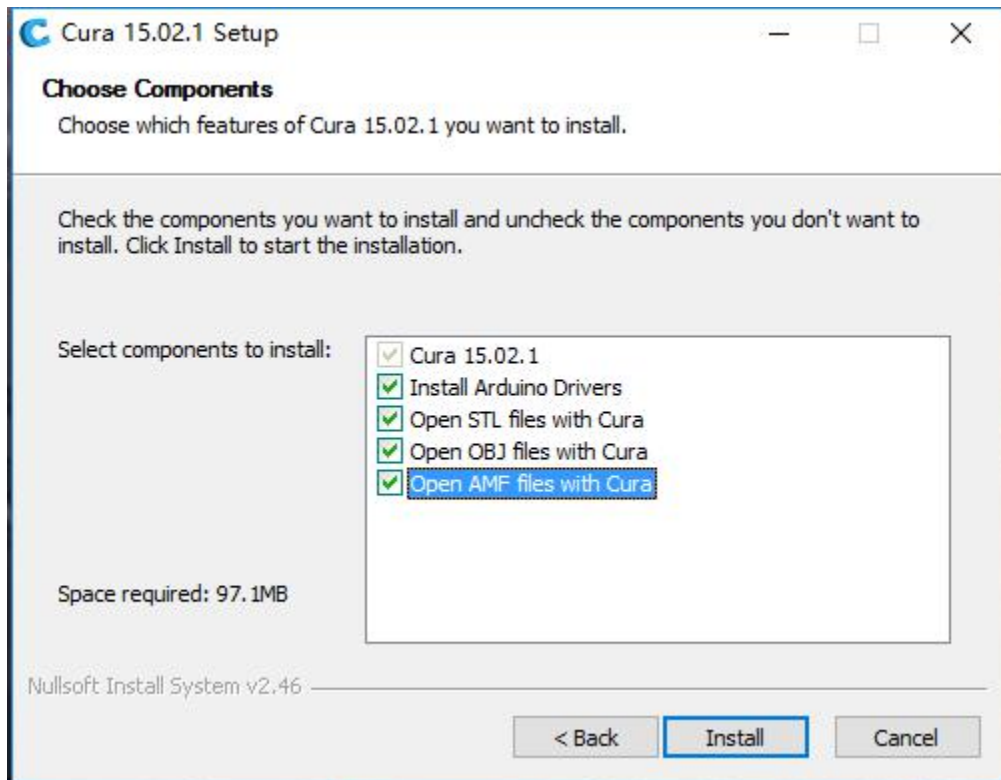
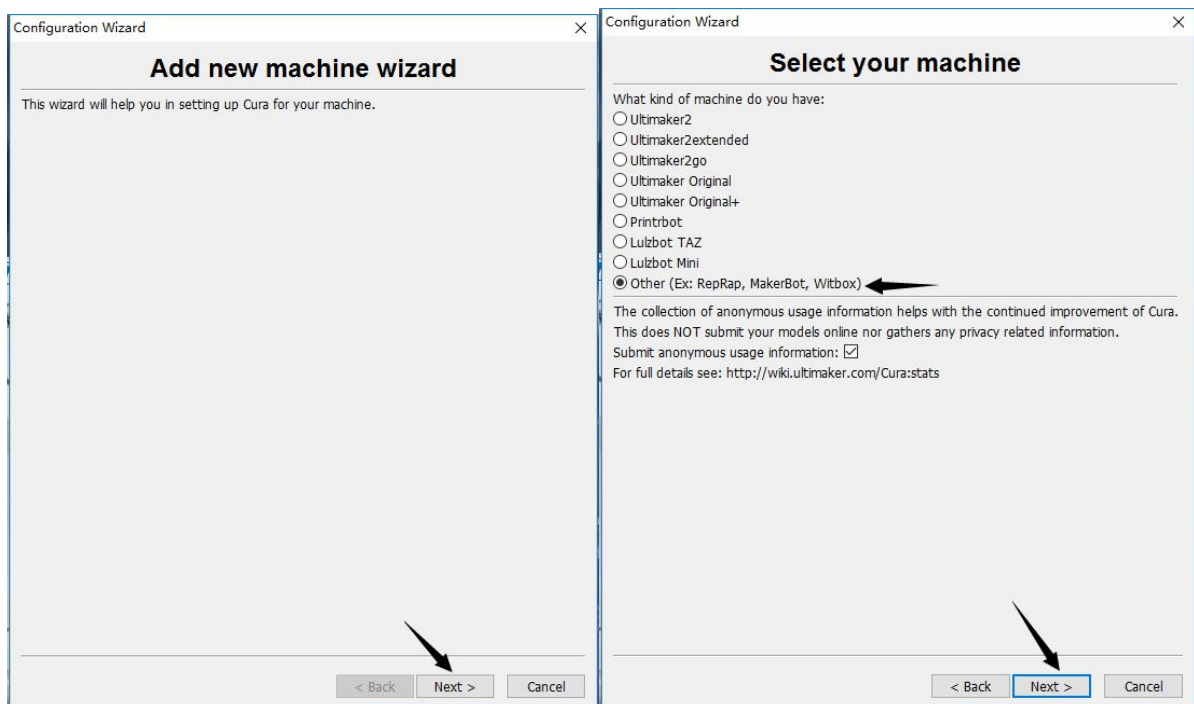


