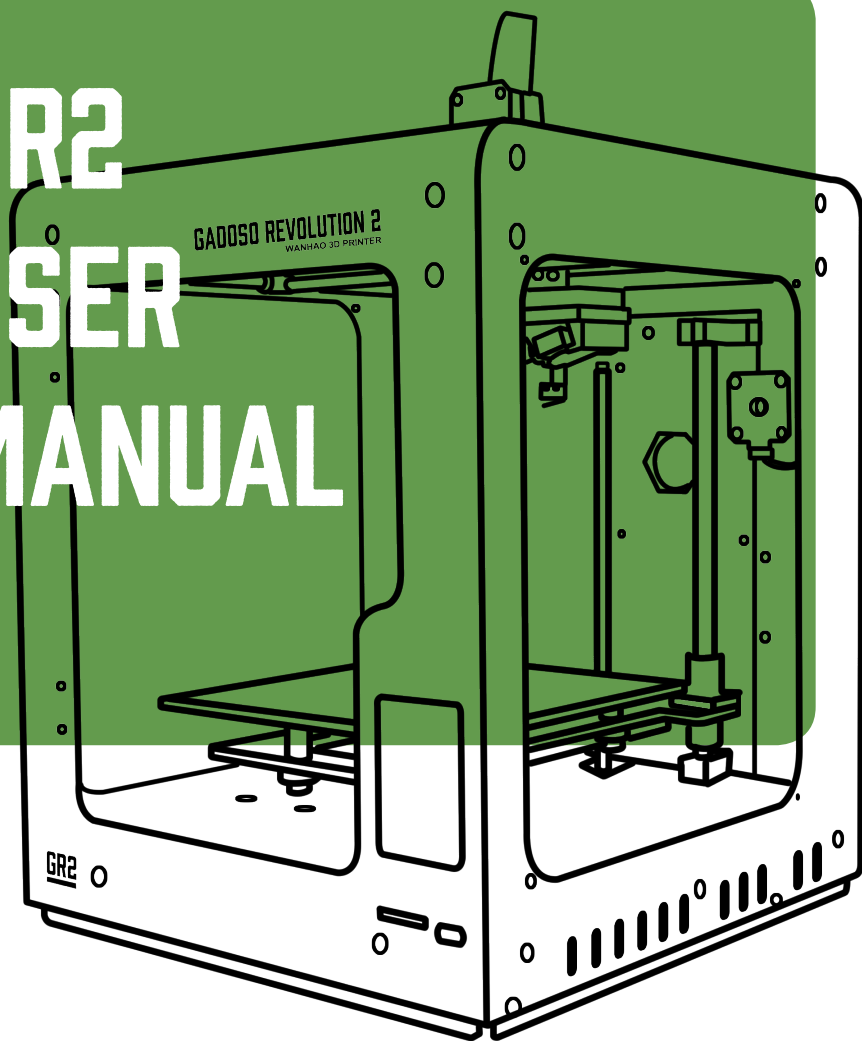
No. 18 Zhen Shan Road, Chang Kou Town, Fuyang District, Hangzhou
City, Zhejiang Province, China

Email: support@wanhao3dprinter.com

Website: www.wanhao3dprinter.com



GR2 USER MANUAL



GADOSO

APP REMOTE OPERATION