

# LUGO G3

## Quick Start Guide



# CONTENT

## 01 LUGO G3 Overview

Specifications	1
What is in the box?	3
Safety and compliance	4
Diagram	5

## 02 Install the software

Installing LUGO Cura	7
Print Mode (Single)	9
Print Mode (Dual)	11
Slicing preview	13

## 03 UI Menu structure

Home	15
Preheat	16
PIT Zone	17
More	18
Temp	19
Move	20
XYZ Home	21
Fan	22
Feeding	23
Setting	24

## 04 Preparing a print

Leveling	25
Preheat	29
Checking Feeding	31
Setting the bed	33
Stting the Nozzle	36
Setting the filter pan	37
Drybox switch adjustment	38

## 05 Print

Print	39
While printing	40
Separating the output	41

## 06 Maintenance

Nozzle management	43
Extruder management	53
Flter management	55
Diagnosis of poor extrusion	57
Bed set-up problem	59

## Specifications

### LUGO **G3**<sup>HT</sup>

<b>Printing Type</b>	FFF
<b>Printing Head</b>	DCH Dual
<b>Build Volume(WxDxH)</b>	190 x 165 x 180 / 15kg
<b>Machine Size(WxDxH)</b>	457 x 370 x 532
<b>Positioning Resolution</b>	12.5, 12.5—, 20 micron
<b>Moving Speed</b>	up to 200mm/s
<b>Printhing Speed</b>	up to 20mm <sup>^3</sup>
<b>Nozzle temperature</b>	up to 400C
<b>Bed temperature</b>	up to 165C
<b>Leveling</b>	Gauge assisted leveling
<b>Platform</b>	Spring plate
<b>Slicing Software</b>	LUGO cura
<b>Filter type</b>	Carbon + Hepa filter
<b>Supported Materials</b>	1.75mm Filaments up to 400C

### LUGO **G3**<sup>MAX</sup>

<b>Printing Type</b>	FFF
<b>Printing Head</b>	DCH Dual
<b>Build Volume(WxDxH)</b>	330 x 240 x 300 / 22kg
<b>Machine Size(WxDxH)</b>	595 x 450 x 655
<b>Positioning Resolution</b>	12.5, 12.5—, 20 micron
<b>Moving Speed</b>	up to 200mm/s
<b>Printhing Speed</b>	up to 20mm <sup>^3</sup>
<b>Nozzle temperature</b>	up to 285C
<b>Bed temperature</b>	up to 130C
<b>Leveling</b>	Gauge assisted leveling
<b>Platform</b>	Spring plate
<b>Slicing Software</b>	LUGO cura
<b>Filter type</b>	Carbon + Hepa filter
<b>Supported Materials</b>	1.75mm Filaments up to 285C

## What is in the box?



Power cable



2 filament (LUTAN 1kg)



5 nozzles



Φ1.5 push pin(150mm)



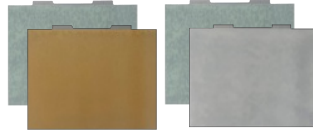
Nozzle clamp  
(including spring  
module)



USB



Carbon card



Build plate  
Epoxy, PA, PP, HT epoxy



Hex wrench



Scraper



Tweezers



Φ0.38  
needle



Gel for bed adhesion



Leveling gauge

## Safety and compliance

- Excessive current flowing through power lines or outlets can overheat and cause a fire hazard.
- Do not touch hot parts such as the nozzle part and heating bed while the machine is on. Touch after 15 minutes after power off.
- When removing material, be sure to adjust it with the fitting handle without pushing it out by hand. It may cause malfunction.
- Do not place in a humid place as it may cause deformation or malfunction of the product.
- Do not insert objects or body parts between parts while the printer is in operation.
- Always use in an open or well-ventilated area.
- Always wear gloves when handling the output plate. Corners may be sharp.



The scraper has sharp blades, so be careful when using it. Do not use it for any purpose other than removing sculptures. In particular, do not make physical contact with the blade part.

## Diagram

1. HEPA/Carbon filter
2. Teflon tube
3. Nozzle
4. Core module
5. Drybox fan switch
6. Print bed
7. USB port
8. Touch screen
9. Dry box
10. Material insert
11. LED dimmer switch



12. Teflon tube
13. Filter fan switch
14. Power switch
15. Power socket
16. PC connect port



## Install LUGO cura

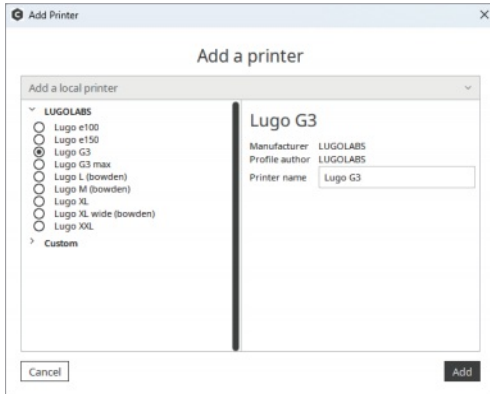
LUGO Cura is a user-friendly, free 3D printing software designed to prepare 3D models for printing. It emphasizes convenient settings for a variety of materials, with a focus on compatibility with DCH heads. Custom printers and previous versions of the LUGO series can also be used with this software.

You can download the LUGO cura software installation and usage manual on the website <https://www.lugolabs.xyz/lugocura>

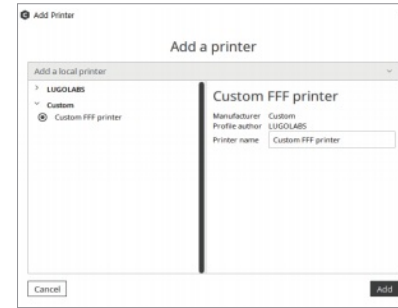
For details, please refer to the LUGO Cura manual.

## Add Printer

When you open LUGO cura for the first time after installation, a message window for selecting a printer appears. If you choose LUGO G3, G3 single and G3 dual will be added.



If you have other FFF 3D printers other than the LUGO series, you can also add custom printers.



## How to open a file

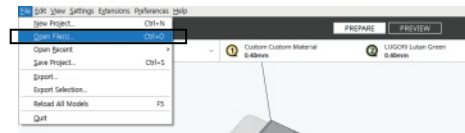
### • Open button

Click the 'Open File' button in the top left corner of the stage menu.



### • file menu

Click 'File > Open File' in the LUGO Cura menu at the top left of the open screen.



## Single print

### 1 Select a printer

The printer selection panel is located on the second button of the Stage menu. Select the printer that corresponds to single printing.



### 2 Nozzle and material settings

The third button on the Stage menu has the Configuration panel. Click to display the current nozzle holes and material settings in the panel.



### 3 Print setting

The rightmost button of the Stage menu contains a print settings panel that contains all the settings that define how to print. Opens in Recommended mode by default. This mode is ideal for fast printing with an optimized print profile.

