Cura is an open source 3D printer slicing application. It works by slicing the user’s model file into layers and generating a printer-specific g-code. Once finished, the g-code can be sent to the printer for the manufacture of the physical object.

The open source software, compatible with most desktop 3D printers, can work with files in the most common 3D formats such as STL, OBJ, X3D, 3MF as well as image file formats such as BMP, GIF, JPG, and PNG.
As of the writing of this article, Cura is in version 4.8. It works on all common OS platforms: Windows, Mac, and Linux. The minimum system requirements for Cura are:

Windows Vista or newer
Mac OSX 10.7 or newer
Linux Ubuntu 15.04, Fedora 23, OpenSuse 13.2, ArchLinux or newer

To install Cura, first download it for your OS from this page. When the Cura download is complete, here’s what you need to do on each platform.

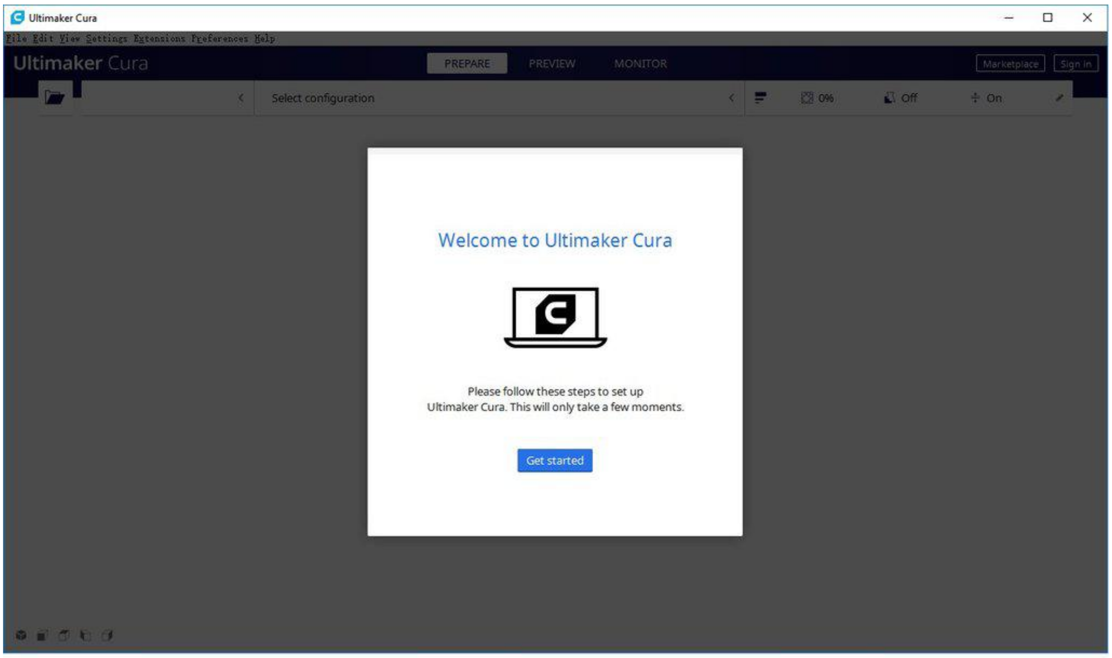
Cura Download and Installation

Run the Cura installer and go through the usual steps. The only non-trivial part of the installation is the following screen, which gives you the option to install additional components.

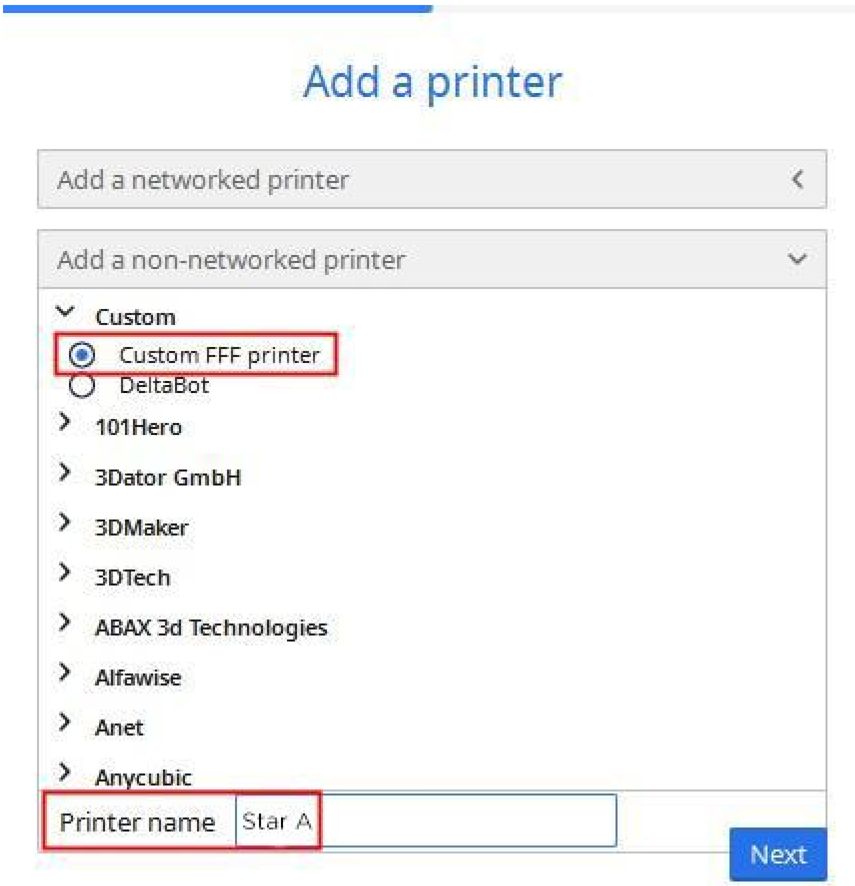
Follow the instructions and set it up.

