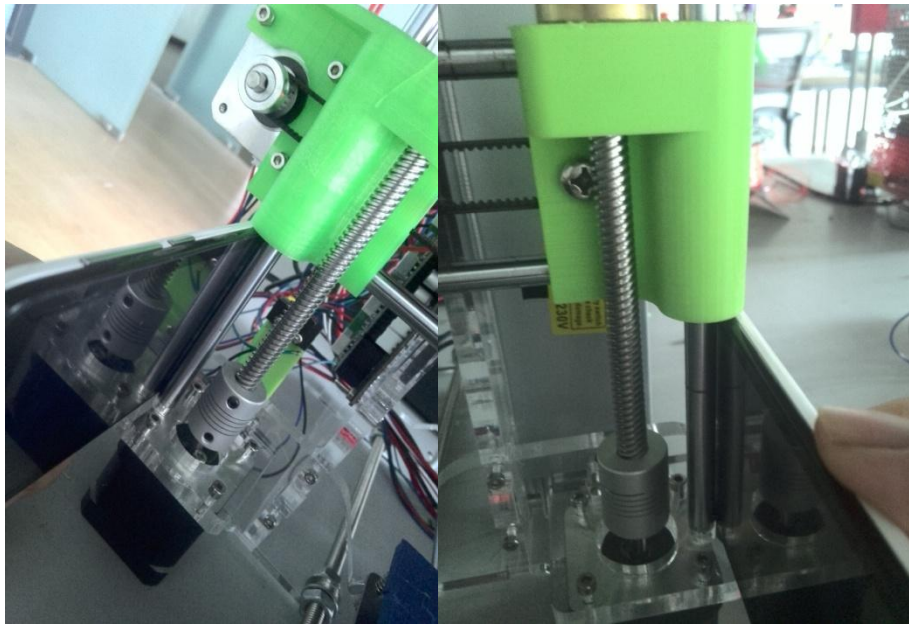


RAISCUBE 3D Printer User Manual

1. X-Motion Leveling.

Keep a same distance between top surface of Z-Axis Motor Fixed Plate and bottom face of Z-Axis Link Block (Left&Right). If not, turn a side of "Flexible Coupling" to keep two sides same to make it level.



2. Hotbed Level.

2.1 In the control panel, press control button and spin and press it to "Prepare" – "Z Home", then Extruder moves towards hotbed. Check if nozzle top almost touches hotbed surface. If not, find the long screw behind Left Z-Axis Link Block, and adjust it to be.

2.2 Panel Control, "Prepare" – "Auto Home", to move nozzle to origin point. And still check if the distance between nozzle top and hotbed surface. It should be only allow a piece of A4 paper to pass through. If not, make clockwise (or counterclockwise) rotation of the Wing Nuts beneath hotbed surface. Manually move hotbed or slide Extruder to other vertexes of hotbed to make it level evenly.

