



Repetier-Host Operation Guide

Version : 1.0

Operation Process

3D Printing Process



*3D printer can only support gcode files, before placing the 3D model into 3d printer, need to convert the 3D model file (usually they are stl, obj, AMF files) to gcode files, this process is called slicing.

Repetier-host installation and setting process



Slicing and print process



* Donot need to connect 3D printer to PC if you donot print from PC

Note: This document only provides a brief description of Repetier-Host. For more details, please search on the web.

Install the driver (Windows 64-bit as an example)

1. install driver

Driver path: ..\driver\CP2102\windows\CP210x_Windows_Drivers\CP210xVCPInstaller_x64

Note: You may need to unzip the compressed file before installation.

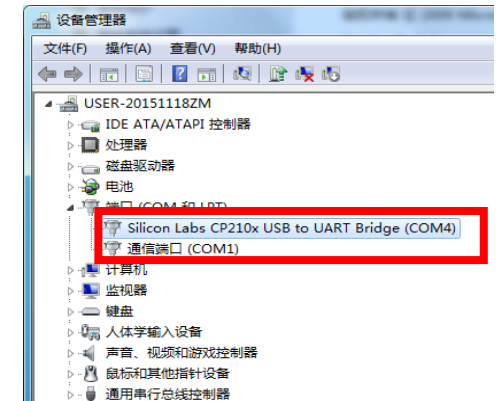


2. Connect the printer to the PC with a USB cable.