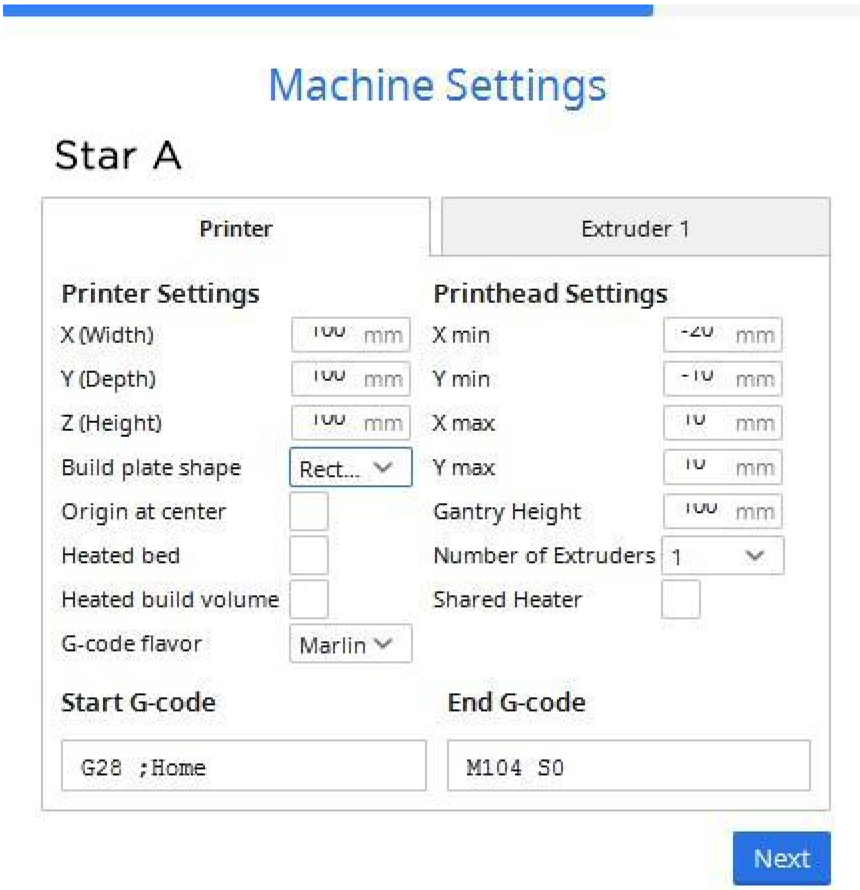

Ultimaker Cura 4.8 Setup
Welcome to the Ultimaker Cura 4.8 Setup Wizard
This wizard will guide you through the installation of Ultimaker Cura 4.8.
It is recommended that you close all other applications before starting Setup. This will make it possible to update relevant system files without having to reboot your computer.
Click Next to continue.
Next >
Cancel