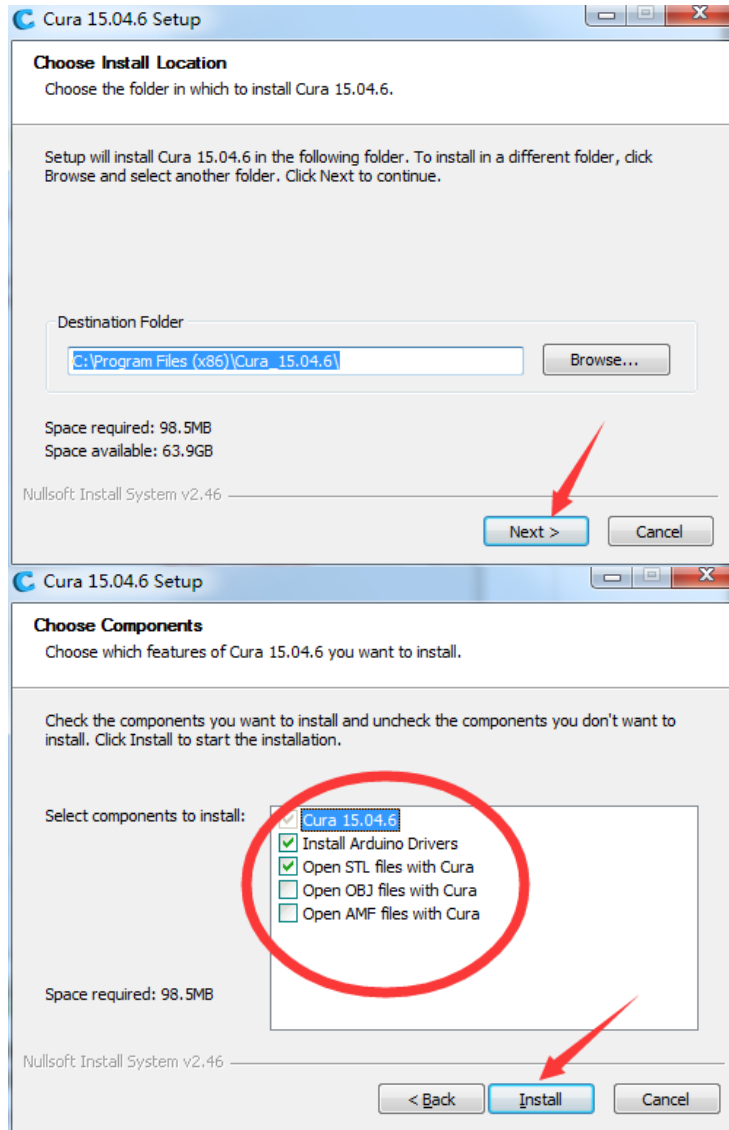
3. CURA 3D Slicing Software

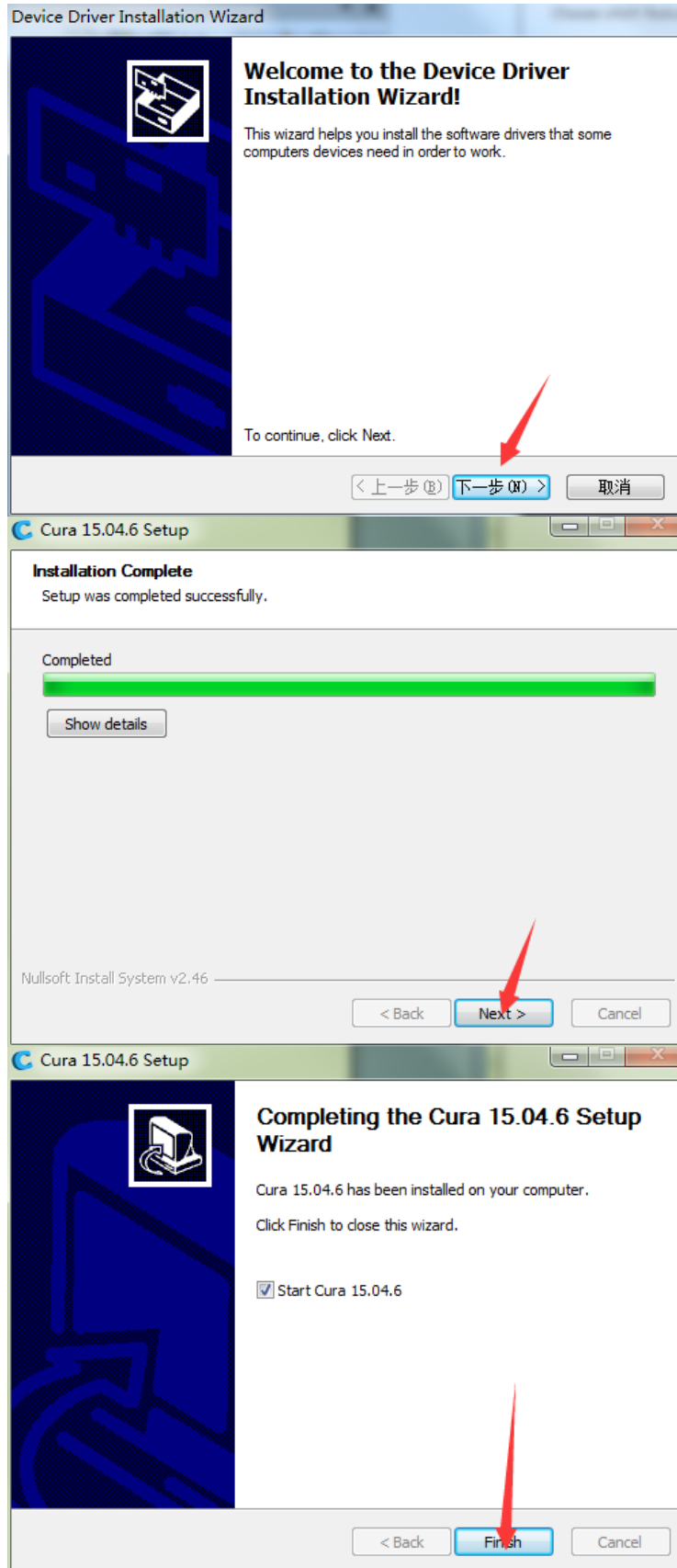
3.1 What can CURA help us?

