
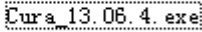
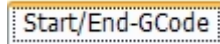



Cura slicing software instruction


Step 1 : Update the firmware of printer to V1.3 version


Step2: click   to install Cura software

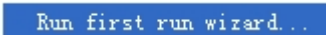
Step 3 : After installation,open cura software choose



, replace the  of cura by the content of Cura_start.txt

replace the  of cura by the content of Cura_end.txt

Step 4 choose the  in the cura menu ,in the sub-menu ,choose

 , will see the following window



choose “next” ,will come following window



Set according to the red pane,then click "next" to following window

RepRap information

RepRap machines are vastly different, and there is no default configuration in Cura for any of them.
If you like a default profile for your machine added, then make an issue on github.

You will have to manually install Marlin or Sprinter firmware.

Machine width (mm)	<input type="text" value="250"/>
Machine depth (mm)	<input type="text" value="150"/>
Machine height (mm)	<input type="text" value="160"/>
Nozzle size (mm)	<input type="text" value="0.4"/>
Heated bed	<input type="checkbox"/>
Bed center is 0,0,0 (RoStock)	<input type="checkbox"/>

Set as this picture,click"finish" to end it.

Step 5: basic setup

Basic item

- 1、Layer height (mm) | 0.27 layer thickness,set as you like
- 2、Platform adhesion type | Raft platform setup,choose Raft data
- 3、Fill Density (%) fill percentage setup
- 4、Diameter (mm) | 1.75 filament diameter setup.set as 1.75

Advanced item

- 1、Nozzle size (mm) | 0.4 nozzle diameter setup,set as 0.4