

## Cura setup for S3D printer

### Download Cura

Visit this link <http://t.cn/RSI22Ib>

Choose the latest version for your computer (available for Windows, macOS and Linux)

Choose "I don't want to share any information" to download

### You're about to download Cura 2.5 For Windows

Before you download Cura please fill in these details. This will help to improve our products. Thank you!

\* Required Fields

\*  
I don't want to share any information ▼

Download

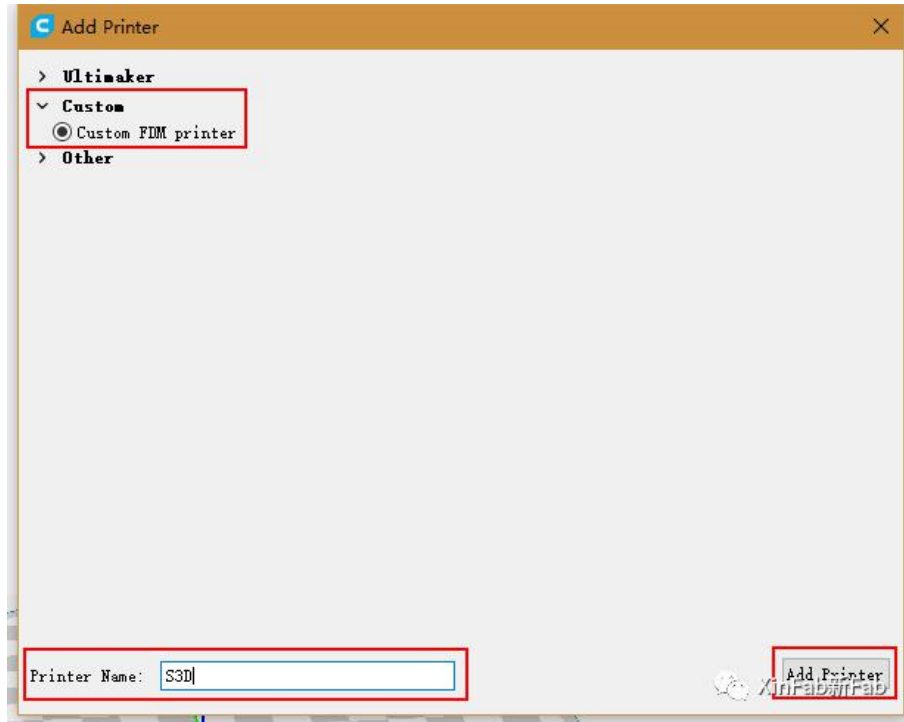
XinFab 新Fab

## Add Printer

1. Choose "Custom - Custom FDM printer"

Name the printer as "S3D"

Click the button "Add Printer"



## 2. Machine Settings

### Printer Settings

X (Width): 160 mm

Y (Depth): 160 mm

Z (Height): 160 mm

Build Plate Shape: Rectangular

Heated Bed: Checked

GCode Flavor: RepRap(Marlin/Sprinter)

### Printhead Settings

X min: 20 mm

Y min: 20 mm

X max: 30 mm

Y max: 30 mm

Gantry height: 60 mm

Nozzle size: 0.4 mm

The screenshot shows the 'Add Printer' dialog box with the 'Machine Settings' tab selected. The settings are organized into two main sections: 'Printer Settings' and 'Printhead Settings', both highlighted with red boxes. The 'Printer Settings' section includes input fields for X (Width), Y (Depth), and Z (Height), all set to 160 mm. It also features a dropdown for 'Build Plate Shape' set to 'Rectangular', a checkbox for 'Machine Center is Zero' which is unchecked, a checked checkbox for 'Heated Bed', and a dropdown for 'GCode Flavor' set to 'RepRap (Marlin/Sprinter)'. The 'Printhead Settings' section includes input fields for X min, Y min, X max, and Y max, all set to 20 mm, 20 mm, 30 mm, and 30 mm respectively. It also has a dropdown for 'Gantry height' set to 60 mm and a dropdown for 'Nozzle size' set to 0.4 mm. At the bottom, there are two text areas for 'Start Gcode' and 'End Gcode'. The 'Start Gcode' area contains the following text: 'G28 ;Home', 'G1 Z15.0 F6000 ;Move the platform down.', ';Prime the extruder', 'G92 E0', 'G1 F200 E3', and 'G92 E0'. The 'End Gcode' area contains the following text: 'M104 S0', 'M140 S0', ';Retract the filament', 'G92 E1', 'G1 E-1 F300', 'G28 X0 Y0', and 'M84'. At the bottom right, there are 'Back' and 'Finish' buttons, with the 'Finish' button highlighted by a red box.