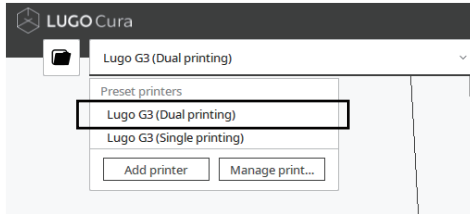


! For details, please refer to the LUGO Cura manual.

## Dual print

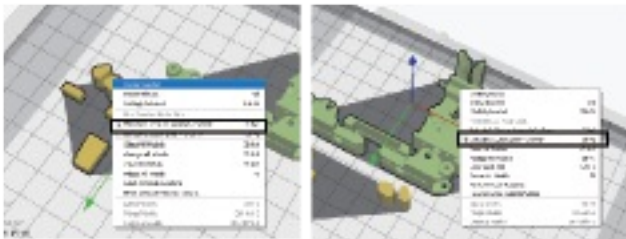
### 1 Select a printer

The printer selection panel is located on the second button of the Stage menu. Select the printer that corresponds to dual printing.



### 2 To set the output nozzle by object

Right-click each model to set the output nozzle.



### 3 Nozzle and material settings

The third button in the Stage menu contains the Configuration panel. Click to display the current nozzle aperture and material settings on the panel. Set each nozzle.



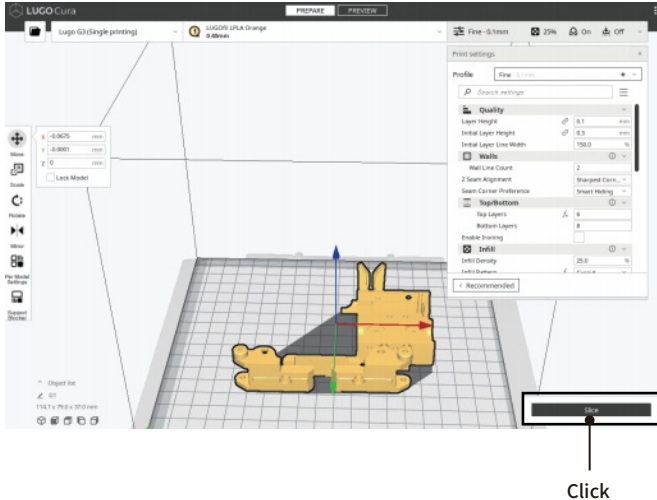
① You can use the main material for the left nozzle (No. 1) and the sub material for the right nozzle (No. 2). Supports or rafts are created as sub-materials.



## Slicing preview

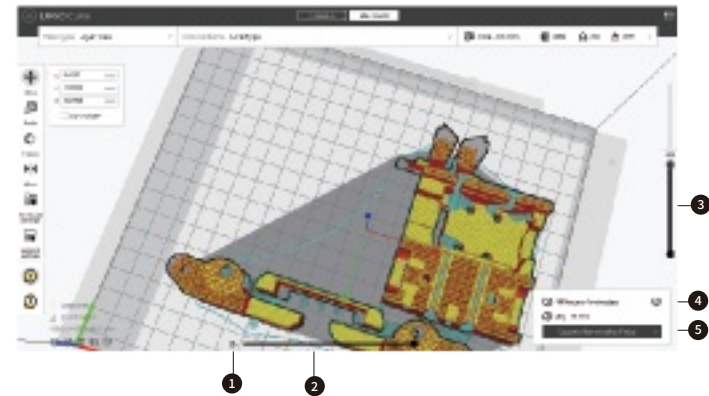
### 4 Slice

Press the 'Slice' button in the lower right corner when printing setup is complete. When the process is complete, a 'Preview Button' will be displayed immediately, when clicked, to go to the Preview step. Previewing is a very important process for determining what the inside of a print looks like and how it will be printed. Use the Layer slider and the Simulation view to identify important parts of the 3D slice.



### 5 Preview and save

When the slicing process is complete, click the Save to File button in the lower right corner. You can save files to a removable disk, such as a USB that came with your computer or printer. Safely remove the USB from your computer and insert it into the printer.

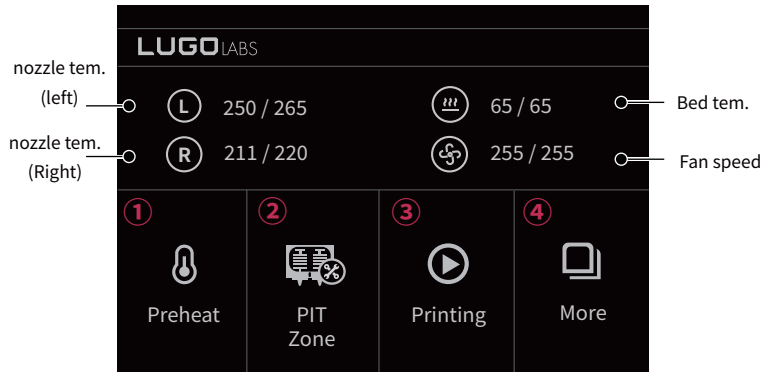


- 1 **Start simulation:** This starts the simulation of the currently active layer to provide visual feedback on the printing process.
- 2 **Path slider:** This indicates the current position of the simulation view. This handle can be dragged or moved with the 'arrow keys' or 'shift + arrow keys'.
- 3 **The layer slider:** This can be dragged to drag a layer range up and down.
- 4 **Print information:** Hover the mouse over the icon to view detailed print time information per printed feature.
- 5 **Print over network:** This will send the print job directly to the printer over the network. Non-networked printers have options to save to removable disk or save to file instead.

# UI menu structure

## Home

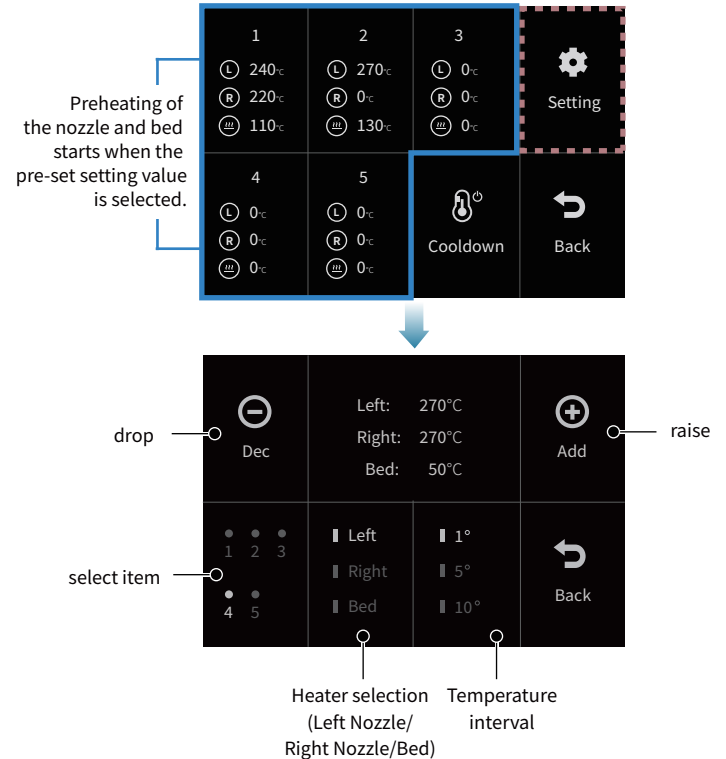
On the Home screen, there are items for basic operations frequently used by users.