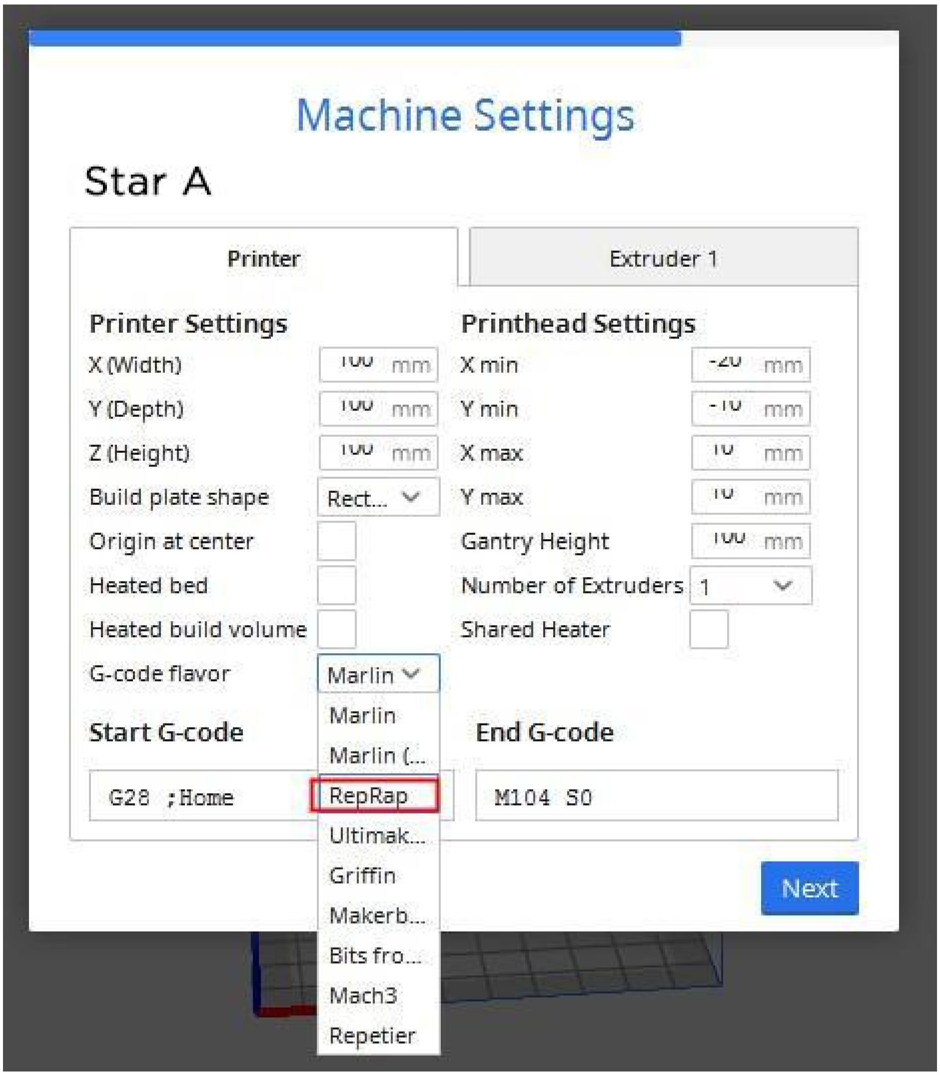

Ultimaker Cura 4.8 Setup
Completing the Ultimaker Cura 4.8 Setup Wizard
Ultimaker Cura 4.8 has been installed on your computer.
Click Finish to close this wizard.
Run Ultimaker Cura 4.8
< Back
Finish
Cancel



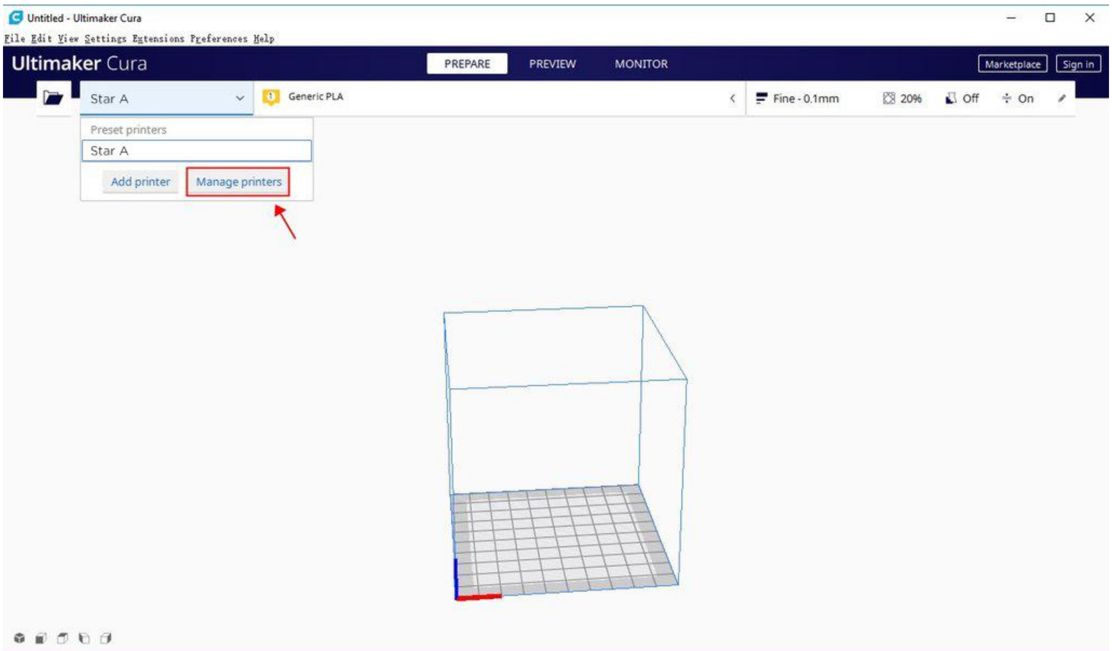
Add a printer - Custom - Custom FFF printer - Printer Named "Selpic Star A"

