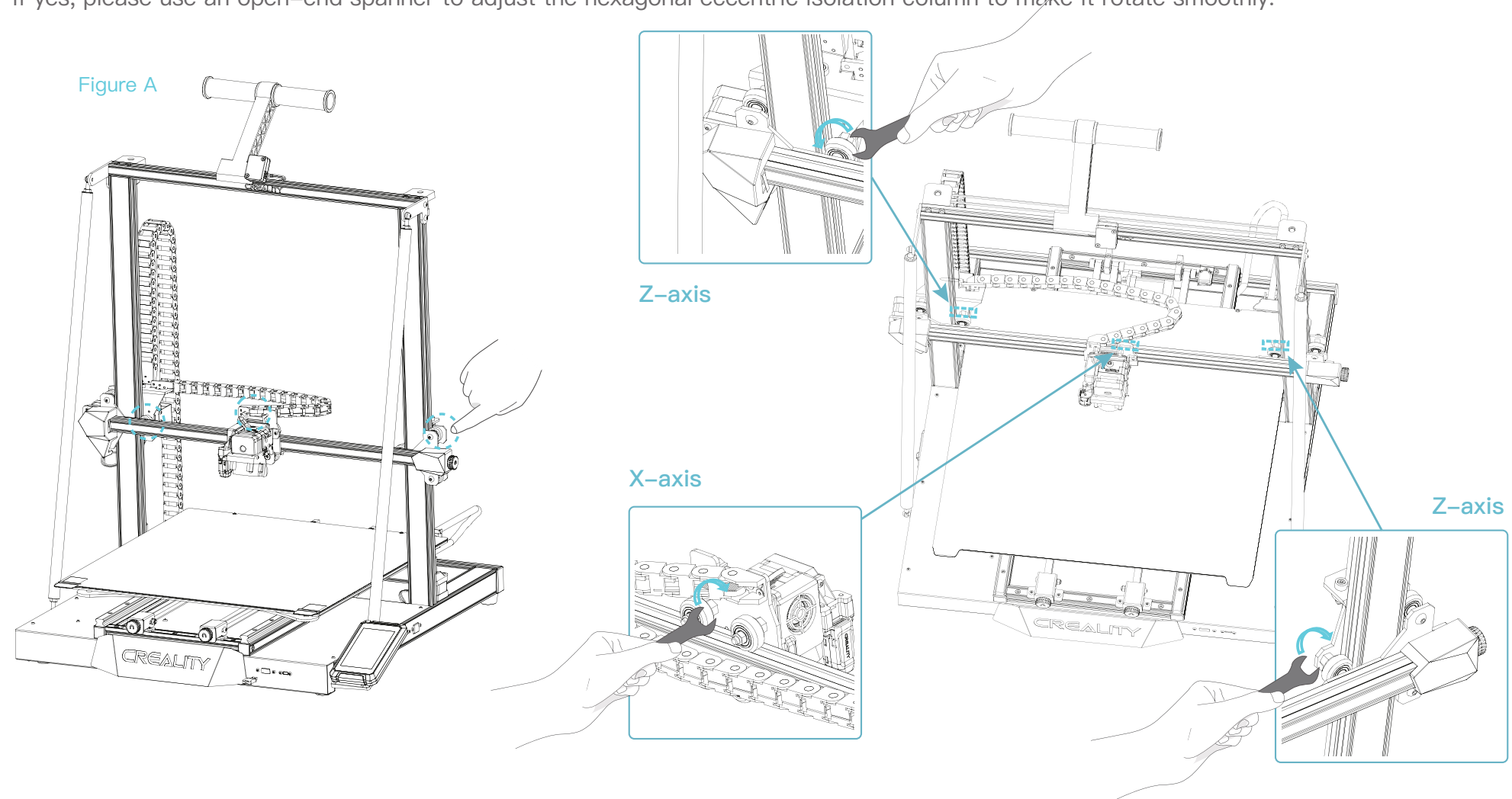


## 6.Adjusting Pulley Tightness

### X/Z-axis Pulley Adjustment

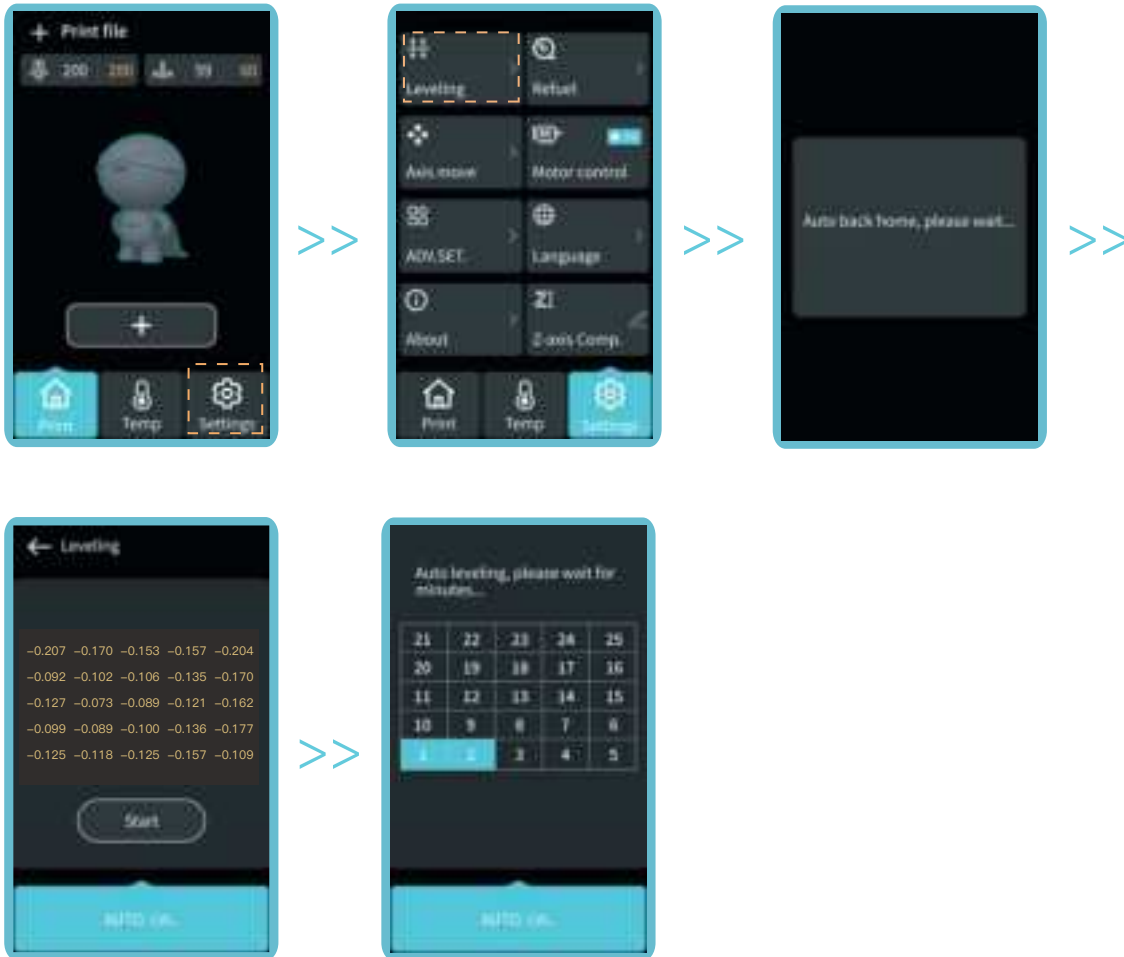
Before switching on the machine, please check the tightness of the pulley by gently toggling it with your hand and check if it is idling or stuck (as shown in Figure A).

If yes, please use an open-end spanner to adjust the hexagonal eccentric isolation column to make it rotate smoothly.



## 7.Auto Leveling

7.1 Click on "Settings → Leveling" in the main interface, and then click on "Start" after the machine has returned home, and wait for the machine to detect the platform to finish the auto leveling.



The above interface is for reference only.

## 7.Auto Leveling

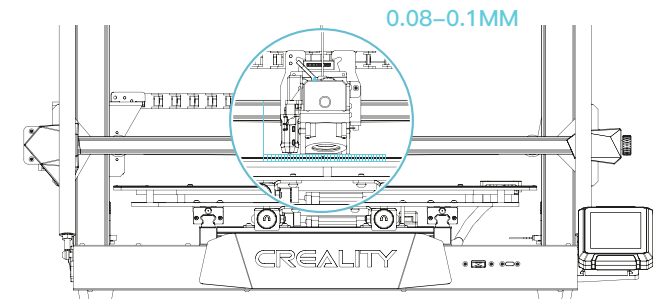
7.2 Click on "Move Axis" in the settings screen, observe the height value of the Z-axis, set the value to 0 and then click on "Move Axis" to return.

Click on "Z-axis Compensation" and observe the gap between the nozzle and the platform by adjusting the Z-axis compensation value so that the distance between the nozzle and the platform are about the thickness of a piece of an A4 paper.