- ① **Preheat** : Used to preheat before printing.
- ② **PIT Zone** : The head automatically moves to the center when nozzle maintenance or material replacement is required.
- ③ **Printing** : Used to print files in USB.
- ④ **More** : Used to set the necessary settings for output such as temperature, fan, and leveling.

## Preheat > Setting

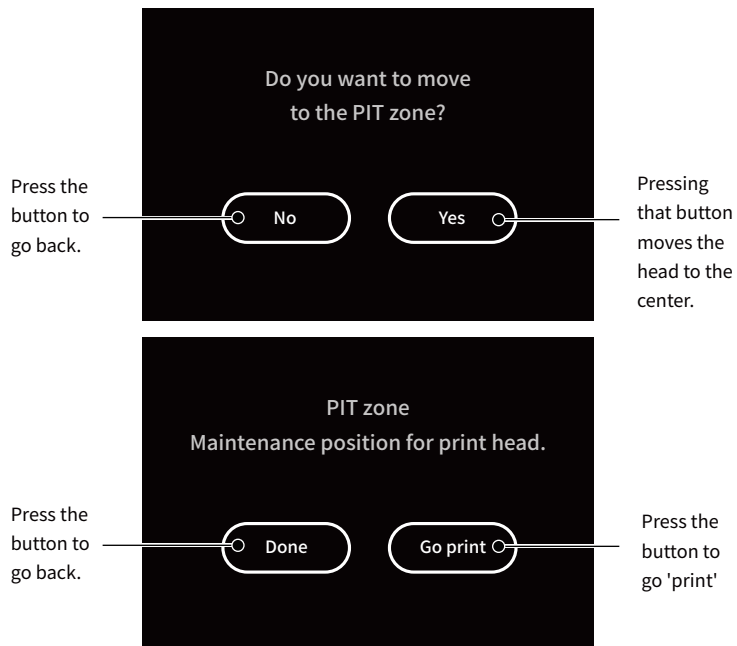
Used to preheat the nozzle and bed before printing. Frequently used set temperature can be saved and used as 5 items. To prevent unnecessary power consumption, nozzle preheating starts after the bed temperature has been preheated to some extent.



## UI menu structure

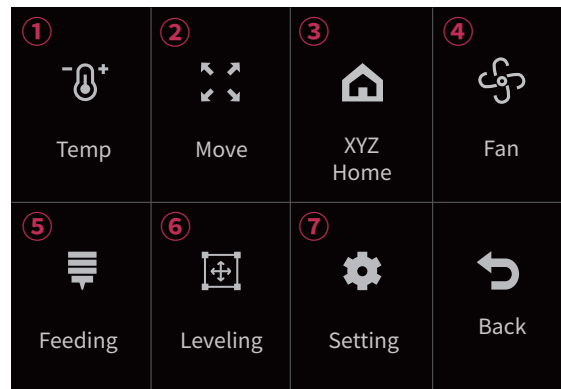
### PIT Zone

You can perform tasks such as replacing a nozzle or replacing a material.



### More

It is necessary for user-specific settings such as printer temperature, fan heater, and leveling.

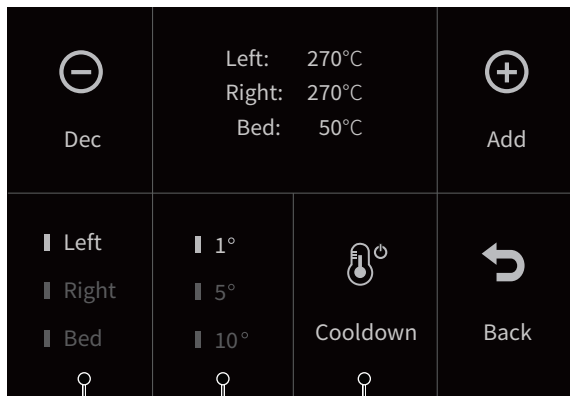


- ① **Temp** : Used to adjust bed and nozzle temperature.
- ② **Move** : Used to move the head.
- ③ **XYZ Home** : The extruder moves to the home position.
- ④ **Fan** : Used to adjust the fan speed.
- ⑤ **Feeding** : Used for extrusion testing.
- ⑥ **Leveling** : Used to set leveling.
- ⑦ **Setting** : Used to set other environment settings.

## UI menu structure

### More > Temp

There are items for operations used when adjusting nozzle and bed temperatures.



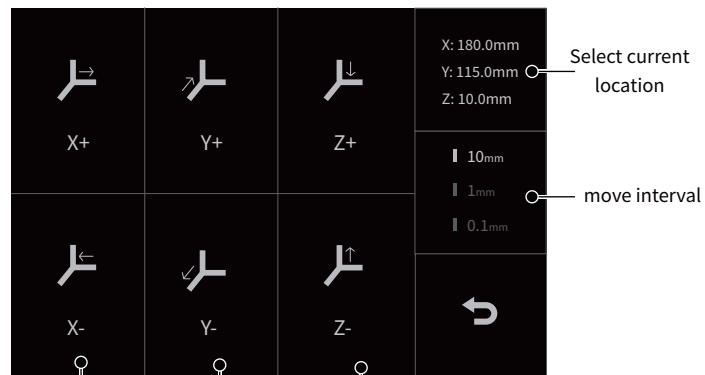
Heater selection  
(Left Nozzle/  
Right Nozzle/  
Bed)

Temperature  
interval

Turn off the  
corresponding heater

### More > Move

There is an entry for the operation used when automatically moving the head.



X-axis movement  
(move left / right)

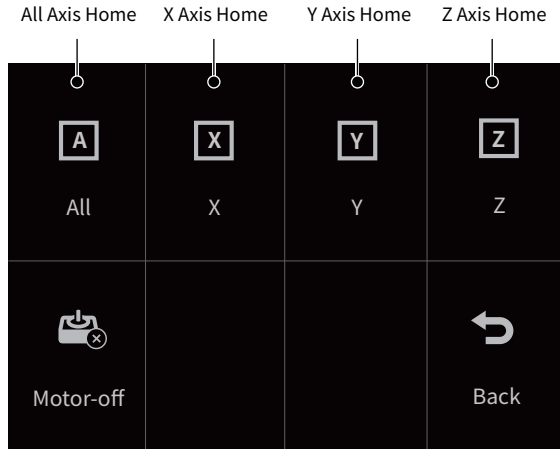
Yaxis movement  
(move forward/backward)

Z-axis movement  
(move down/up)

## UI menu structure

### More > XYZ Home

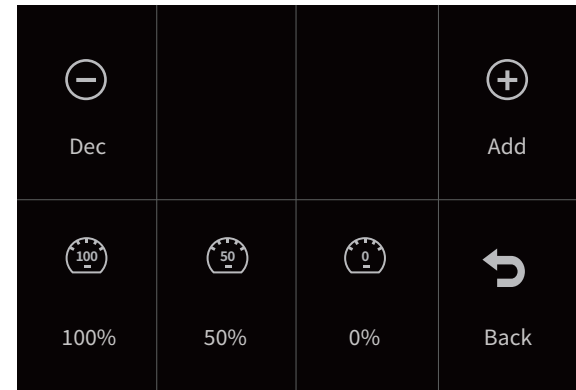
It is used to move the head to the home position.