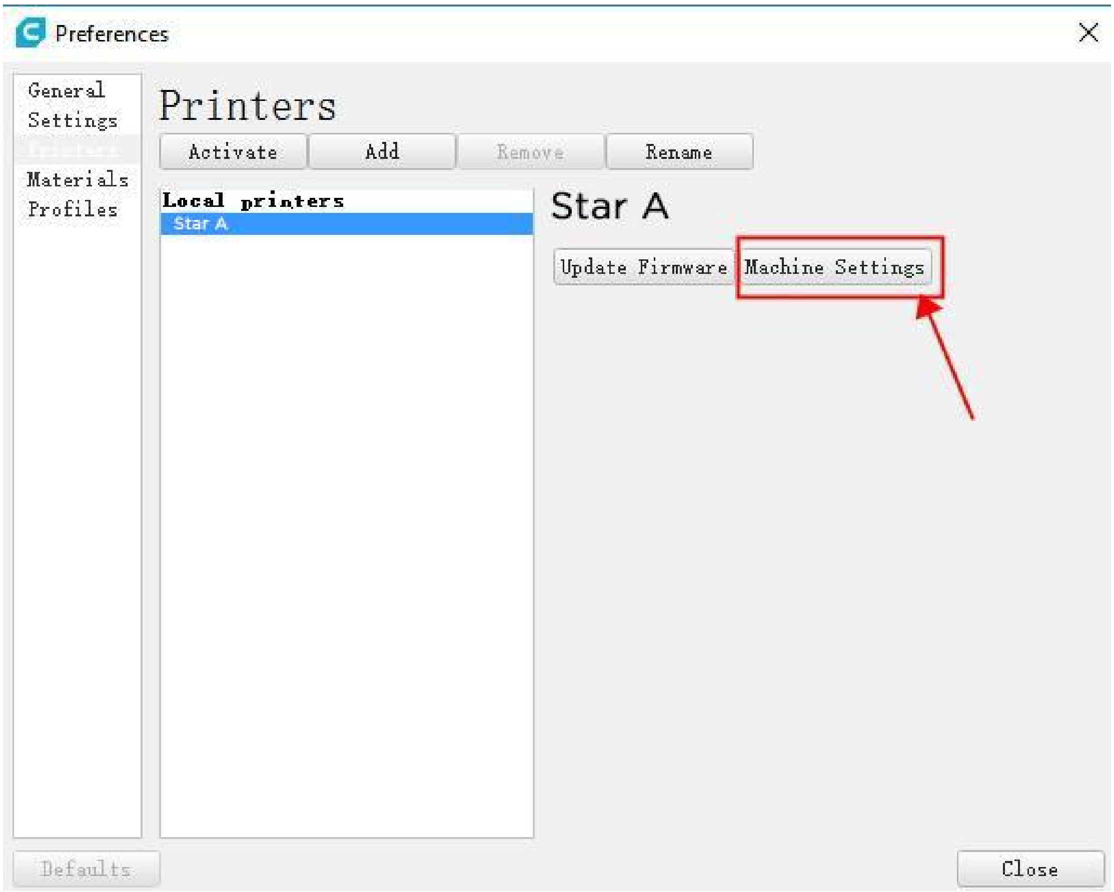


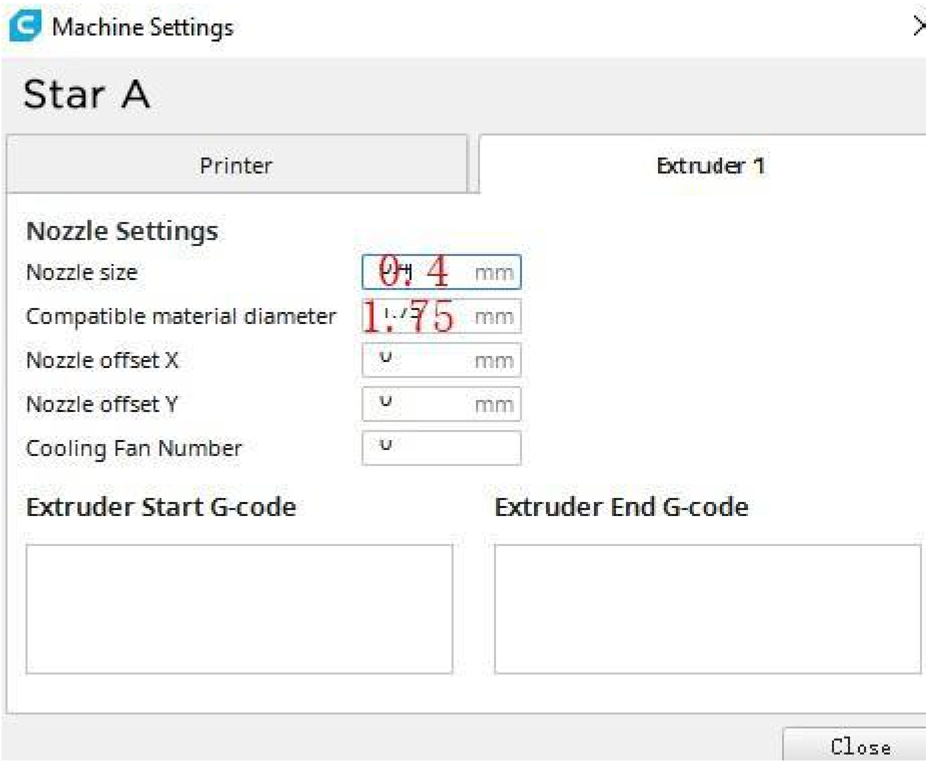

Machine Settings
Star A
Printer
Extruder 1
Printer Settings
X (Width) mm
Y (Depth) mm
Z (Height) mm
Build plate shape Rect...
Origin at center
Heated bed
Heated build volume
G-code flavor Marlin
Printhead Settings
X min mm
Y min mm
X max mm
Y max mm
Gantry Height mm
Number of Extruders 1
Shared Heater
Start G-code
G28 ;Home
End G-code
M104 S0
Next