7.3 Observe the print result after starting printing, and adjust the Z-axis compensation again if any exceptions occur.

⚠		
		❌ The nozzle is too far away from the platform, so the filaments can not adhere to the platform.
		✅ The nozzle is too close to the platform, leading to insufficient filament extrusion, even scraping the platform.
		⚠ Filaments are extruded evenly, just sticking on the platform.



The above interface is for reference only.

## 8. Filament Infill

### 8.1. Preheating Method

- A. Preheating mode 1: Click on "Temperature" to select the auto temperature setting and choose PLA/ABS according to the type of filaments.
- B. Preheating mode 2: Click on "Temperature" and select manual temperature setting, enter the corresponding values in the "Extruder Preheating" and "Hotbed Preheating" boxes.

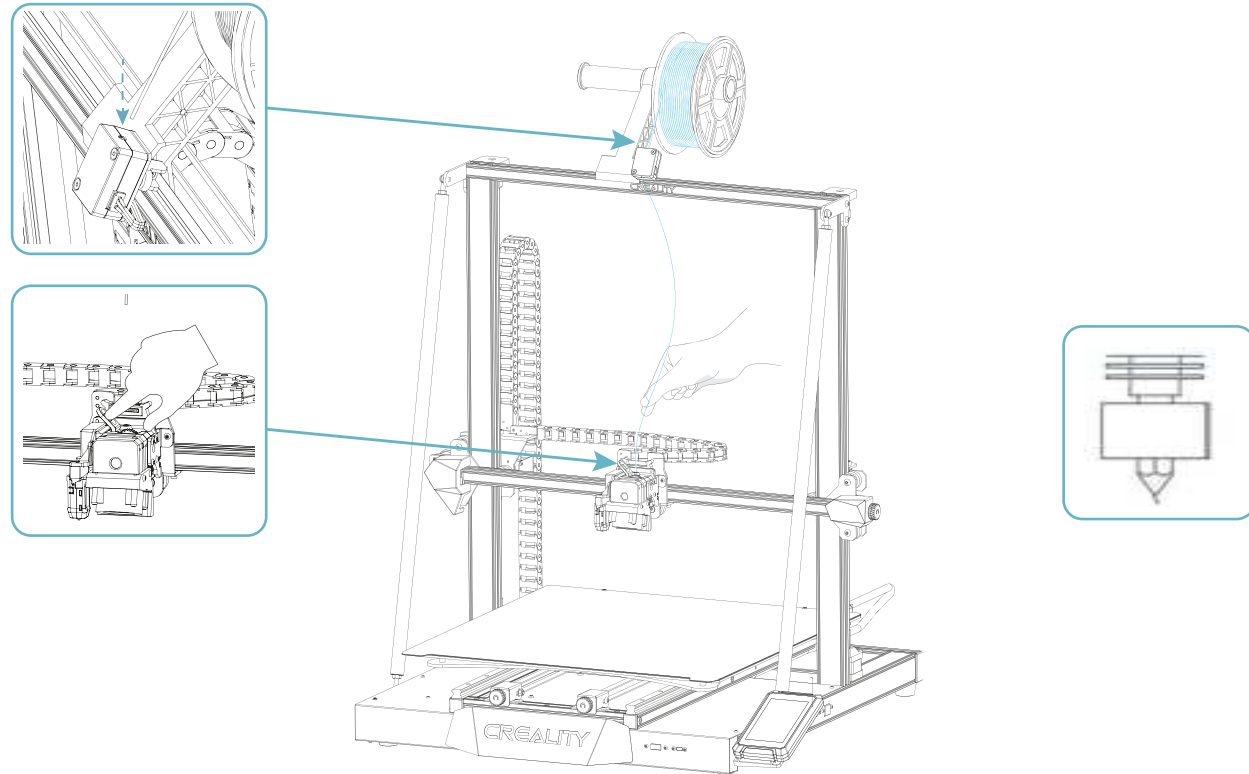
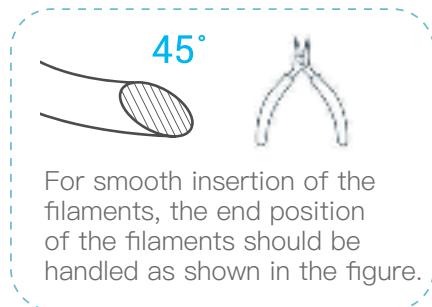


Note: The standard printing preheating parameters have been adjusted upon delivery. For PLA preheating: nozzle temperature 200°, hotbed temperature 60°; for ABS preheating: nozzle temperature 250°, and hot bed temperature 100°. If you need to make adjustments due to reason of material to be printed or other reasons, you can adjust it separately under the "Temperature → Manual setting" on the home screen.