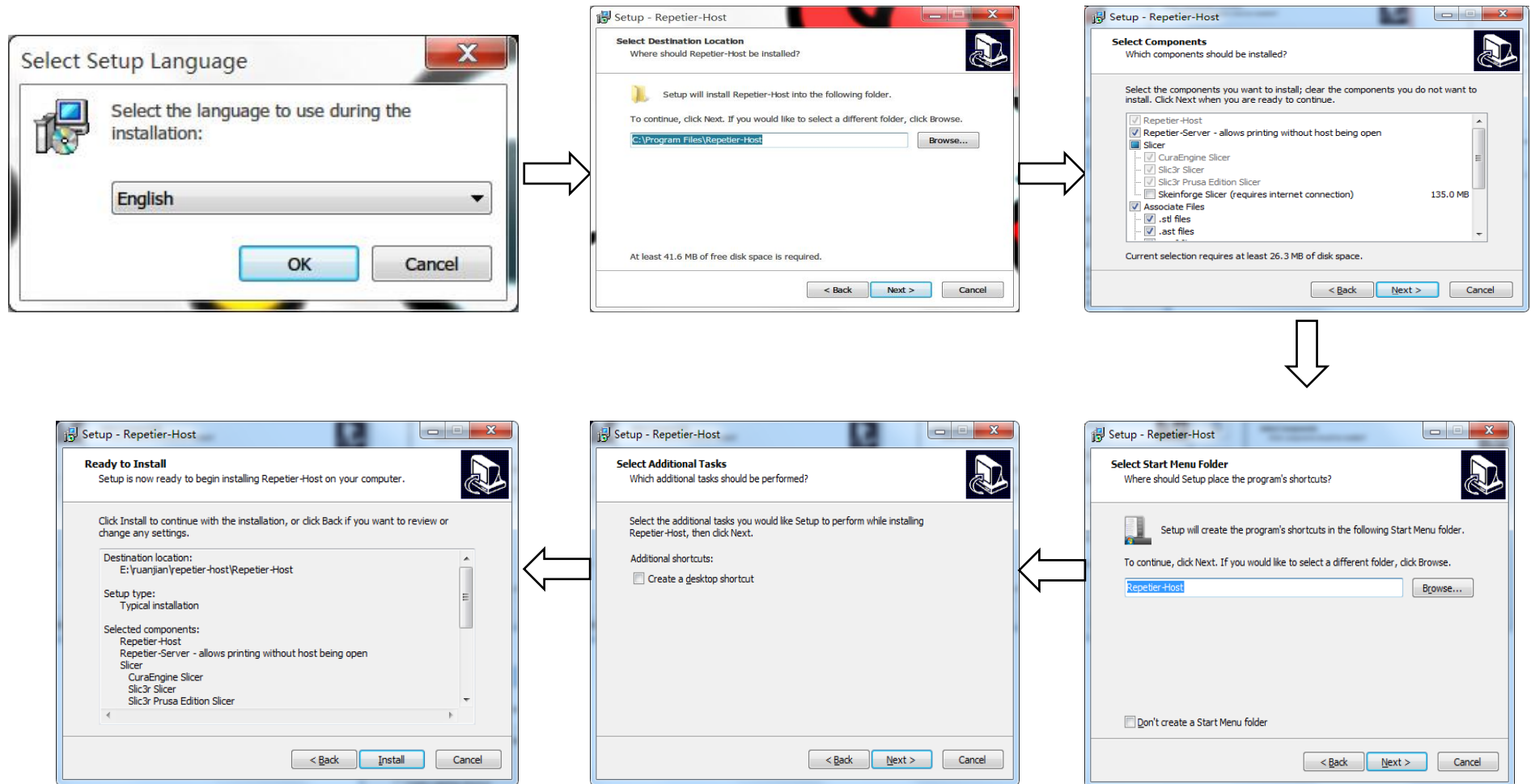
3. Find communication port

Right click on "Computer" >> "Properties" >> "Device Manager" >> "Port", expand to find the device you just installed, pay attention to the port number (example is COM4).

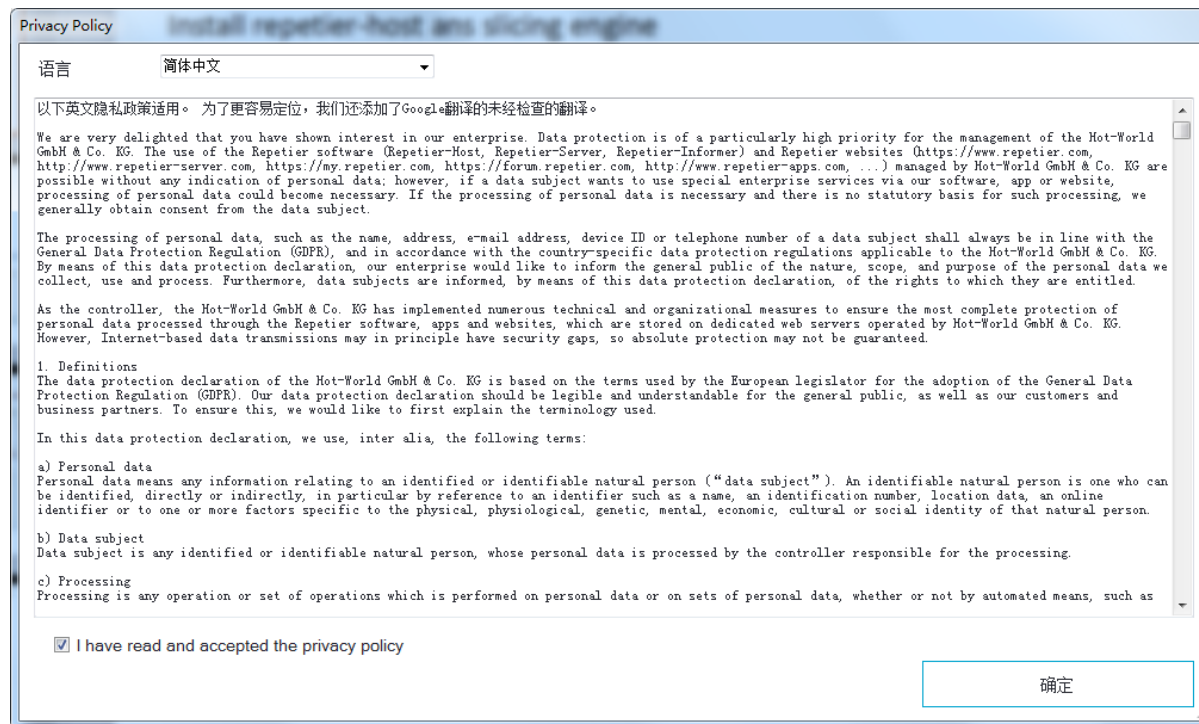
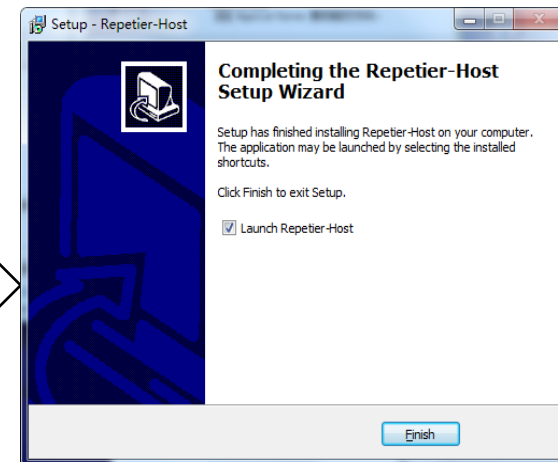
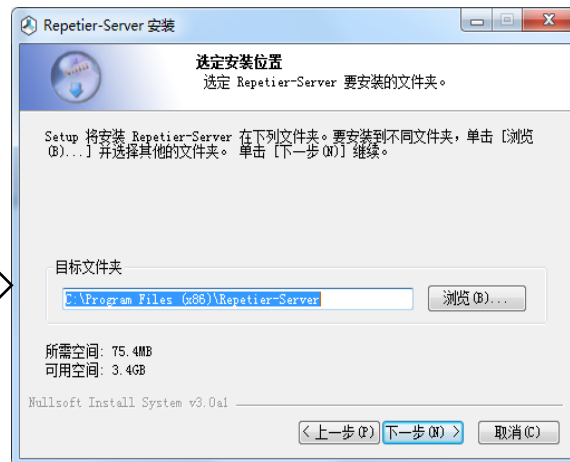
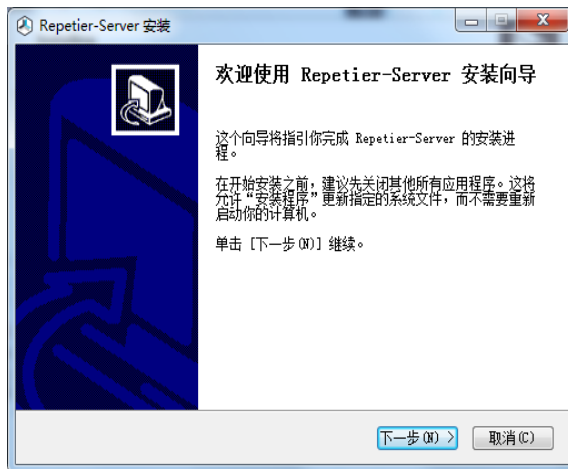