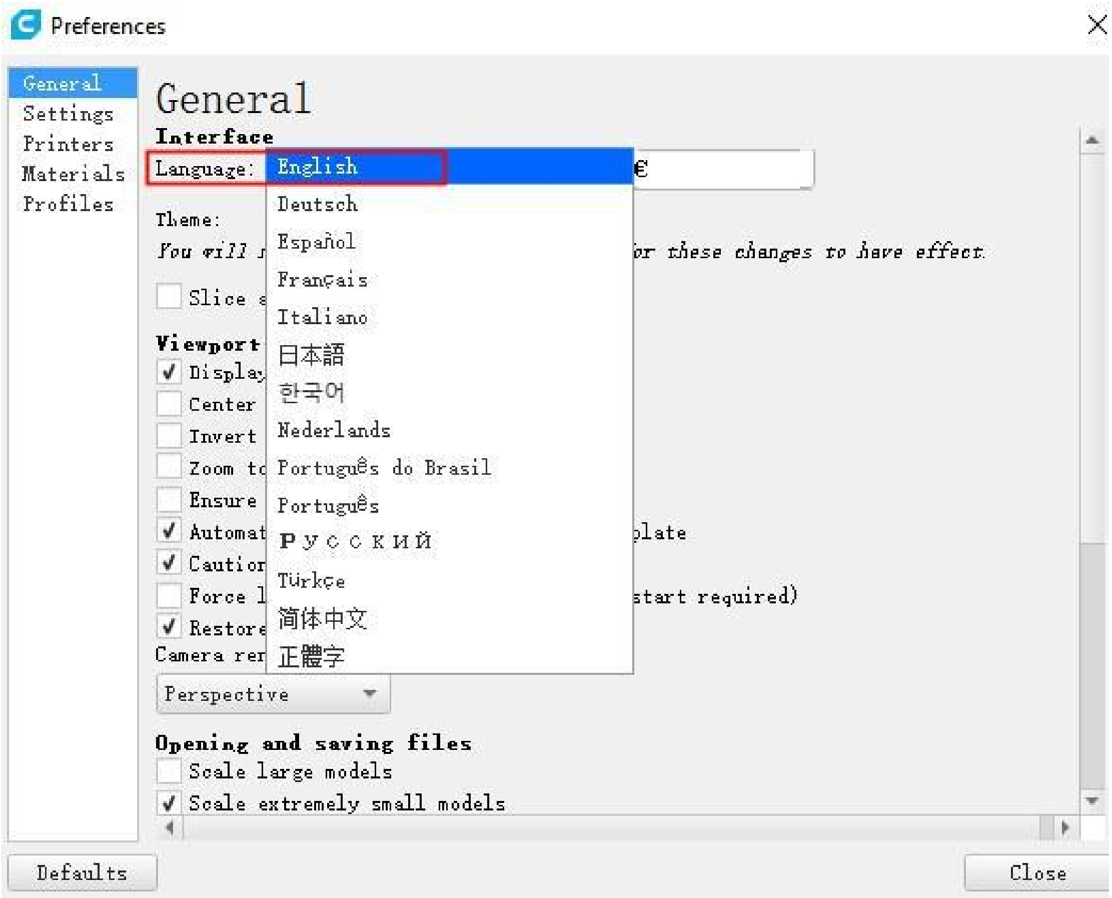


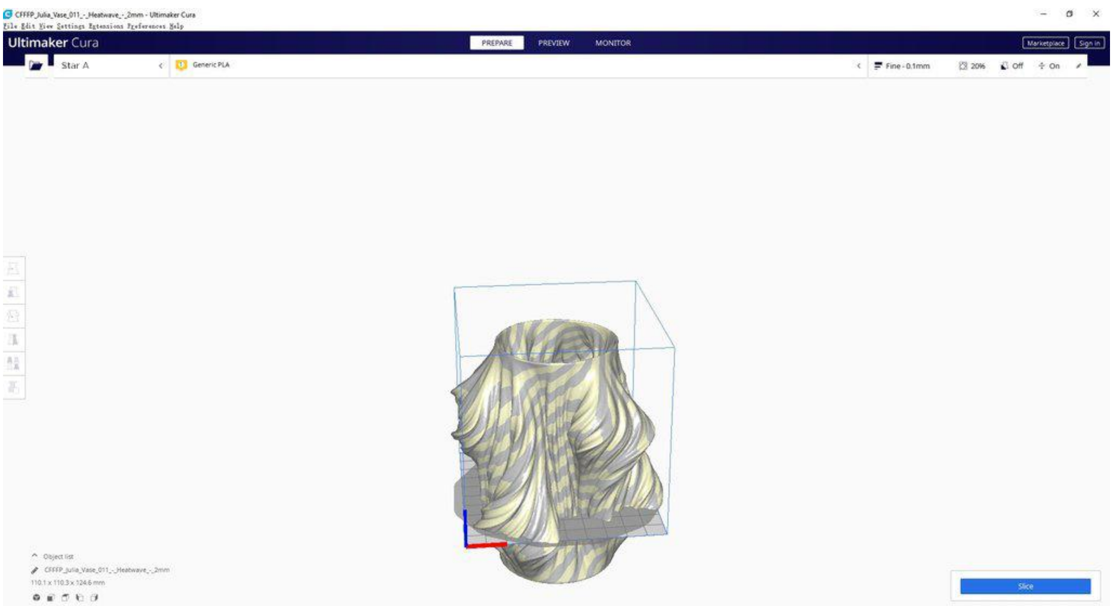
Manage Printers - You can change the printer settings here.