## 8. Filament Infill

### 8.2 Filament Infill

- A. When you wait for the temperature to rise, hang the filament over the material rack.
- B. Pass through the Filament Sensor, then press the extruder handle and insert the filament along the extruder hole to the nozzle position. If the filament flows out of the nozzle when the temperature reaches the target value, the filament is properly loaded.



#### Replacement of filaments:

1. When the machine has just started printing, cut filaments near the extruder and slowly feed filaments until they are fed into the material barrel.
2. When printing has finished and the machine has cooled down, heat the nozzle to 185°C+, then draw the filaments out and replace them by referring to step 1.
3. To change the filaments during printing, adjust the printing speed to 10% and refer to step 2 for replacement.

## 9.Start Printing

### Installation and use of the Creality Print slicing software

Creality Print



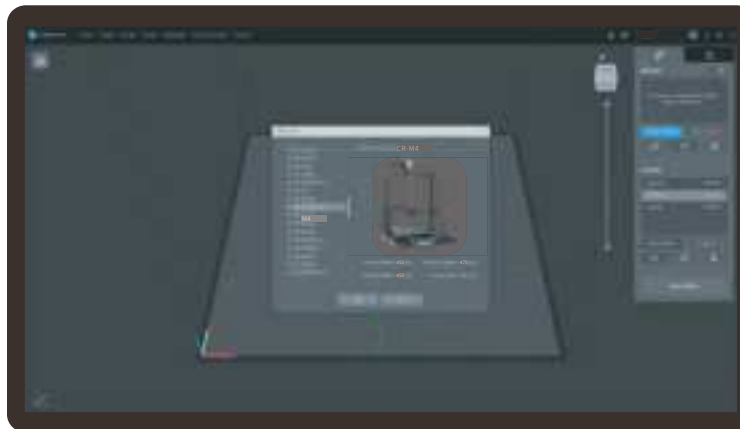
① Download at <https://www.crealitycloud.cn/software-firmware/software?type=7> or find the software on a USB flash drive and install it;



② Choose a language;



③ Choose a region;



④ Choose a model;



⑤ After the "Add Printer" is completed, adjust the basic parameters through the printer "Management": **the screen size should be 480\*800 and the preview image type should be jpg, otherwise the previews cannot be generated;**  
Note: Previews cannot be generated when using other slicing software;

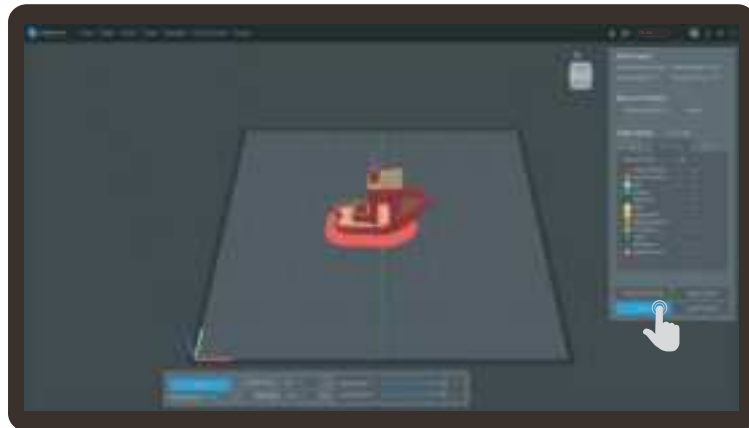
## 9.Start Printing



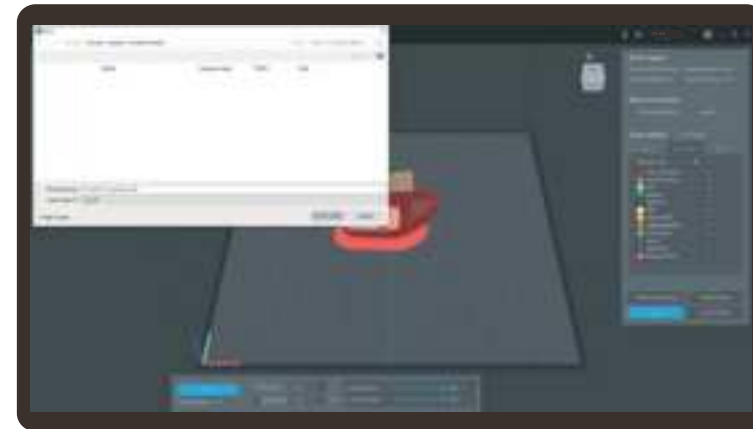
⑥ Click on "Import Model" and select the model file;



⑦ Open the model and click on "Start Slicing";



⑧ Click on "Export" when the slicing is finished and then save the finished file;



⑨ Generate a g-code file → Save to USB flash drive.