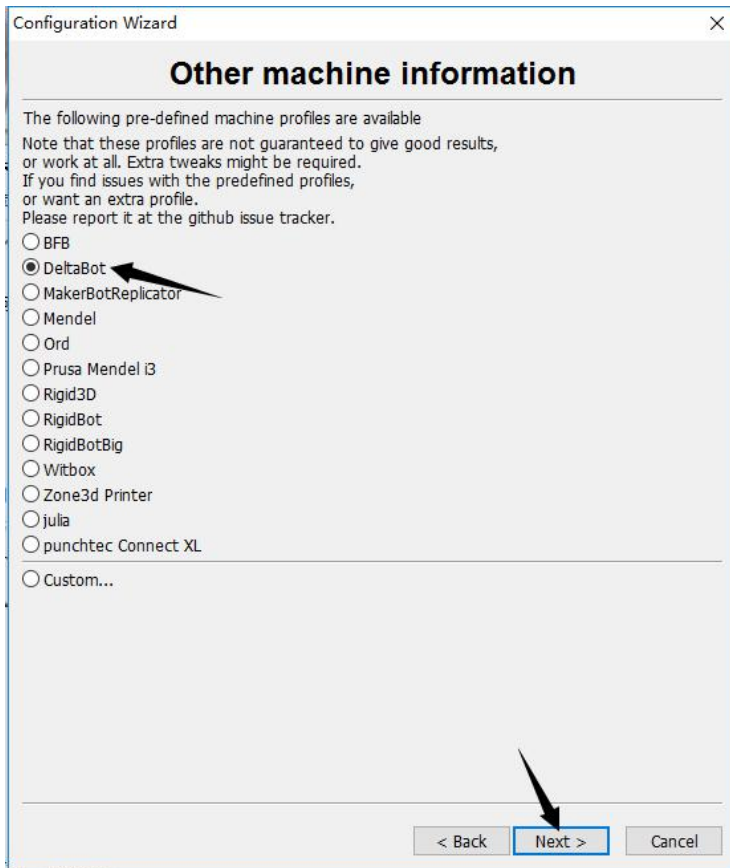
## 1. Printing Models

1) Installing slicer. Taking Cura for example, double click “Cura\_15.04.exe” to install it. Open Cura after installation, and the startup guide dialogue box appears, as figure 6.16 shows.

