

# Cura 4.6 Tutorial

## General Process

1. Before printing, save the 3D model as a STL file.
2. Import the file into Cura to slice.
3. Export the sliced 3D model as a GCODE file.
4. Copy the GCODE file into a TF card and insert the card into the 3D printer  
- pay attention to the inserting direction.
5. Select the sliced file and start printing.

## How to use Cura?

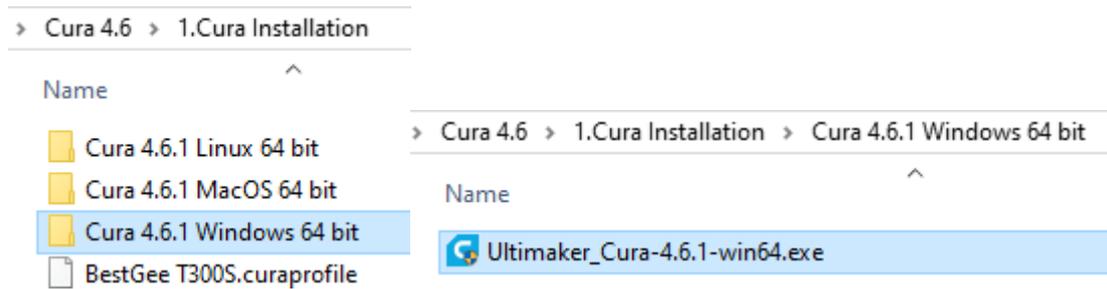
1. We suggest beginners to use Cura to slice the 3D model.
2. Slicing parameters are set according to the characteristics of different models.
3. Each time before you start printing, make sure that the printer is in good condition, leveling has been done and the distance between the heated bed and the nozzle is about 0.1 mm (thickness of a printing paper).
4. Most of the parameters are default. If you need to make changes, the changes should be set depend on the 3D model that you are going to print. At the same time, you need to be proficient in the slicing software.



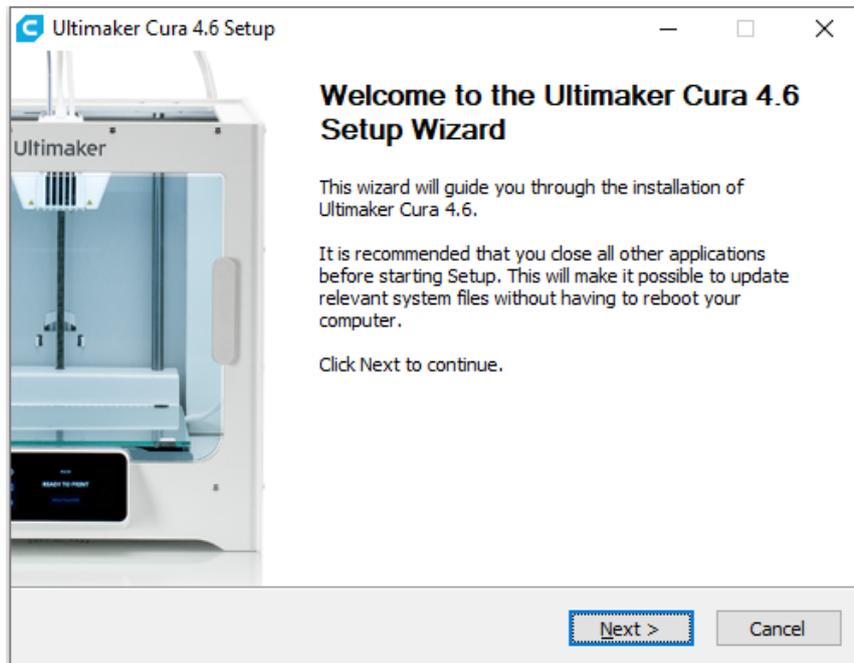
Cura supports Windows, MacOS and Linux. This tutorial takes Windows for example, MacOS and Linux are similar.