ⓘ After each axis moves, electricity is being supplied to the motor so you can't move it by hand. When the motor is turned off, the electricity entering the motor is cut off, so you can move all axes by hand.

### More > Fan

There is an item about fan speed control.

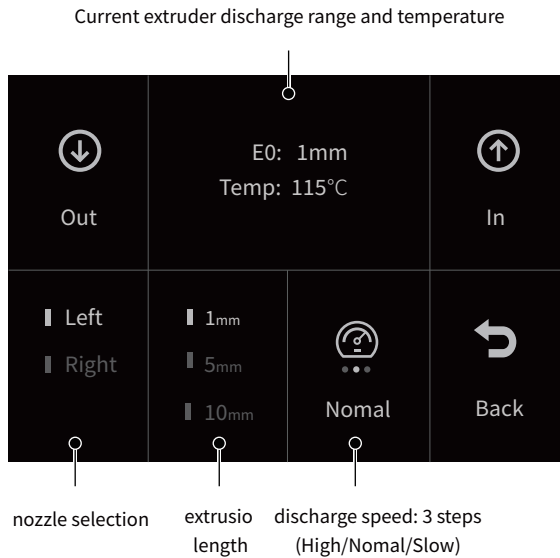


ⓘ The fan speed can be adjusted from 20% to 100%. Below 20% the fan will not run.

## UI menu structure

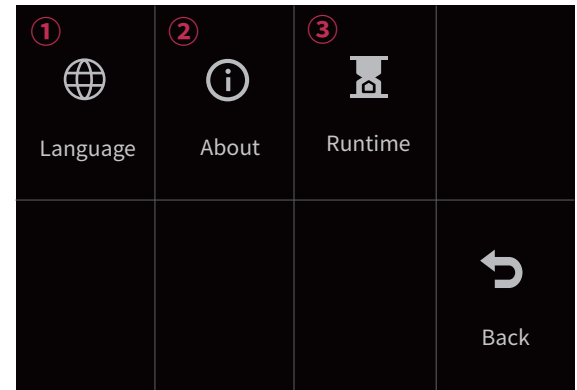
### More > Feeding

There is an entry about extruding or extruding materials.



### More > Setting

There are topics about customizing the printer.



- ① **Language** : Language setting
- ② **About** : Firmware version
- ③ **Runtime** : Runtime overview

## Preparing a print

### Leveling

Leveling is a process that is essential for printing by leveling the bed so that the gap between the nozzle through which the filament is extruded and the bed remains constant no matter where it is located.

- 1 Plastic residue attached to the end of the nozzle before leveling remove it. When removing residue, it is recommended to wipe with nippers or a wire brush after raising the temperature.

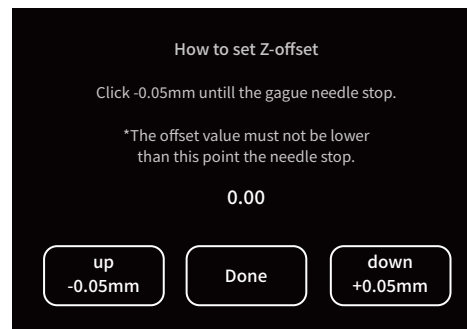
[How to raise the temperature: Home > More > Temp > add]



- 1 The leveling of the device is set before shipment, but the leveling may be out of order during delivery, so please make sure to perform leveling before proceeding with printing.



- 2 Select [More > Leveling > Z-offset > OK] on the Home screen, adjust the Z-offset value to 0.00, and press the [done] button.

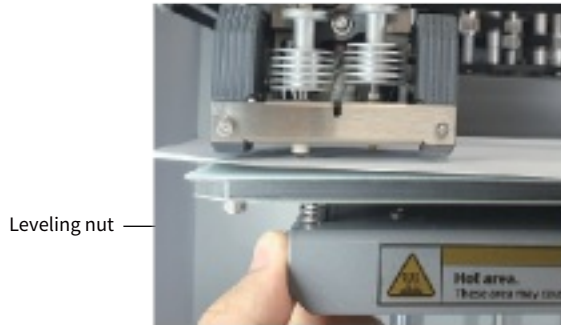


- 3 Press the [point 1] button to move the head to position 1.



## Preparing a print

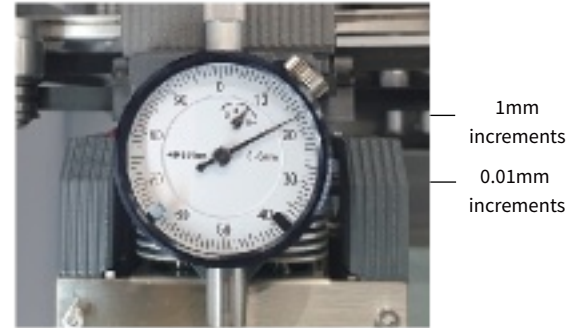
- 4 Insert plain paper between the print plate and the nozzle tip and adjust the leveling nut leaving a gap of about 0.5mm between the nozzle and the A4 paper when viewed with the naked eye.



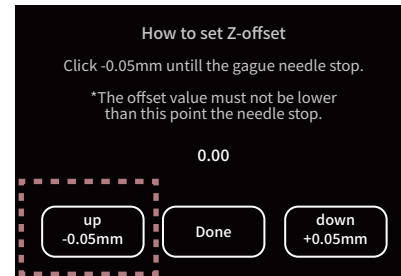
- 5 Install the gauge leveling by pushing it by hand.



- 6 Read the scale of the leveling gauge and make sure that the scale has the same value at the remaining positions 2, 3, and 4. adjust it. Repeat this process at least twice. If you have only done it once, leveling will be It may not fit.



- 7 After executing Z-offset at position 1, press the up/down button to adjust the interval. When you feel the A4 paper being squeezed into the nozzle, the setting is good.



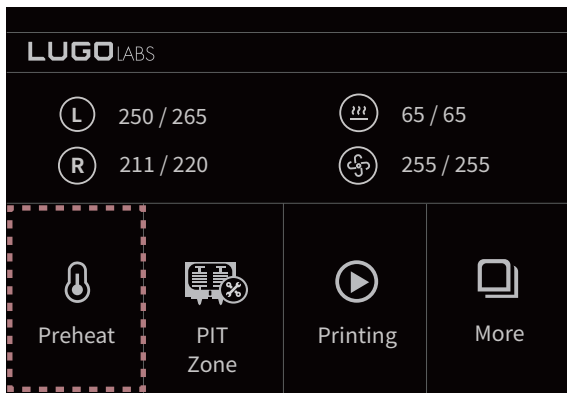
\* Z-offset can be adjusted even during printing.