The above interface is for reference only.

## 10. Print Method

### 10.1 Local printing from a USB flash drive



- ① Generate a g-code file using Creality Print slicing software and save it to a USB flash drive.



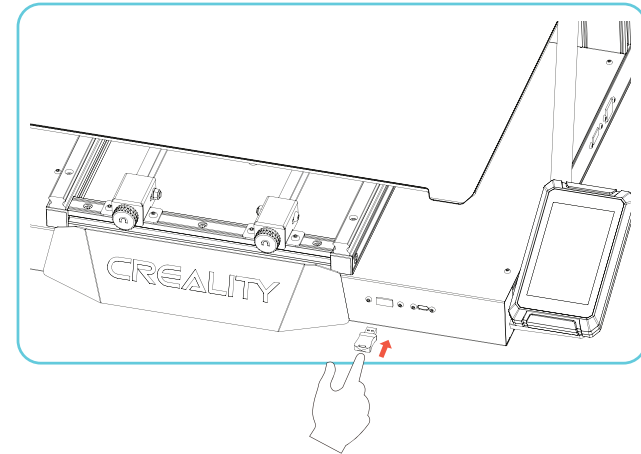
For details on using the software, please refer to the slicing software user manual on the storage card.



Do not plug or unplug the USB flash drive during the printing process.



Tips: The file name must only contain Latin letters and digits. Chinese characters or special symbols must not be used.



- ② Insert the USB flash drive and select the file you want to print.

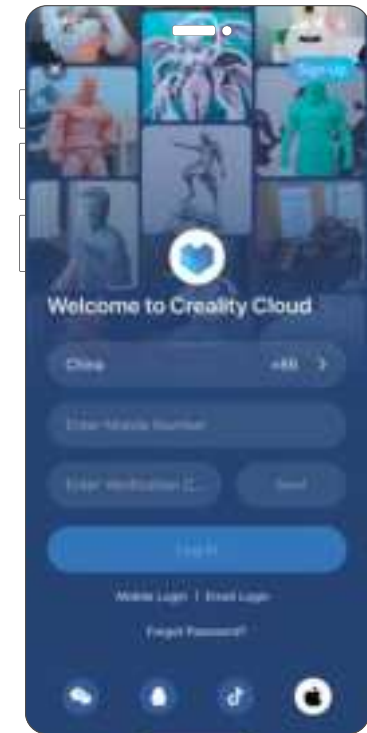
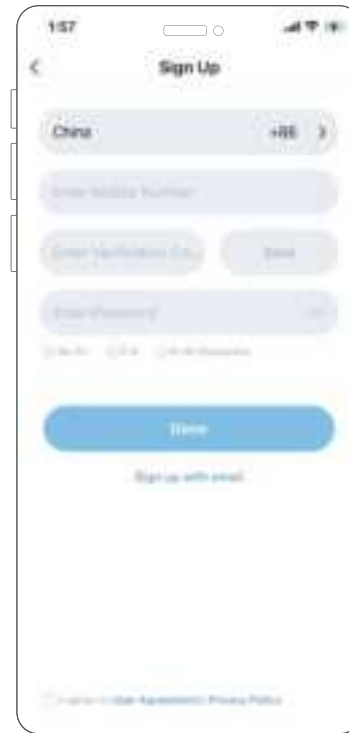
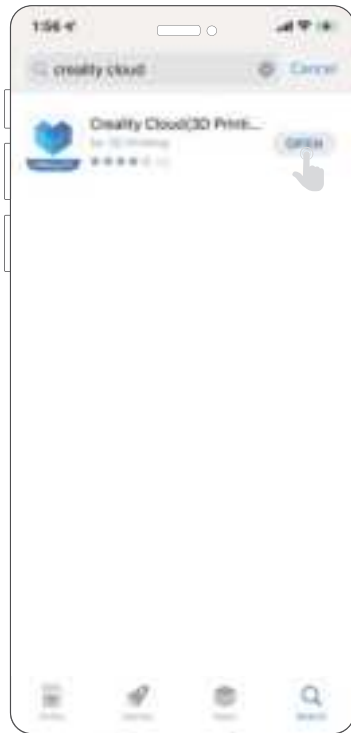




## 10. Print Method



### 10.2 Download and install Creality Cloud



① Search for "Creality Cloud" in the app shop, download and install it.

② Sign up for an account.

③ Log in.



The above interface is for reference only.