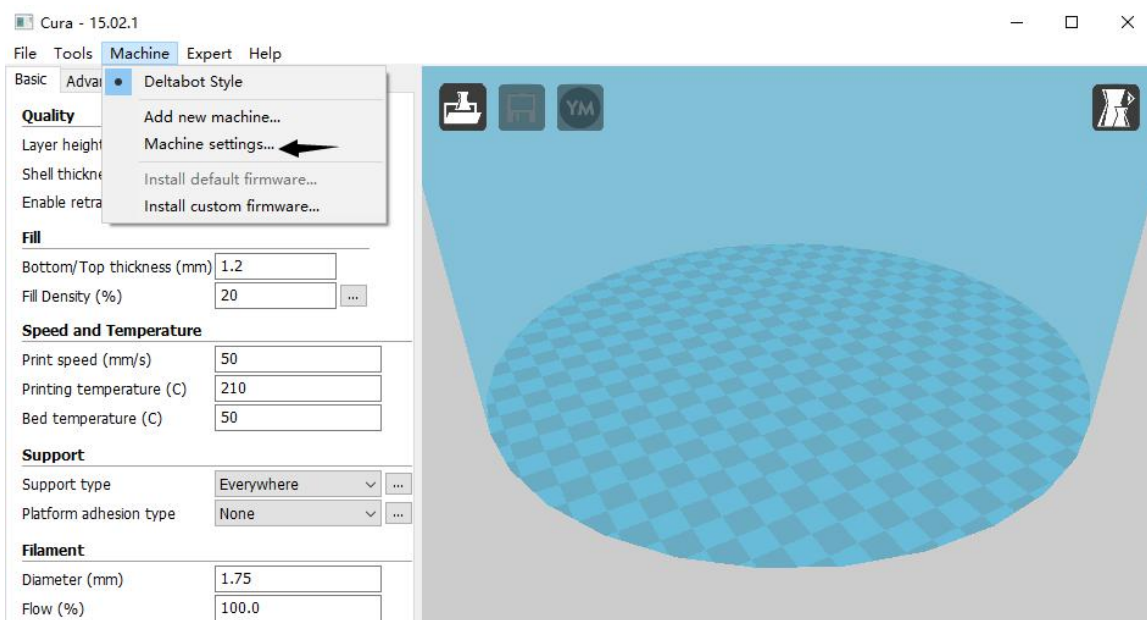


2) Select the language and click “Next”, select the kind of machine “Other(Ex: RepRap, Makerbot, Witbox)” and click “Next”, then choose “DeltaBot” and click “Next”, “Finsh”, and appears the main interface.





Click “ Machine” on the menu bar, and “machine settings”, change the maximum width and depth to 200, maximum height to 300, "Hot bed" need to be checked, machines have added hot bed. The serial port and baud rate can just be AUTO, or change serial port to your device, and baud rate to 250000.



Machine settings

Deltabot Style

**Machine settings**
E-Steps per 1mm filament 0
Maximum width (mm) 250
Maximum depth (mm) 250
Maximum height (mm) 400
Extruder count 1
Heated bed ☒
Machine center 0,0 ☒
Build area shape Circular
GCode Flavor RepRap (Marlin/Sprinter)

**Printer head size**
Head size towards X min (mm) 0.0
Head size towards Y min (mm) 0.0
Head size towards X max (mm) 0.0
Head size towards Y max (mm) 0.0
Printer gantry height (mm) 0.0

**Communication settings**
Serial port AUTO
Baudrate AUTO

Ok
Add new machine
Remove machine
Change machine name

Click “OK” to back to main interface, then we should change the parameters in option card “basic” as figure 6.21, it will show the meaning of every parameter when the mouse stays beside. Then switch to option card “Advanced”, as figure 6.22 shows, change nozzle size to 0.4, retraction speed to 40~60, distance to 3~4, and infill speed to 40. The other parameters can keep original or change them as figure 6.22.

Cura - 15.02.1

File Tools Machine Expert Help

Basic Advanced Plugins Start/End-GCode

**Machine**
Nozzle size (mm) 0.4

**Retraction**
Speed (mm/s) 60
Distance (mm) 8

**Quality**
Initial layer thickness (mm) 0.3
Initial layer line width (%) 100
Cut off object bottom (mm) 0.0
Dual extrusion overlap (mm) 0.15

**Speed**
Travel speed (mm/s) 80
Bottom layer speed (mm/s) 20
Infill speed (mm/s) 0.0
Top/bottom speed (mm/s) 0.0
Outer shell speed (mm/s) 0.0
Inner shell speed (mm/s) 0.0

**Cool**
Minimal layer time (sec) 5

Cura - 15.02.1

File Tools Machine Expert Help

Basic Advanced Plugins Start/End-GCode

**Quality**
Layer height (mm) 0.15
Shell thickness (mm) 1.2
Enable retraction ☒

**Fill**
Bottom/Top thickness (mm) 1.2
Fill Density (%) 20

**Speed and Temperature**
Print speed (mm/s) 50
Printing temperature (C) 210
Bed temperature (C) 50

**Support**
Support type Everywhere
Platform adhesion type None

**Filament**
Diameter (mm) 1.75
Flow (%) 100.0

Cura - 15.02.1

File Tools Machine Expert Help

Basic Advanced Plugins Start/End-GCode

**Machine**

Nozzle size (mm)

**Retraction**

Speed (mm/s)

Distance (mm)

**Quality**

Initial layer thickness (mm)

Initial layer line width (%)

Cut off object bottom (mm)

Dual extrusion overlap (mm)

**Speed**

Travel speed (mm/s)

Bottom layer speed (mm/s)

Infill speed (mm/s)

Top/bottom speed (mm/s)

Outer shell speed (mm/s)

Inner shell speed (mm/s)

**Cool**

Minimal layer time (sec)

Cura - 15.02.1

File Tools Machine Expert Help

Basic Advanced Plugins Start/End-GCode

**Quality**

Layer height (mm)

Shell thickness (mm)

Enable retraction ☒ ...

**Fill**

Bottom/Top thickness (mm)

Fill Density (%)  ...

**Speed and Temperature**

Print speed (mm/s)

Printing temperature (C)

Bed temperature (C)

**Support**

Support type  ...

Platform adhesion type  ...

**Filament**

Diameter (mm)

Flow (%)

Cura loads a robot model when you open it, you can print it directly. Click “File”-“Print”, and the printing dialog box appears, the click “print” to star to print. Or you can load the model file we provide, 20\*20\*10.stl (as figure 6.23), to print and measure the size to adjust the parameter.