# 1. Install Cura

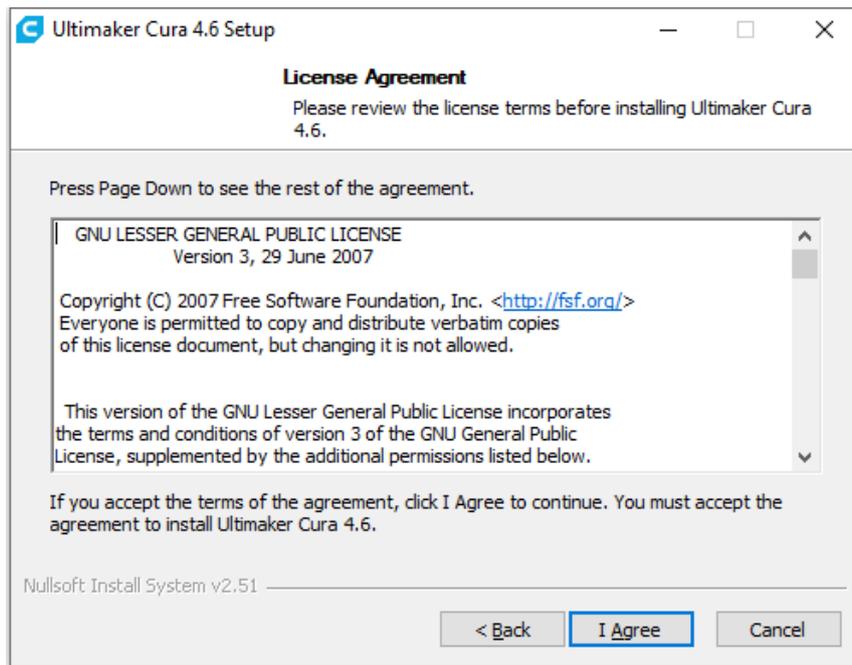
1.1. Open the installation file  **Ultimaker\_Cura-4.6.1-win64** in the TF card at “\\Cura 4.6\1.Cura Installation\Cura 4.6.1 Windows 64 bit\” (take Windows for example).



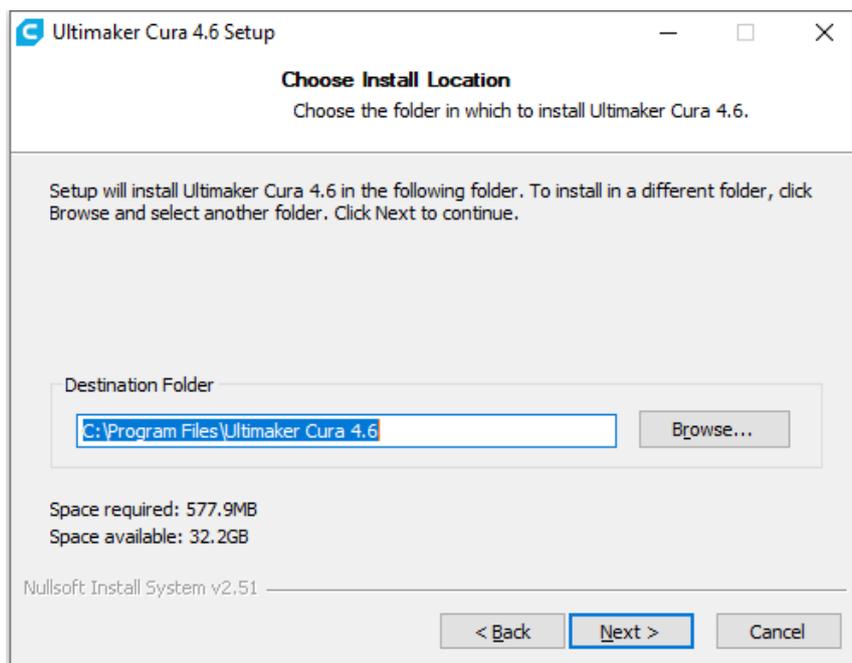
1.2. Click “Next”.