**Machine Settings**

Please enter the correct settings for your printer below:

**Printer Settings**

X (Width) 160 mm

Y (Depth) 160 mm

Z (Height) 160 mm

Build Plate Shape Rectangular

☐ Machine Center is Zero

☒ Heated Bed

GCode Flavor RepRap (Marl...

**Printhead Settings**

X min 20 mm

Y min 20 mm

X max 30 mm

Y max 30 mm

Gantry height 60 mm

Nozzle size 0.4 mm

**Start Gcode**

```
G28 ;Home
G1 Z15.0 F6000 ;Move the platform down.
;Prime the extruder
G92 E0
G1 F200 E3
G92 E0
```

**End Gcode**

```
M104 S0
M140 S0
;Retract the filament
G92 E1
G1 E-1 F300
G28 X0 Y0
M84
```

Back Finish

## Print Setup

1. Look at the settings window in the right,  
Material: PLA  
Print Setup: Custom

The screenshot shows the 'Print Setup' window of a 3D printing software. At the top, there is a dark blue header with 'S3D' on the left, a dropdown arrow in the center, and two icons on the right: a printer and a document. Below the header, the 'Material' is set to 'PLA' and the 'Profile' is 'Common Setting 0.2mm'. The 'Print Setup' section has two tabs: 'Recommended' and 'Custom', with 'Custom' being the active tab. Under 'Print Setup', there are four infill pattern options: 'Hollow', 'Light', 'Dense', and 'Solid'. The 'Light' pattern is selected. Below these are two checkboxes: 'Enable Support' (checked) and 'Build Plate Adhesion' (unchecked). At the bottom, there is a link to 'Need help improving your prints? Read the [Ultimaker Troubleshooting Guides](#)' and a logo for 'XinFab 新Fab'.

S3D

Material: PLA

Profile: Common Setting 0.2mm

**Print Setup**

Recommended Custom

Infill

Hollow Light Dense Solid

Enable Support ☒

Build Plate Adhesion ☐

Need help improving your prints? Read the [Ultimaker Troubleshooting Guides](#)

XinFab 新Fab

2. Make sure your default settings are same as below:

The screenshot shows the S3D software interface with the following settings:

- Material:** PLA
- Profile:** Common Setting 0.2mm
- Print Setup:** Recommended (Custom is selected)
- Quality:**
  - Layer Height: 0.2 mm
  - Initial Layer Height: 0.3 mm
- Shell:**
  - Wall Thickness: 0.8 mm
  - Top/Bottom Thickness: 0.8 mm
  - Top/Bottom Pattern: Lines
- Infill:**
  - Infill Density: 20 %
  - Infill Pattern: Grid
- Material:**
  - Printing Temperature: 210 °C
  - Build Plate Temperature: 60 °C
  - Diameter: 1.75 mm
  - Flow: 100 %
  - Enable Retraction: ☐
- Speed:**
  - Print Speed: 60 mm/s
  - Travel Speed: 150 mm/s
- Support:**
  - Enable Support: ☒
  - Support Placement: Everywhere
  - Support Pattern: Grid
  - Support Density: 15 %
- Build Plate Adhesion:**
  - Build Plate Adhesion Type: Brim
  - Brim Width: 8.0 mm

If you don't have the complete settings showing in the picture, please hover your mouse on the column of "Quality" and a gear will appear.