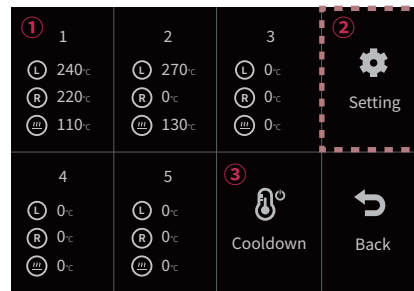
# Preparing a print

## Preheat

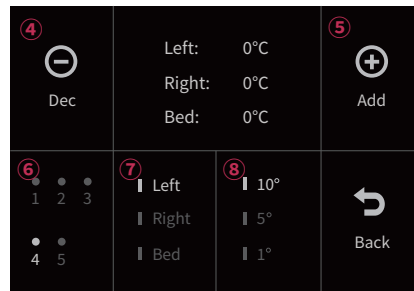
- 1 Press the [Preheat] button on the Home screen.



- 2 You can set frequently used nozzle and bed temperatures. You can set it in Setting, and if you press the frequently used setting value number, the temperature is automatically adjusted. To prevent unnecessary power consumption, nozzle preheating starts after the bed temperature has been preheated to some extent.



- 1 Frequently used temperature settings 1-5
- 2 Change frequently used temperature settings
- 3 turn off all heaters
- 4 Reduced nozzle and bed temperatures

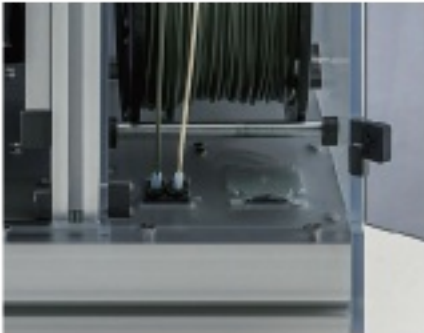


- 5 Increased nozzle and bed temperatures
- 6 Select temperature value number
- 7 Left/right nozzle and bed selection
- 8 Temperature increase/decrease range setting

## Preparing a print

### Checking Feeding

- 1 Adjust the temperature inside the drybox according to the filament you want to use. (Refer to page 38)



drybox  
temperature  
control dial

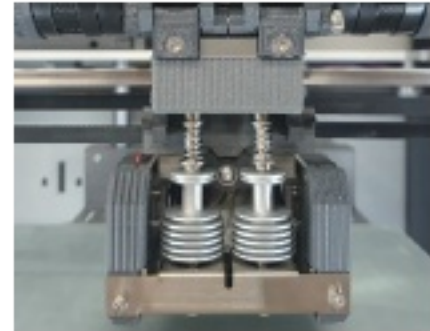
- 2 Raise the lever first and then remove the tube installed on the head. When separating, be sure to press the fitting clip with one hand and pull out the tube with the other hand.



lever  
fitting clip



- 3 Place the dual switch bar in neutral and put the ingredients in.



- 4 After putting the material to the end of the tube, close the lever and check if the material comes out of the nozzle end with the feeding handle.



feeding  
handle

## Preparing a print

### Bed setting



#### **Epoxy-plate**

Most materials can be used by using with all-round print beds such as PLA, PETG, and ABS.



#### **Yellow-PA-plate**

PA or PA-based filament.



#### **Brown-PA-plate**

PA or PA-based filament.



#### **PP-plate**

For PP or PP based filaments. Generally used at temperatures below 40 degrees.



#### **PC-plate**

For polycarbonate filament. Adhesion becomes stronger at bed temperature of 130~140 degrees or more



#### **HT epoxy-plate**

It is the same as the epoxy bed, but it can be used at a bed temperature of 160 degrees and a nozzle temperature of 400 degrees. When using for high temperature, it is essential to use adhesive paste for high temperature.

# Preparing a print

## Bed setting







### Adhesive gel

It is good to use for epoxy bed for the purpose of bonding parts to the bed more strongly.

Basic configuration adhesive gel can be used for PLA, PETG, ABS, and ASA. It is also good to use adhesive glue for other materials on the epoxy bed.

## Nozzle setting

	Tube type	Full-metal type
AM	 AM-T <ul style="list-style-type: none"><li>• material with low heat resistance</li><li>• Flexible material</li><li>• Most materials can be printed</li><li>• Max temperature 275 degrees ex) PLA, PETG, TPU ...</li></ul>	 AM-F <ul style="list-style-type: none"><li>• Engineering material with high heat resistance</li><li>• Materials requiring high-temperature output</li><li>• Maximum temperature over 400 degrees ex) ABS, PA, PC, PEI, PEEK ...</li></ul>
EM	 EM-T <ul style="list-style-type: none"><li>• Reinforcing materials such as GF and CF</li><li>• material with low heat resistance</li><li>• Most materials can be printed ex) CF-PLA, CF-PETG ...</li></ul>	 EM-F <ul style="list-style-type: none"><li>• Reinforcing materials such as GF and CF</li><li>• Engineering material with high heat resistance</li><li>• Materials requiring high-temperature output</li><li>• Maximum temperature over 400 degrees ex) CF-PA, CF-PP, CF-PEEK ...</li></ul>

## Preparing a print

### Setting the filter fan

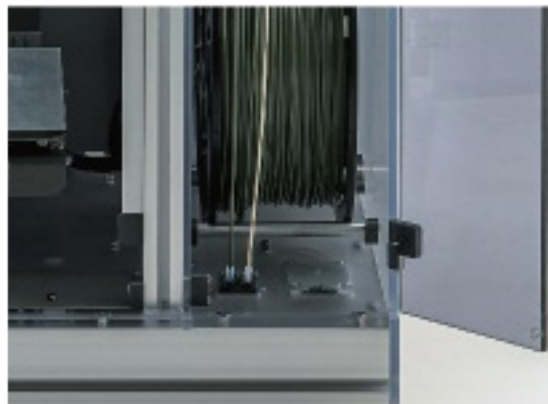
before printing You can select the strength of the fan for each material used. It can be adjusted in 3 modes, see the description below.



- **Red (weak):** For materials that require cooling due to severe shrinkage. (ABS, PC, PEI ... etc)
- **Green (medium):** For materials that require cooling but may shrink. (PETG, PVA ... etc)
- **Blue (strong):** For materials that require strong cooling. (PLA, TPU ... etc)

### Setting the temperature of the drybox

You can set the temperature for each material used before printing. Filament condition can be a part that has a great influence on the output result, so if you use the drybox well, you can get a satisfactory result.

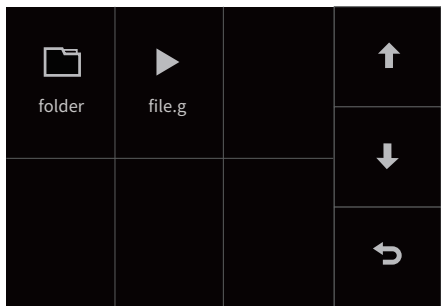
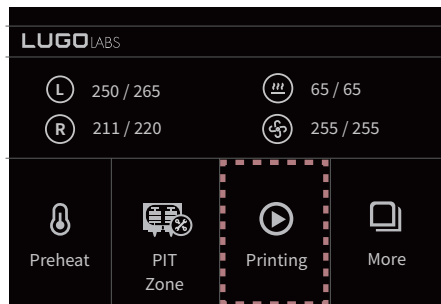


- **4~5 Level:** PLA, PLA based ... etc.
- **6~8 Level:** PETG, PBT, TPU ... etc.
- **9~10 Level:** PA, PC, PEI, PEEK ... etc.