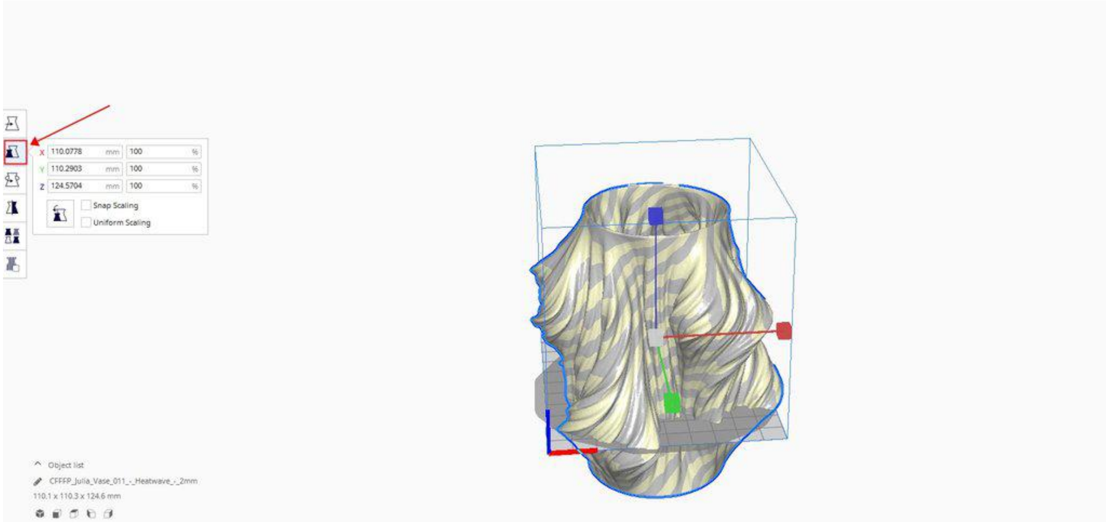

Preferences
General
Settings
Materials
Profiles
Printers
Activate
Add
Remove
Rename
Local printers
Star A
Star A
Update Firmware
Machine Settings
Defaults
Close


Machine Settings
Star A
Printer
Extruder 1
Nozzle Settings
Nozzle size
0.4
mm
Compatible material diameter
1.75
mm
Nozzle offset X
mm
Nozzle offset Y
mm
Cooling Fan Number
Extruder Start G-code
Extruder End G-code
Close

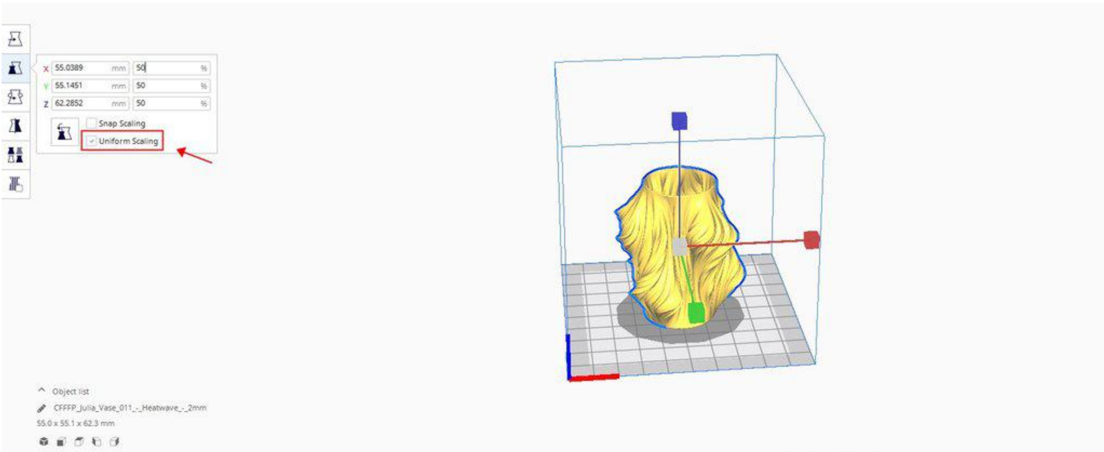


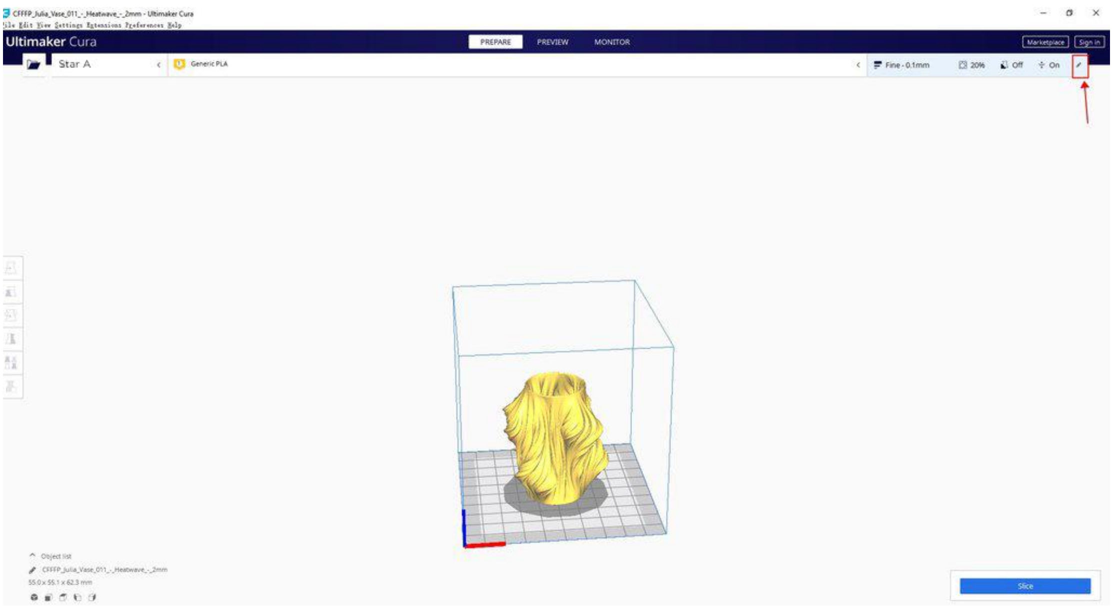


Ops, looks like the object is too big for the printer. Some parts of it are outside of the printing square. Let's change the size and make it printable.