Install Repetier-Host

Install repetier-host and slicing engine

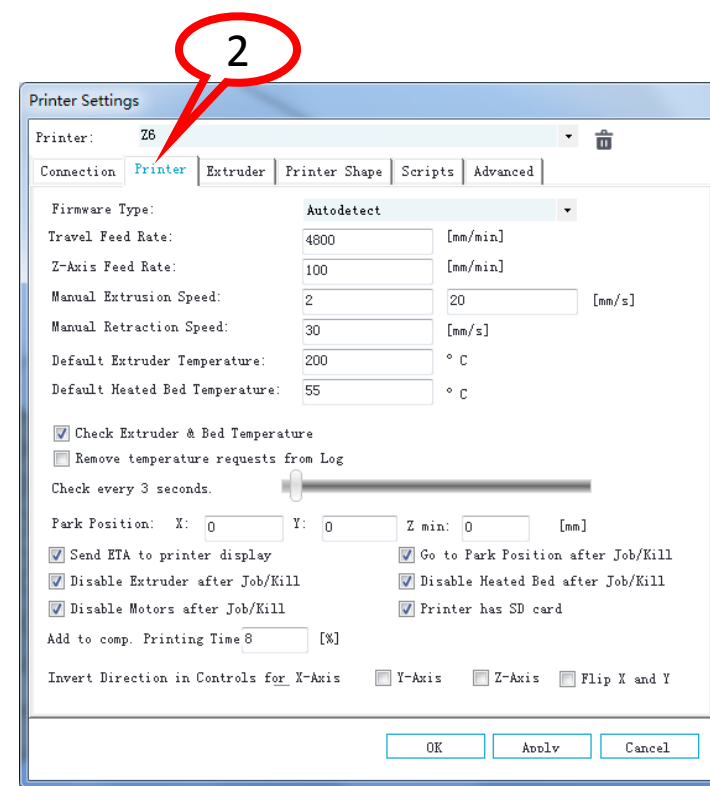
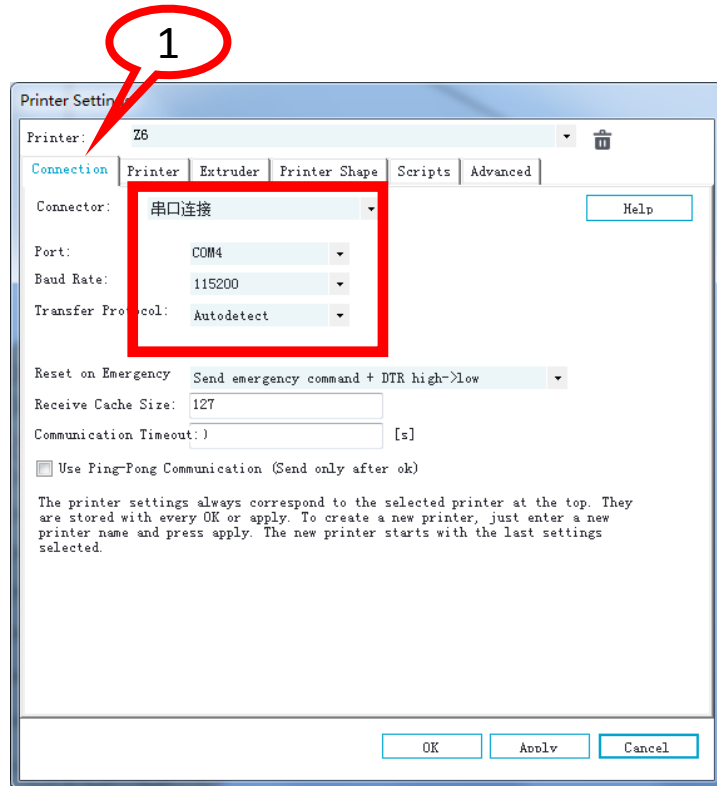