! When drying by raising the temperature too excessively than the material to be used, the filament may melt or deform. Do not raise the temperature unreasonably for quick drying.

## Print

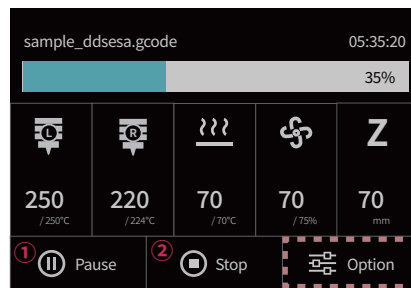
After inserting the USB containing the sliced file into the USB port, press the **[Printing]** button and select the file you want to print.



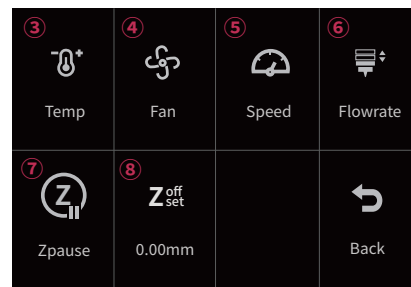
- ① When you press the button for the file you want to print, the temperature of the nozzle and bed automatically rises to the set temperature value in slicing.

## While printing

When printing is in progress, you can control various tasks on the screen during printing.



- ① Pause: resume is possible  
② Stop: resume impossible  
③ Temp: Nozzle and bed temperature control  
④ Fan: Fan speed control



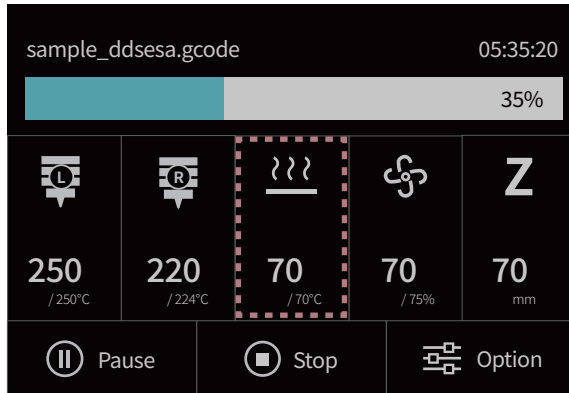
- ⑤ Speed: output speed control  
⑥ Flowrate: Adjust the amount of extrusion  
⑦ Zpause: Function to pause at a certain height (filament can be replaced)  
⑧ Z-offset: Z-axis movement during printing

## Separating printed materials

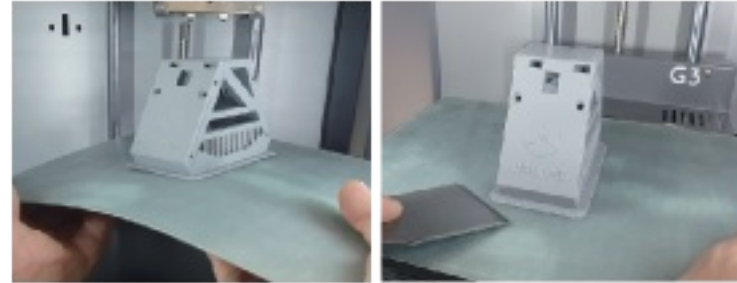


The scraper is sharp, so be careful when handling it.  
Be careful not to get injured by the scraper when removing the printout.

- 1 After printing is completed, wait for 15 minutes until the bed temperature drops below 35 degrees. Deformation of the print may occur if the print is removed forcibly while the bed temperature has not cooled down.



- 2 If you hold the bed on both sides and bend it slightly up and down, it will come off easily. Be careful when using a scraper or sharp object as it may damage the bed surface. When re-installing after removing the print, push it to the end of the bolt and install it properly.



[It is seated all the way to the inside of the bed]

- 1 If you bend it with too much force, the output plate may be bent and the flat end may be deformed, so do not apply excessive force. It does not.

## Maintenance

### Nozzle structure



#### Nozzle tip

area where melted filament is extruded after receiving enough heat.

#### Nozzle throat

passage through which filament is preheated and minimized heat conduction between nozzle tip and heat sink.

#### Heat sink

section where heat is released to prevent filament from melting

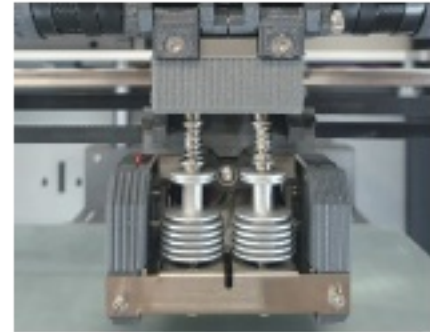
- ⓘ The rear fan of the fan mount serves to cool the heatsink, and the rear fan does not operate normally or if the internal temperature of the printer is too high, the filament in the heatsink section may melt and block the nozzle.

## Maintenance

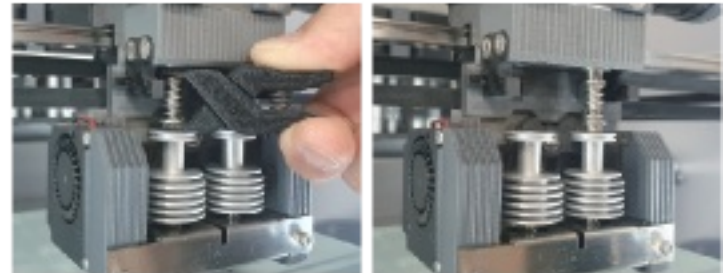


### Nozzle Management > Replacement

- 1 First, set the dual switch bar to neutral.

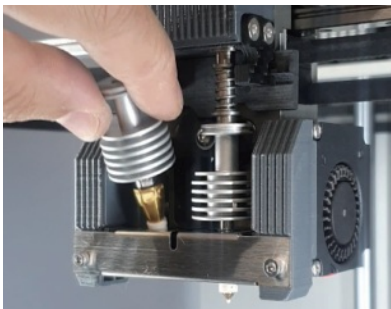


- 2 Use the included nozzle tweezers to connect the nozzle to the extruder, the spring. Take out the module.

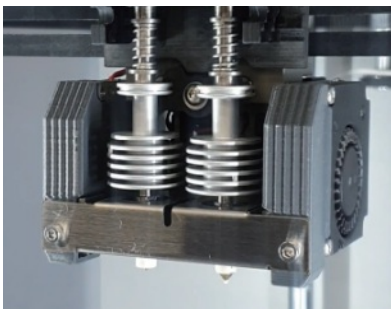




- Remove the nozzle by moving the dual switch bar and insert a new nozzle into the empty seat. (※ Set the direction of the dual switch bar to the right when replacing No. 1 nozzle and to the left when replacing No. 2 nozzle)



- Put the dual switch bar back to neutral and insert the spring module into the groove. (You will hear a click when it is in the correct position.)