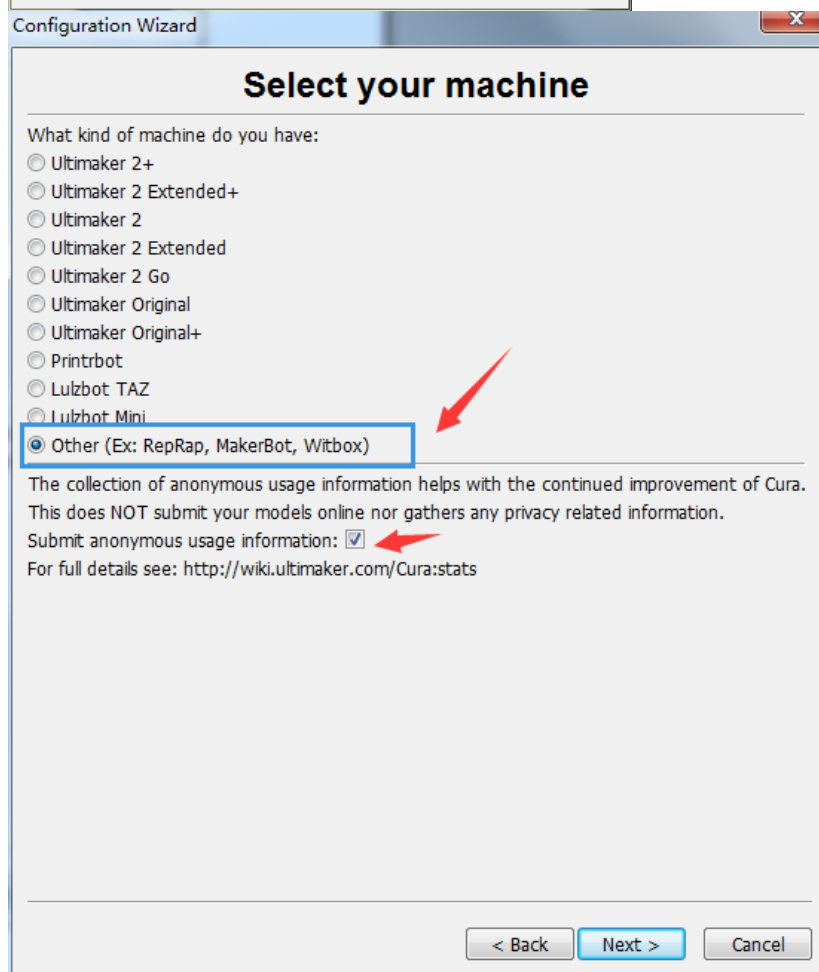
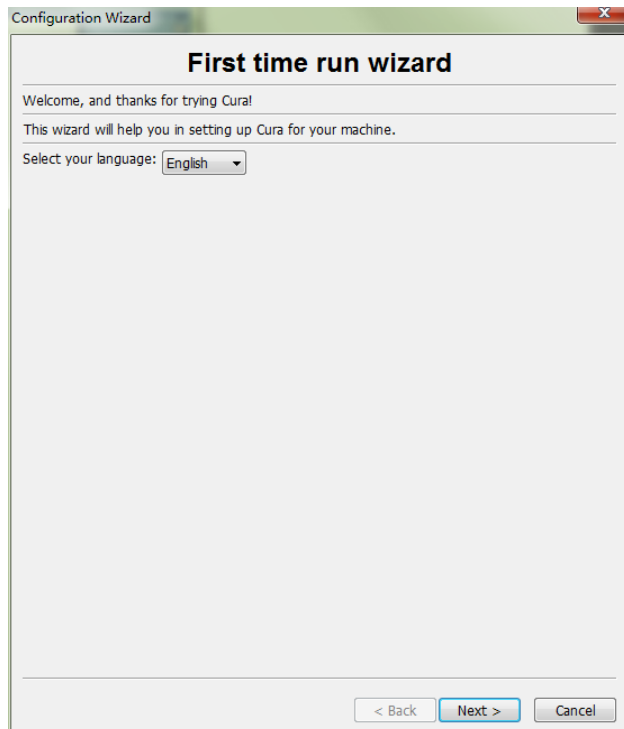
Cura is mainly used for model slicing. It calculates paths upon the model files, and then generates 3D model codes, named ".gcode". "STL" is the most common origin model files. These files can be save as ".stl" in a certain drawing software or download online.