Install Repetier-Host



Set Parameters

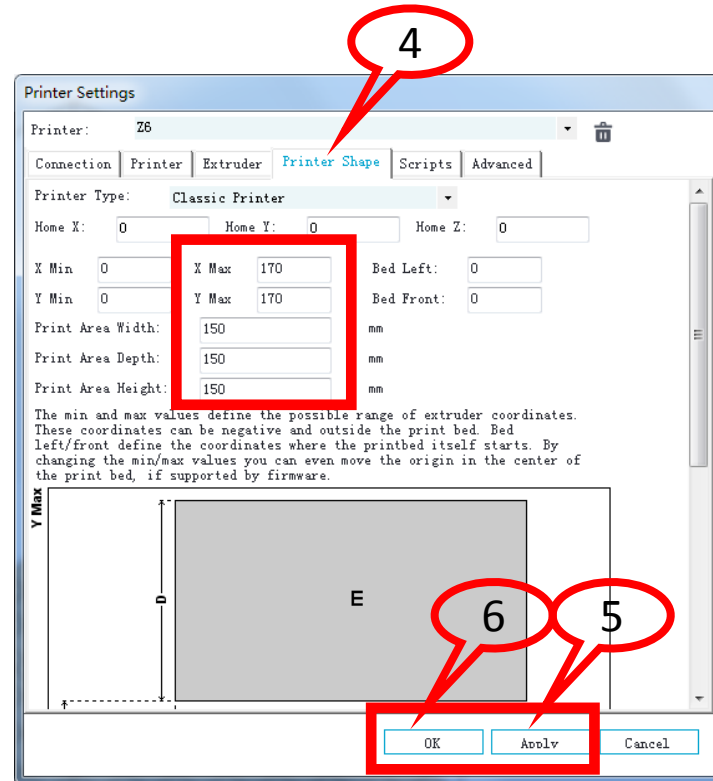
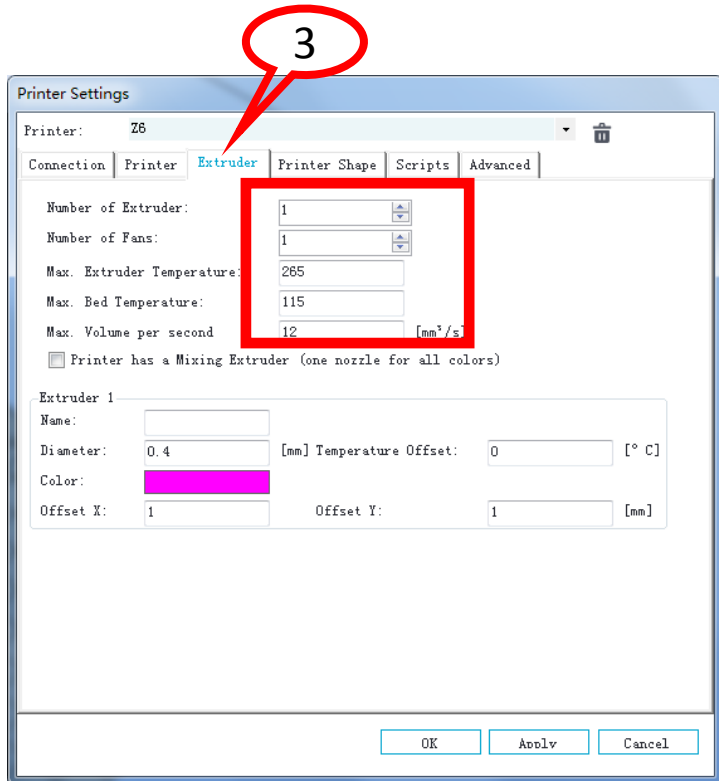
Open Repetier-Host and click on "Printer Settings" in the upper right corner to set the printer parameter as below.



Note: The above settings are all examples of Z6. For the actual setting parameters, please refer to your machine, usually we store the parameter in a directory named "Slicing Setting", please find it in the SD card.

Set Parameters

According to your printer to set **Extruder** and **Printer shape**, and click "Apply", "OK " to save the settings.



Manual Control Menu


Object Placement

Slicer

Print Preview

Manual Control


SD Card


 Heating Bed


G-Code:


Send


X: 0.00 Y: 0.00 Z: 15.00


 X


 Y


 Z



















































































Feedrate

100

Fan

100

Bed Temperature - 42.27°C

50

Extruder 1 - 24.59°C

210

Send gcode to printer

Current position of axis

Move axis

Print Speed

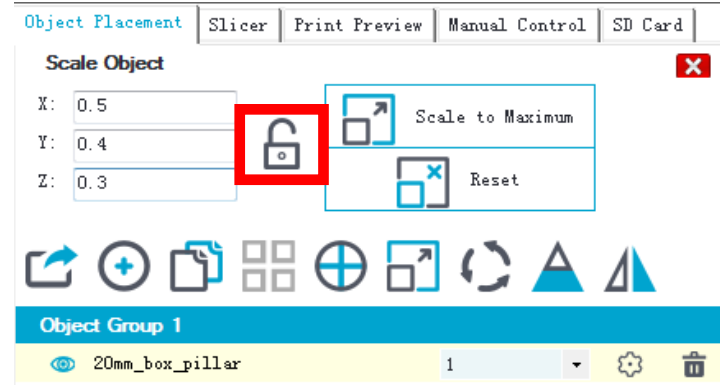
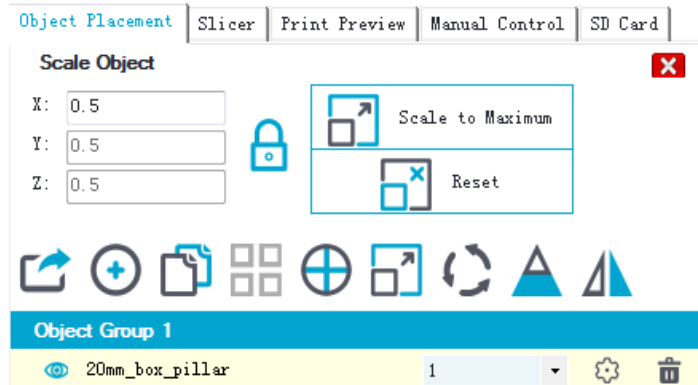
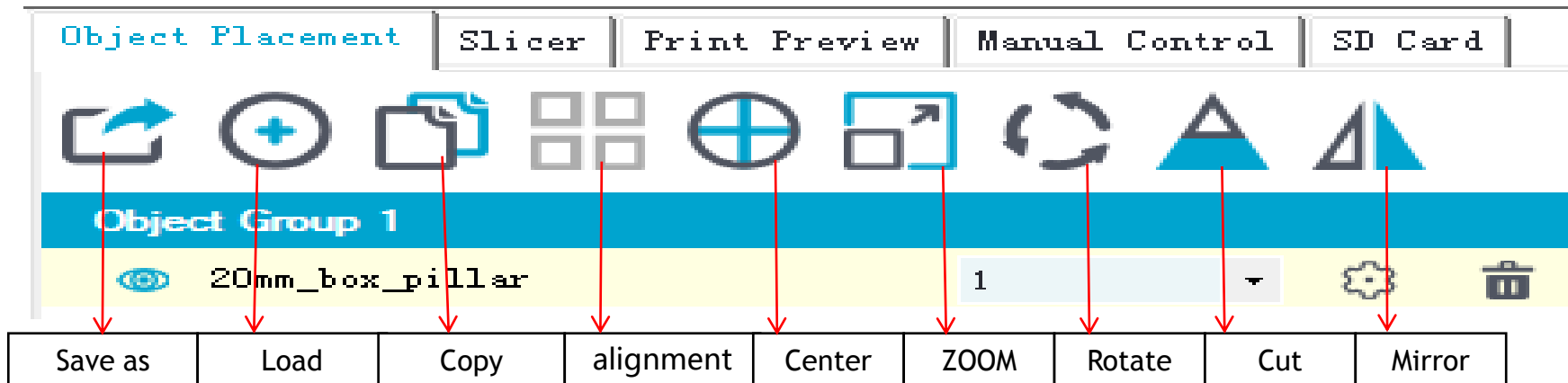
Fan Speed

Hotbed Temperature

Nozzle

Load and align 3D model

1. Load 3D model files (stl, Obj, AMF, etc.).
2. Click on alignment icon to make the 3D model in the middle of the hot bed.
3. Click zoom icon to adjust the size of the 3D model. Click the “lock” icon in the figure to set the size of the model, X, Y and Z axes.



切片参数设置

Select "Slice Software" and click "Configure" to set the slice parameter.

The image shows the Cura software interface with the following settings and navigation steps:

- Step 1:** Click the **Configuration** button in the Slicer menu.
- Step 2:** Click the **Speed and Quality** tab.
- Step 3:** Click the **Extrusion** tab.
- Step 4:** Click the **General Extruder Settings** section.

Speed and Quality Tab Settings:

	Slow	Fast	
Print:	40	60	[mm/s]
Travel:	150	150	[mm/s]
First Layer:	30	30	[mm/s]
Outer Perimeter:	30	60	[mm/s]
Inner Perimeter:	40	80	[mm/s]
Infill:	60	100	[mm/s]
Skin Infill:	30	60	[mm/s]

Quality Tab Settings:

Default Quality: 0.2 mm

Selected Quality Setting:

Name:	Value
Layer Height:	0.2 mm
First Layer Height:	0.3 mm
First Layer Extrusion Width:	100

Extrusion Tab Settings:

General Extruder Settings

- ☐ Spiralize Contour
- ☒ Enable Retraction
- Retraction Speed: 40 [mm/s]
- Retraction Distance: 4 [mm]
- Minimum Travel before Retract: 1.5 [mm]
- Minimum Extrusion before Retract: 0.02 [mm]
- Z Hop: 0 [mm]
- Cut off Object Bottom: 0 [mm]
- Nozzle Diameter: 0 [mm or 0 = use value]
- Minimize Crossing Perimeters: Always