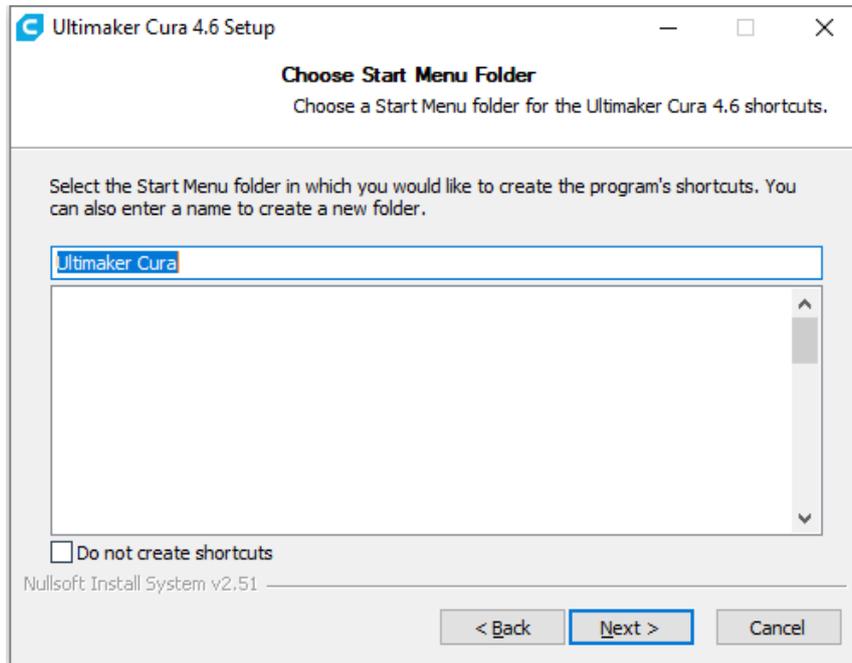
1.3. Click "I Agree".



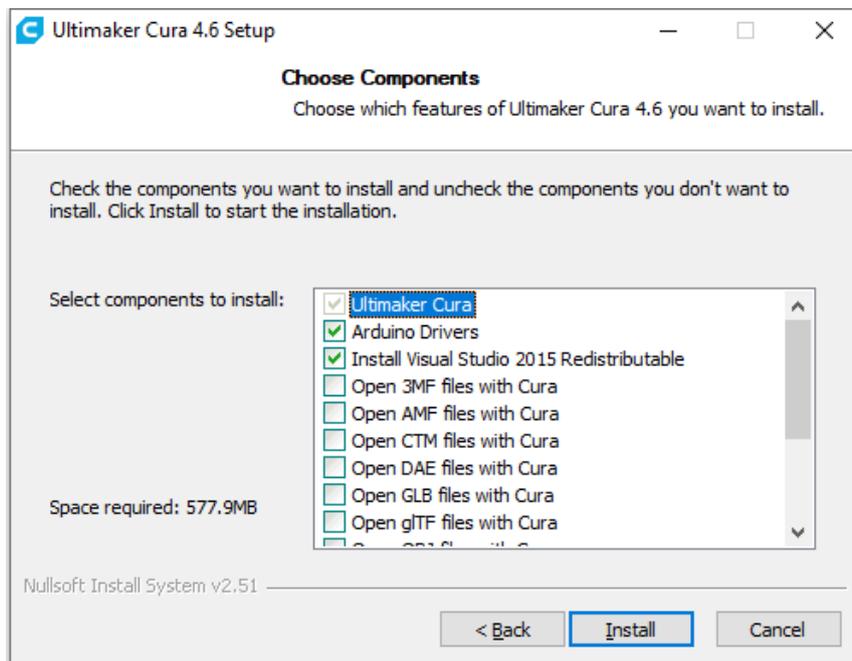
1.4. Choose the folder in which to install Cura, then click "Next".