In the process of removing the nozzle, proceed after the heat of the nozzle block is completely cooled.

If the filament has been fed before replacing the nozzle, proceed with preheating and removing the filament.

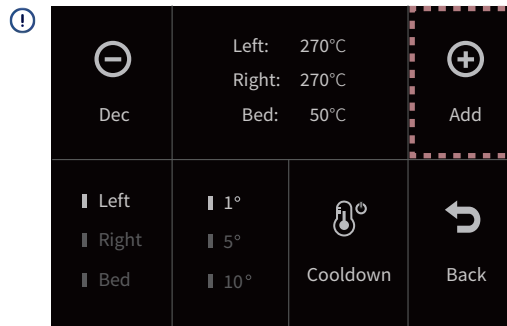
## Maintenance



### Nozzle management > clogged-1

This is a solution when the extrusion volume is insufficient due to foreign substances on the filament entering the nozzle or by-products accumulated as the old filament melts.

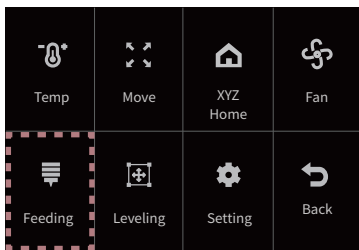
- Raise the temperature of the nozzle to pierce the nozzle.



Preheat the nozzle about 20 degrees higher than the appropriate temperature of the existing material so that the filament melts well.

Set the nozzle switching bar to neutral so that heat can be transferred to the nozzle.

- Push the ingredients in while rotating the feeding handle downward. (Push it continuously for about 30 seconds to remove foreign substances inside the nozzle.)



[Home > More > Feeding]

ⓘ If the melting point of the material being used is 250 degrees and the melting point of the new material to be replaced is 210 degrees, if the remaining material remains, it will not melt at 210 degrees, which can cause nozzle clogging. Therefore, in the state of preheating to 270 degrees, push in a new material to remove all the residues of the old material, then set it to 210 degrees again and use it.

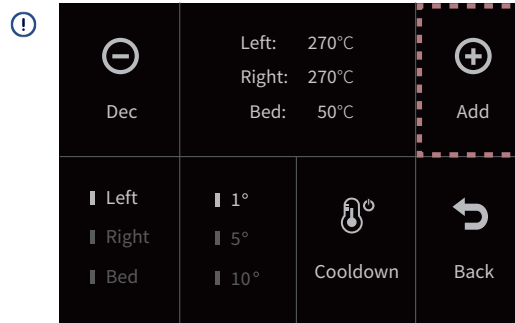
## Maintenance



### Nozzle management > clogged-2

This is a method to pierce the nozzle when the filament hardens near the nozzle neck ~ heat sink and cannot go in anymore, or when the amount of extrusion is insufficient even though the filament is pushed in sufficiently.

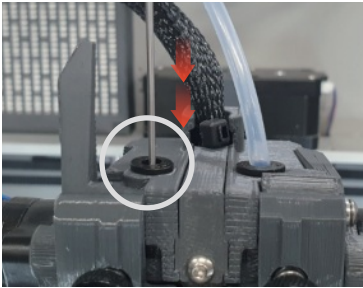
- Raise the temperature of the nozzle to pierce the nozzle.



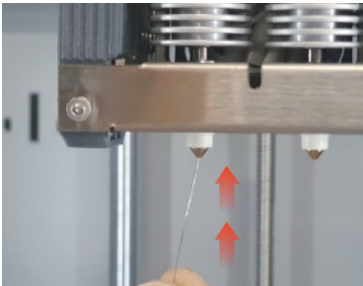
ⓘ Preheat the nozzle about 20 degrees higher than the appropriate temperature of the existing material so that the filament melts well.

ⓘ Set the nozzle switching bar to neutral so that heat can be transferred to the nozzle.

- With the material removed, raise the extruder lever and push the push pin to push the filament, or pierce the nozzle tip using the enclosed  $\Phi 0.38$  needle. The effect is better if you push the filament at the same time.



[push pin]



[ $\Phi 0.38$  needle]

!  $\Phi 0.38$  needle, move it up and down to clean the inside of the nozzle.

## Maintenance

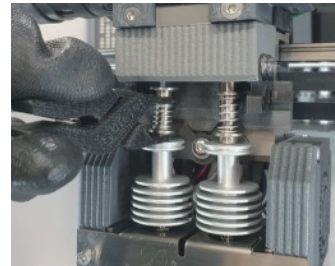


### Nozzle management > clogged-3

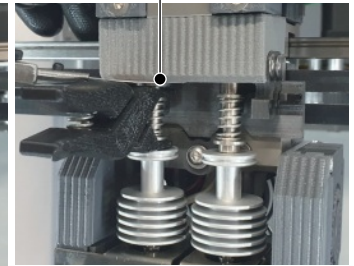
When the filament is clogged inside the heat sink ~ extruder, this is how to pierce the nozzle when it is not pushed with a push pin.

- Use the nozzle tongs to hold the nozzle spring and cut the filament with a box cutter.

! box cutter



[Grab the spring with the nozzle tongs]



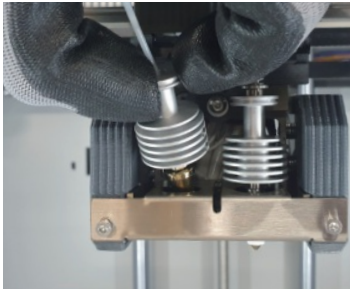
[Cut the filament with the cutter knife using the gap]

! When using a cutter knife, be sure to wear gloves and cut it by pushing it outward from the body.

- 2 Twist the nozzle tongs sideways to remove the spring, then remove the nozzle.



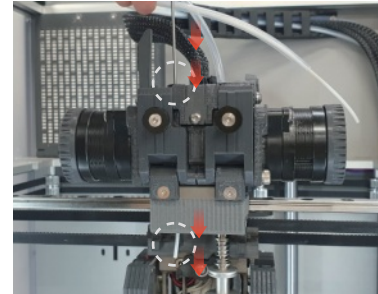
[Remove the spring]



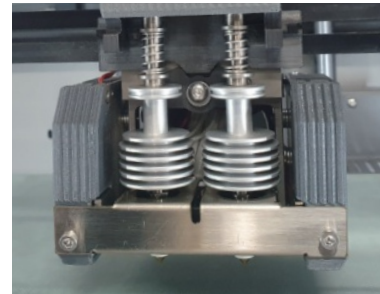
[Remove the nozzle]



- 3 Open the extruder lever, push out the filament inside with a push pin, and then install the nozzle.