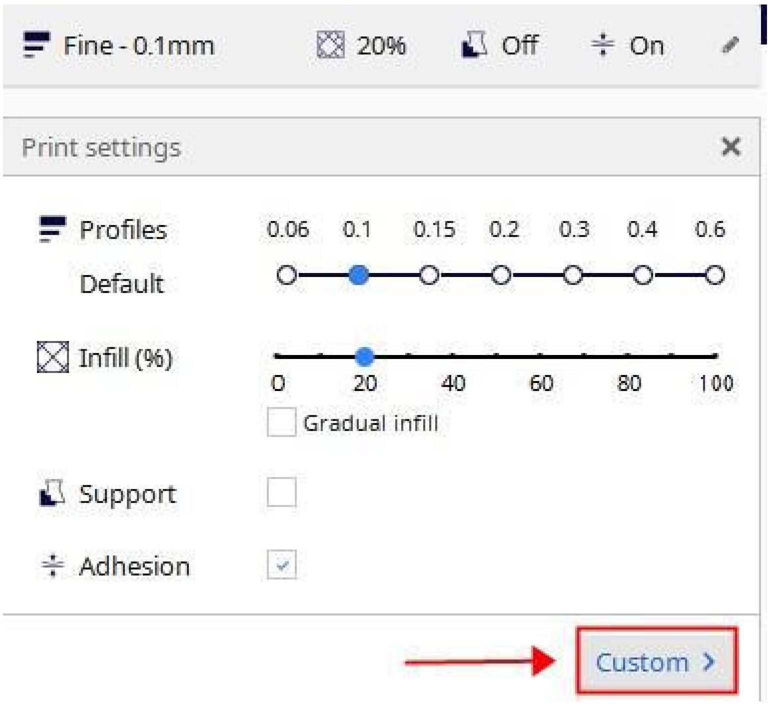
Let's set it to 50%, and please select the "Uniform Scaling". It will decrease the whole project to 50%.





Lets' set the printing settings before slicing. Go to the " Custom" and you can

custom it as you like.



Here is some advice for the settings if you are new to this.