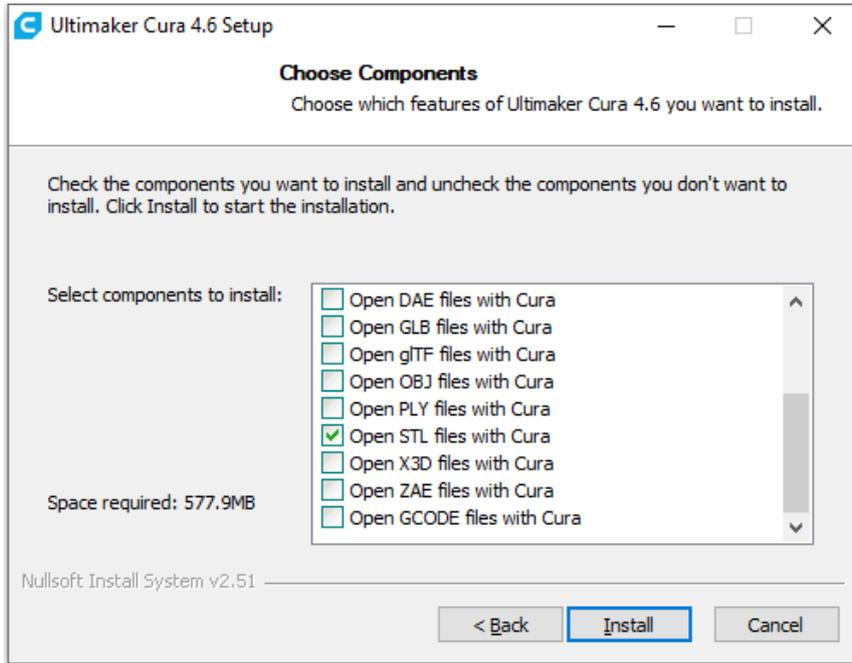


1.5. Click “Next”.

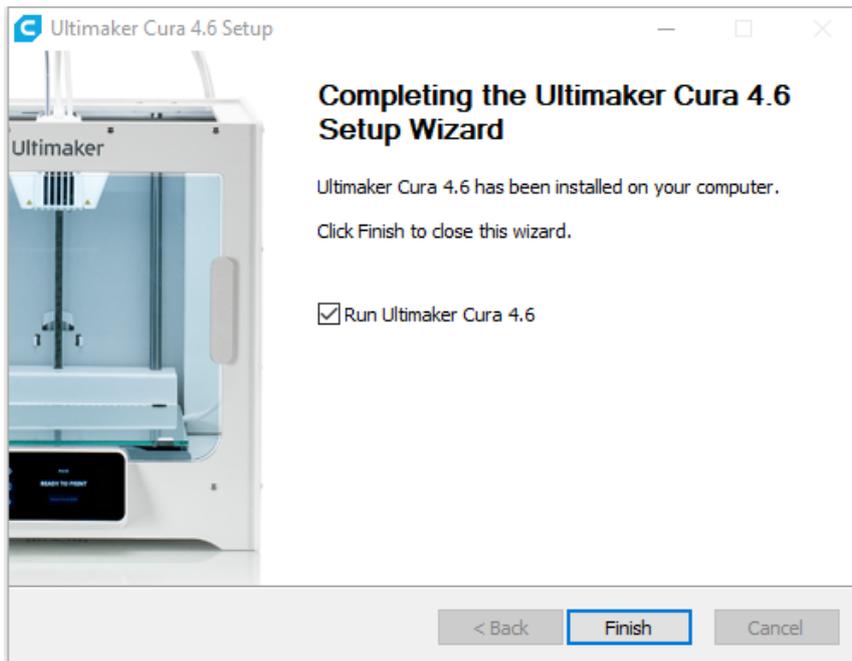


1.6. Choose the supported file format you need, STL is the default and recommended to choose. Click “Install” and start the installation.