[Remove filament]



[Install nozzle]

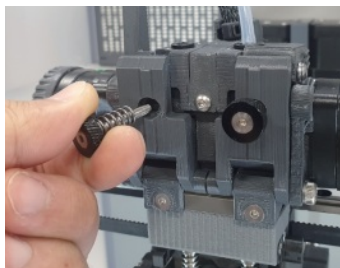
## Maintenance

### Extruder management

#### > The problem that the extruder cannot push the filament

usually, when the nozzle is clogged, the extruder also has problems. Because the filament cannot be pushed out, the filament is damaged by the extrusion gear, and the filament residue is caught inside the extruder or between the extrusion gears, so the filament cannot be pushed normally. Therefore, even if the nozzle is replaced, it may not extrude properly.

- 1 Remove the tension spring of the extruder that does not extrude properly, open the extrusion gear lever, and then clean the extrusion gear with a wire brush or toothbrush.



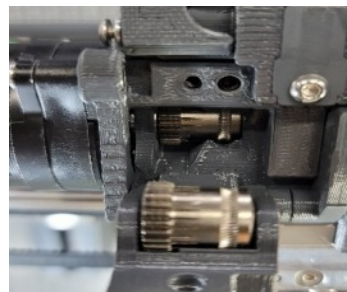
[Remove the tension spring]



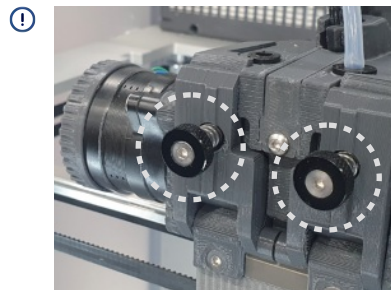
[Clean the extrusion gear]



- 2 Blow air to remove foreign substances inside the extruder, and then fix it again with a tension spring.



[Clean the inside of the extruder]



[Fix the tension spring]

- ! After the thread is engaged, the tension spring turns about 4 turns (the distance between the handle and the output is 8.5mm).

## Maintenance

### Filter management

Replace the filter once every 6 months to 1 year depending on the frequency of use of the printer.

- 1 Unscrew the 2 M3 pan head bolts that secure the filter cover.



- 2 Take out the used filter and check the direction of the new filter. Put it in.



- 3 Close the cover again and fix it with bolts.



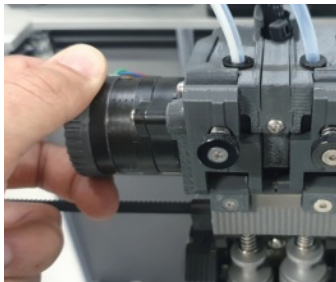
## Maintenance

### Self-diagnosis of defective extrusion

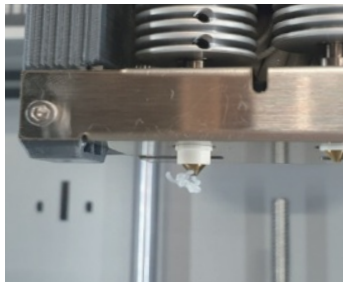
Five things to check when extrusion does not work properly.

#### 1 Check if the nozzle is clogged.

Push the filament into the preheated nozzle to see if it extrudes properly. When the feeding handle is turned slowly, if extrusion does not work properly and the handle is pushed up again, the nozzle is clogged. If the filament being extruded from the tip of the nozzle does not come down all the way down and curls up around the nozzle, it is necessary to drill the nozzle.



[Turning the feeding handle]



[Material twisting and extruding]

#### 2 Check the condition of the extruder.

Remove debris inside the extruder and clean the extrusion gear. Make sure the extruder lever is closed properly. Check if the tension spring is overtightened or too loose.



[check extrusion gear]



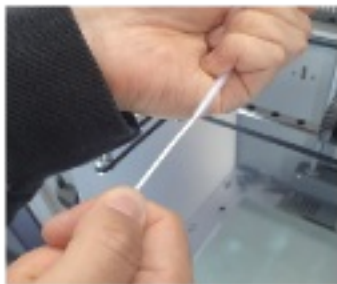
[Checking the tension spring]

### 3 Check the condition of the tube.

If the tube is severely damaged or bent in certain parts, the extruder cannot properly pull the filament due to excessive friction. If the material inserted from the end of the tube on the print head side does not come out stiff when pulled by hand, it may be the cause of poor extrusion.



[Checking for damage to the tube]



[Trying to pull the material from the end of the tube]

### 4 Check the dry box temperature and filament condition.

If the internal temperature of the dry box is higher than the temperature at which the filament dries, deformation such as melting or bending of the filament occurs and the extruder cannot properly pull the filament. (※Beware of temperature when printing PLA, PETG)



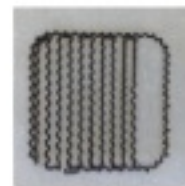
[Melt phenomenon when the temperature is high]

### 5 Check the leveling.

If the level is too tight, the material will not extrude, and in severe cases, the nozzle tip and bed may be damaged. Conversely, if the print is left unattended for a long time in a state where the print is not seated due to falling, the filament sticks to the nozzle and the nozzle block and prints in a lumpy state. In this case, a situation arises in which both the nozzle and the nozzle block (core module) need to be replaced.



[Adequate]



[High]



[Low]

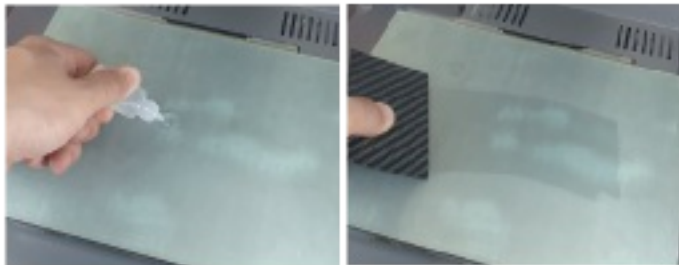


## Maintenance

### Bed seating problem > Output seating problem

Apply gel to the printing bed or check the level.

- 1 Sprinkle gel on the printing bed as much as the size of a 10 won coin, then spread it thinly as much as the printed area using a card.

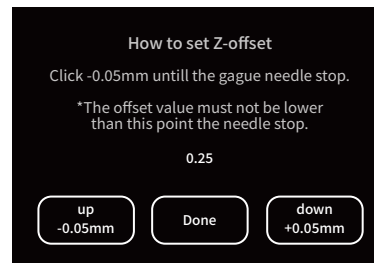
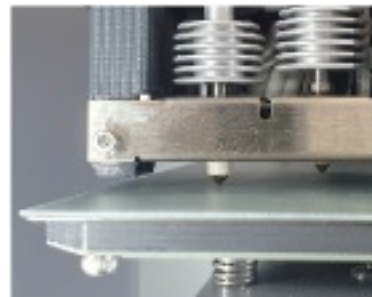


[Apply gel]

[Apply thinly]

- ⓘ If the gap between the nozzle and the bed is too far, the printed object may not be seated or it may come off the floor during printing. may fall.  
If the gap is too close, the printout may get caught in the nozzle and fall off the floor during the mount movement section, so you need to set an appropriate level.

- 2 Check the level to see if there is an appropriate gap between the nozzle and the bed.



[Home > More > Leveling > Z-offset]

- ⓘ During output, adjust the level by using the Z offset function instead of the leveling nut.