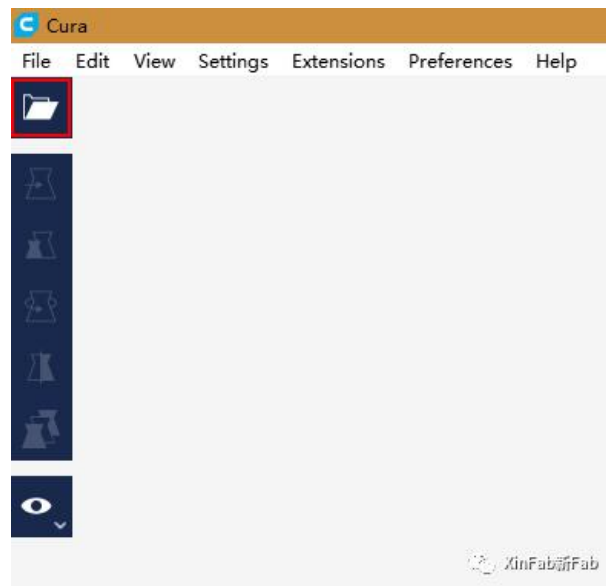
Click it to open the "Setting Visibility" window and select the settings you need.

The screenshot shows the 'Setting Visibility' window with the following settings:

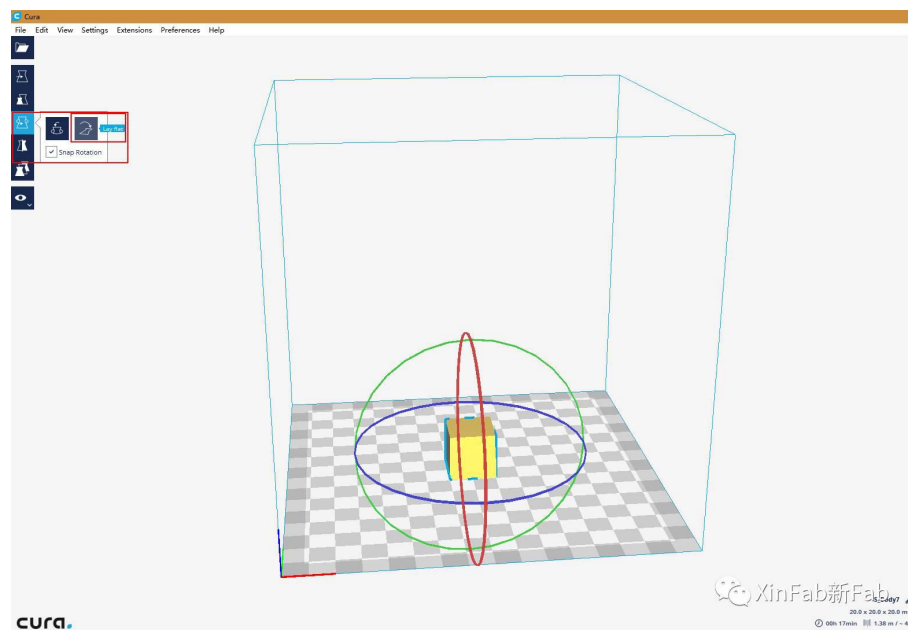
- Check all:** ☒ Filter...
- Quality:**
  - ☒ Layer Height
  - ☒ Initial Layer Height
  - ☐ Line Width
  - ☐ Wall Line Width
    - ☐ Outer Wall Line Width
    - ☐ Inner Wall(s) Line Width
  - ☐ Top/Bottom Line Width
  - ☐ Infill Line Width
  - ☐ Skirt/Brim Line Width
  - ☐ Support Line Width
  - ☒ Support Interface Line Width
  - ☒ Prime Tower Line Width
- Shell:**
  - ☒ Wall Thickness
  - ☒ Wall Line Count
  - ☐ Outer Wall Wipe Distance

## Check the Model

1. Open your stl file



2. Click on your model and use the tools on the left side to find a perfect position to print.



3. Please check the estimated printing time and material cost on the right bottom.

Based on this, you can adjust the settings to what you expect.



4. Save your file as ".gcode" file into the sd card.

Ready to Save to File