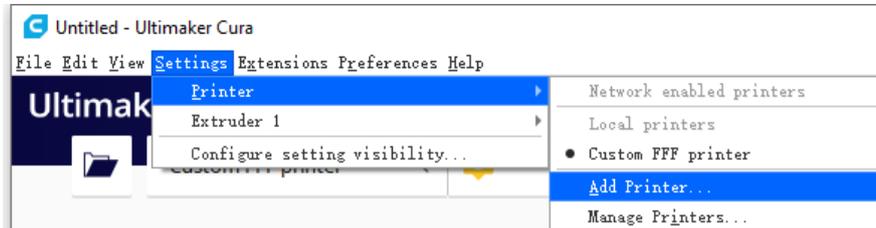


1.7. Wait until the installation is finished, then click “Finish”.

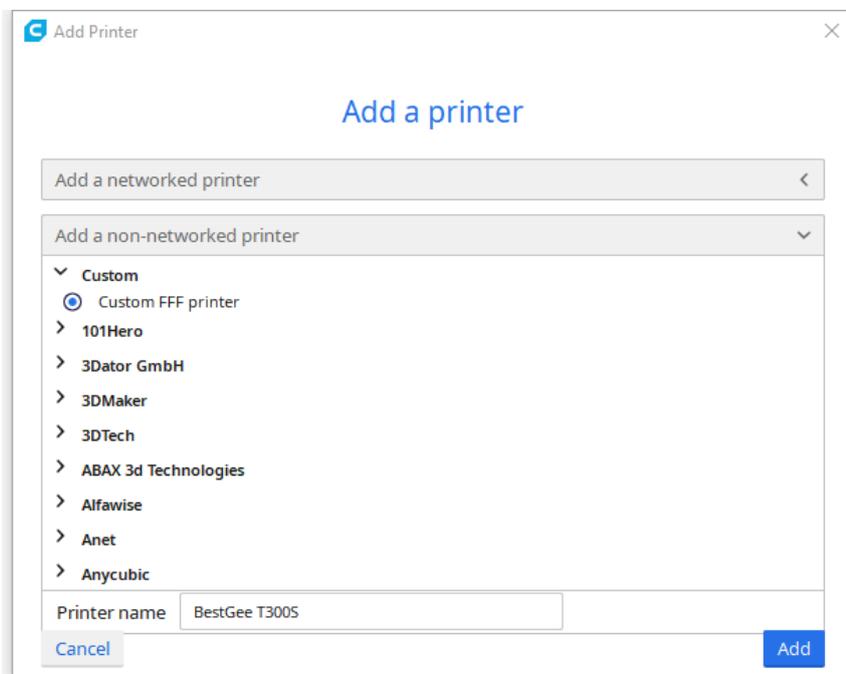


## 2. Add Printer

2.1. Select “Setting – Printer – Add Printer...”.



2.2. Choose “Add a non-networked printer – Custom – Custom FFF printer” and input a printer name, for example BestGee T300S.



2.3. Set printer and extruder as shown below.

**Machine Settings**

**BestGee T300S**

Printer	Extruder 1
<b>Printer Settings</b>	<b>Printhead Settings</b>
X (Width) 300 mm	X min 0 mm
Y (Depth) 300 mm	Y min 0 mm
Z (Height) 400 mm	X max 300 mm
Build plate shape: Rectangular	Y max 300 mm
Origin at center: <input type="checkbox"/>	Gantry Height 400 mm
Heated bed: <input checked="" type="checkbox"/>	Number of Extruders: 1
Heated build volume: <input type="checkbox"/>	Shared Heater: <input type="checkbox"/>
G-code flavor: Marlin	
Start G-code: G28 ;Home	End G-code: M104 S0

**G-code List:**

- G1 F2000 X0 Y0
- M104 S0
- M140 S0
- ;Retract the filament
- G92 E1
- G1 E-1 F300
- G28 XY
- M84

**Machine Settings**

**BestGee T300S**

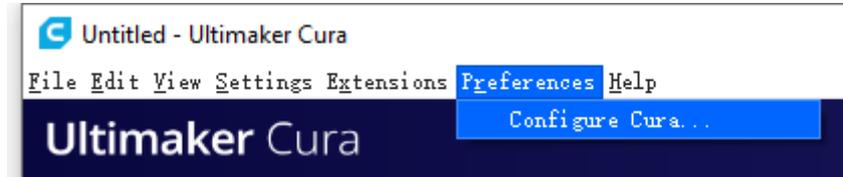
Printer	Extruder 1
	<b>Nozzle Settings</b>
	Nozzle size 0.4 mm
	Compatible material diameter 1.75 mm
	Nozzle offset X 0 mm
	Nozzle offset Y 0 mm
	Cooling Fan Number 0
	Extruder Start G-code
	Extruder End G-code