Layer Height: 0.1mm

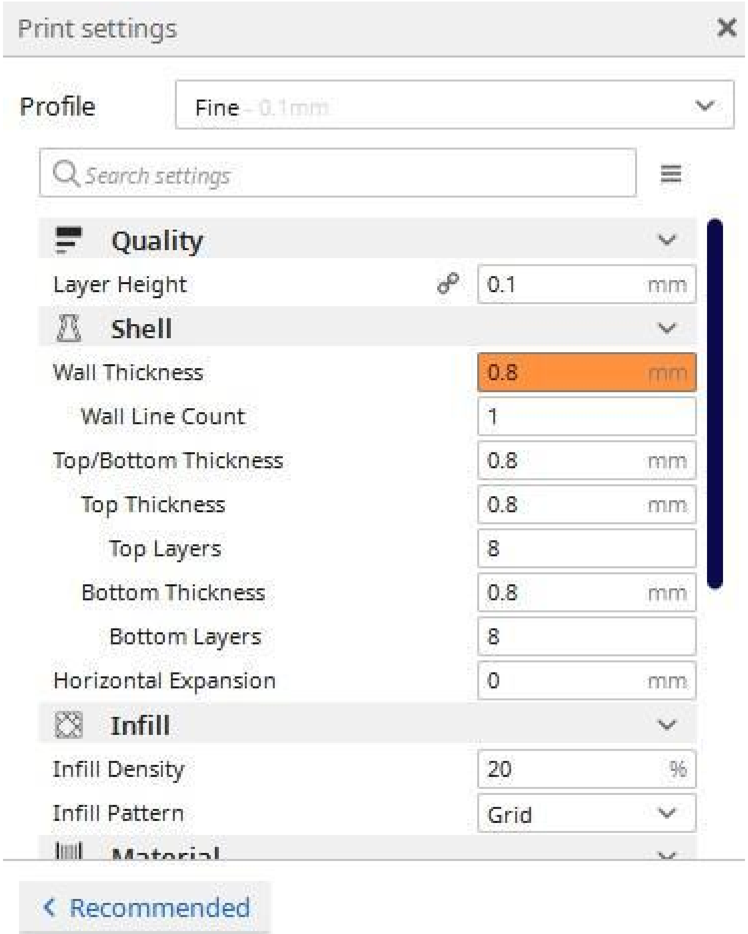
Wall Thickness: 0.8mm

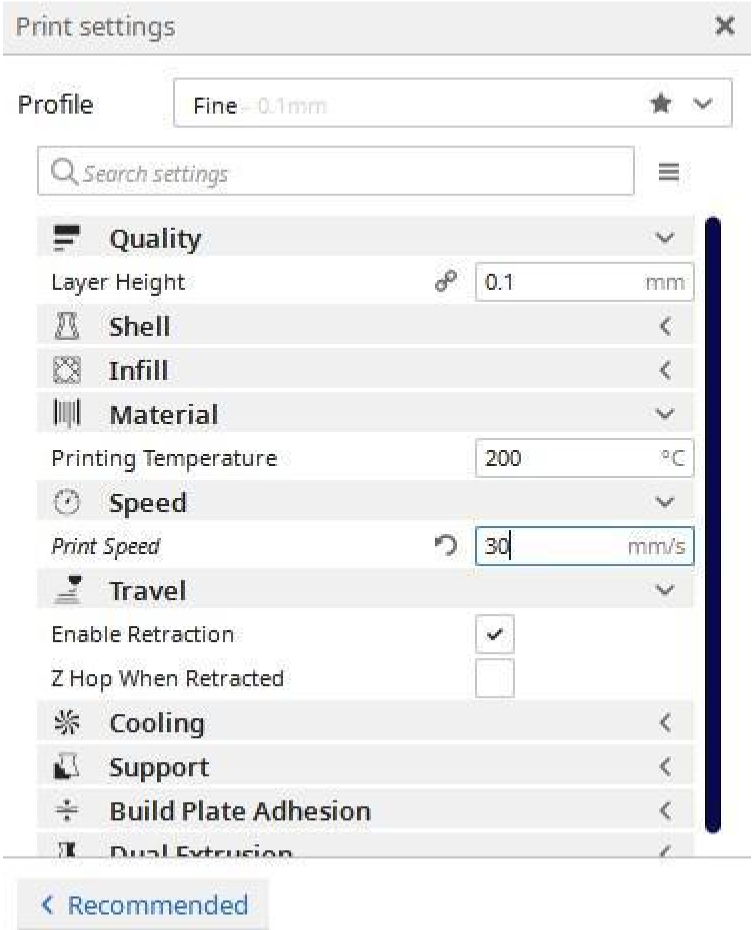
Print temperature: 200

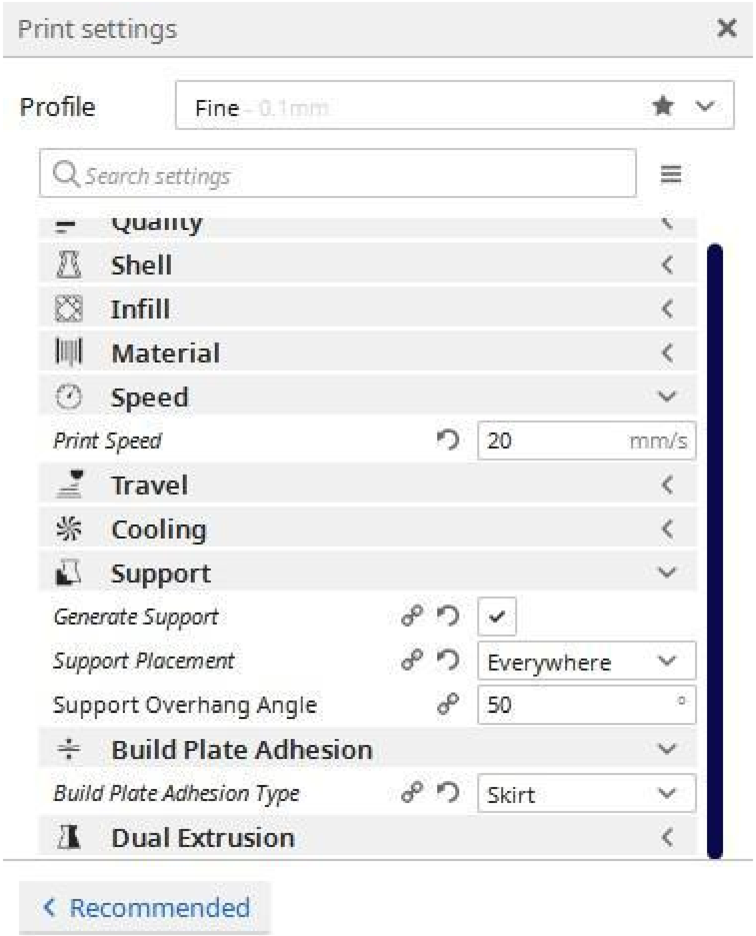
Print speed: 30

Generate support: √

Support Placement: Everywhere


Print settings
Profile
Fine - 0.1mm
Search settings
Quality
Layer Height
0.1 mm
Shell
Wall Thickness
0.8 mm
Wall Line Count
1
Top/Bottom Thickness
0.8 mm
Top Thickness
0.8 mm
Top Layers
8
Bottom Thickness
0.8 mm
Bottom Layers
8
Horizontal Expansion
0 mm
Infill
Infill Density
20 %
Infill Pattern
Grid
Recommended


Print settings
Profile
Fine - 0.1mm
Search settings
Quality
Layer Height
0.1
mm
Shell
Infill
Material
Printing Temperature
200
°C
Speed
Print Speed
30
mm/s
Travel
Enable Retraction
Z Hop When Retracted
Cooling
Support
Build Plate Adhesion
Recommended



After all things done, you can start to slice it. It will show the estimated time for printing.

FAQ: Why my prints goes longer than the estimated time?

A: The estimated time is calculated at a regular speed. But actually it will slow down the speed while goes with the turning and sharp angles.