Multi Extruder Settings

- ☒ Create Wipe and Prime Tower
- ☐ Create Ooze Shield
- Support Extruder: Any Extruder
- Retraction on Extruder Switch: 16 [mm]
- Wipe and Prime Volume: 15 [mm³]
- Volume Overlap: 0 [mm]

Cooling

- ☐ Cool Head Lift
- Fan full at Height: 0.5 [mm]
- Minimum Speed: 0 [mm/s]

Support Tab Settings:

Support Pattern: Grid

Parameter	Value	Unit
Overhang Angle:	60	[°]
Fill Amount:	15	[%]
Distance XY:	0.7	[mm]
Distance Z:	0.15	[mm]

Skirt and Brim

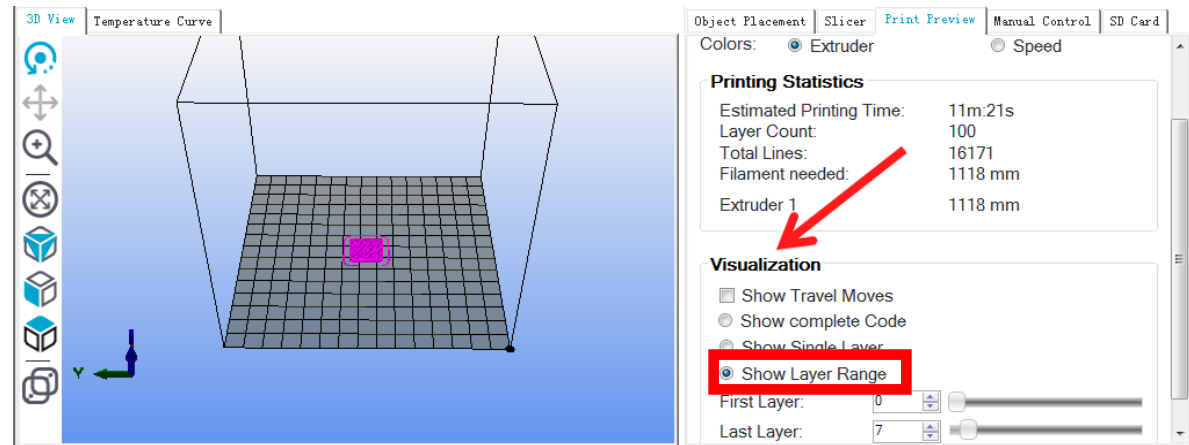
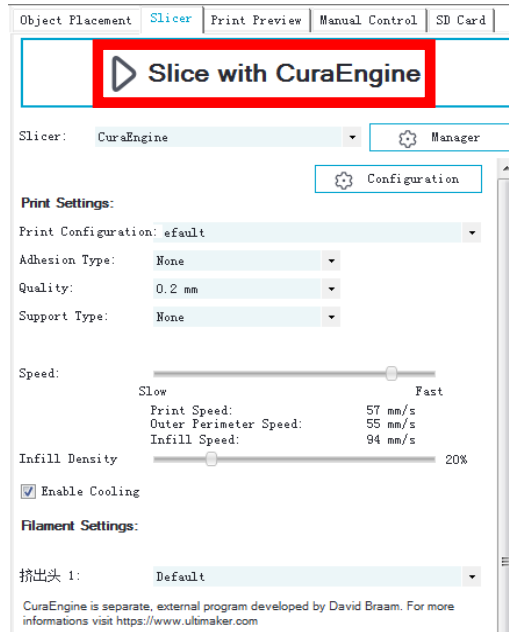
Parameter	Value	Unit
Skirt Line Count:	1	
Skirt Distance:	3	[mm]
Minimum Skirt Length:	150	[mm]

Raft

Parameter	Value	Unit
Extra Margin:	5	[mm]
Base Line Thickness:	0.3	[mm]
Interface Thickness:	0.2	[mm]

Slicing and Pre view

After setting configuration parameters, click "Save" to save it, and then click "start to slice". waiting for completed, and then select "Show specified layer" to preview the result of slicing.



Print it

After sliced successfully , you can print it by:

Click "Save to file" to save the file to the SD card, then insert SD card into a 3D printer and start printing from SD card, or

Click "Print" to print it from PC.