2.4. Click "Next" and done.

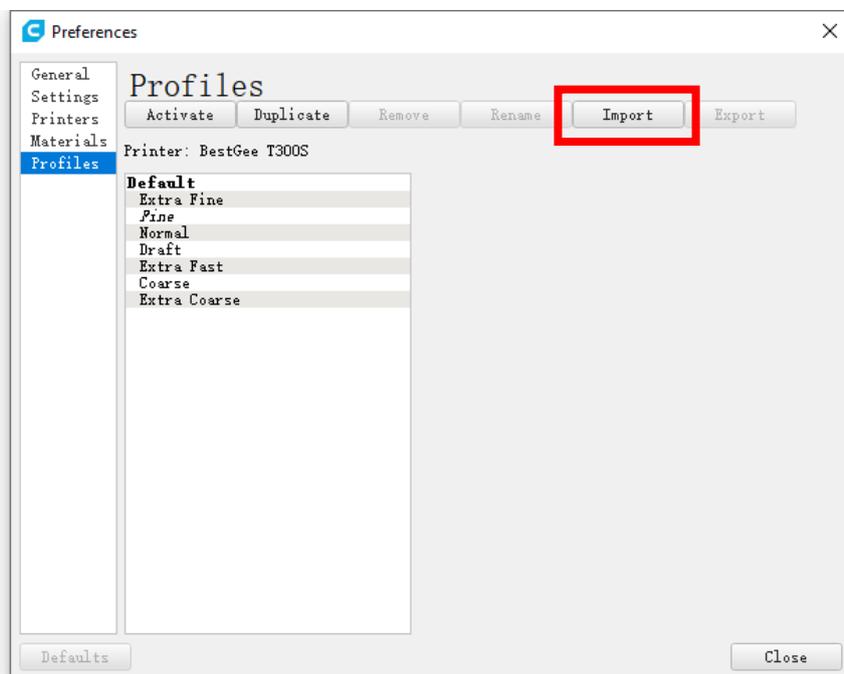


## 3. Import Profile

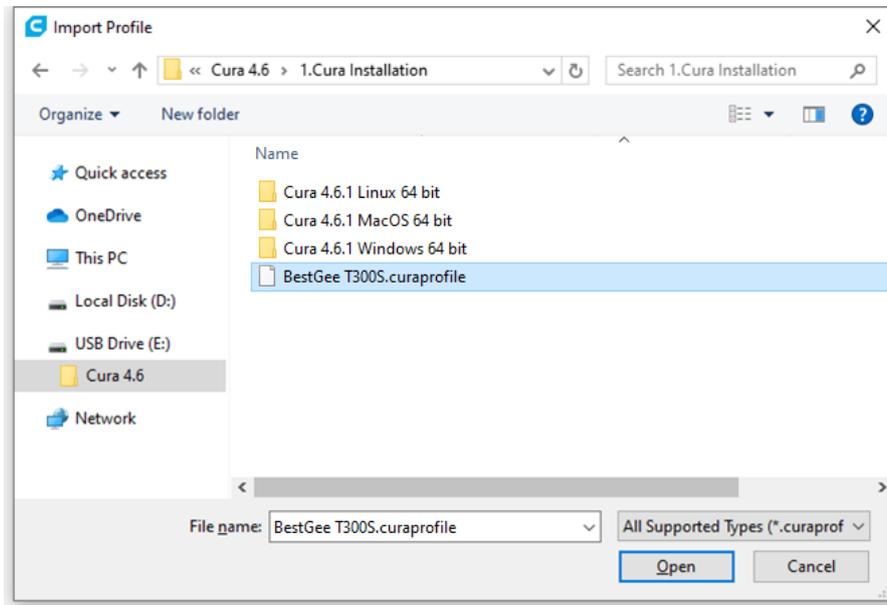
3.1. Select “Preferences – Configure Cura...”



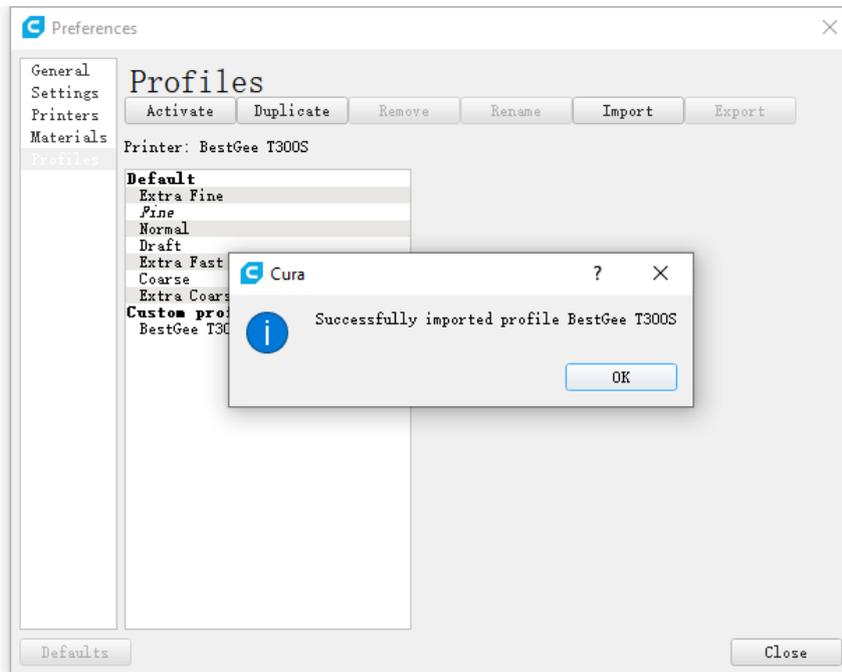
3.2. Select “Profiles” and Click “Import”.



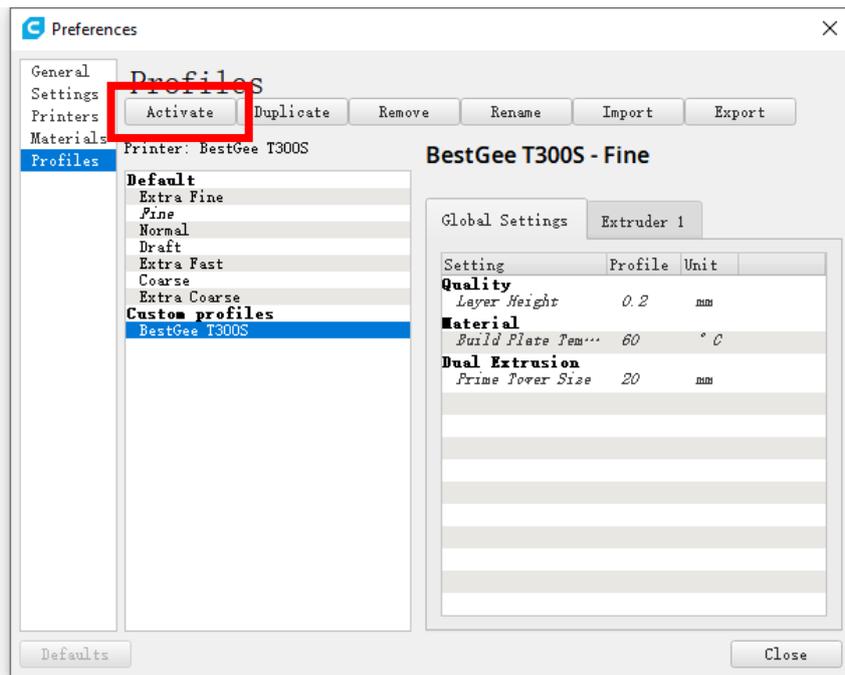
3.3. Open the profile file at “\\Cura 4.6\1.Cura Installation”.



3.4. Click “OK”.



3.5. Select the imported profile and click “Activate” to set the profile.



3.6. Click “Close” and done.