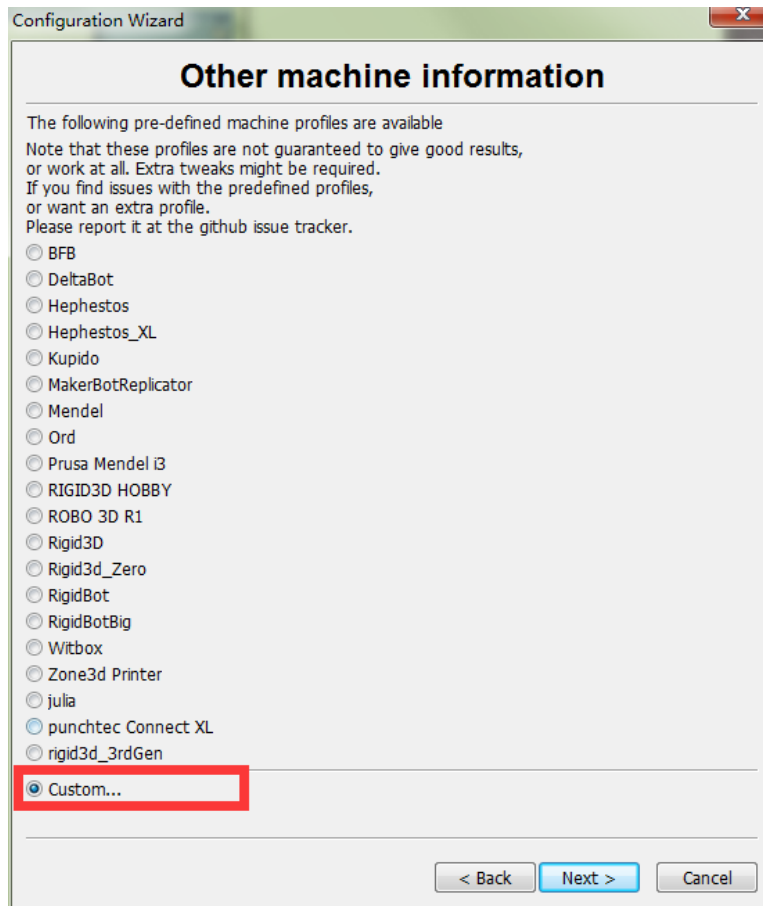
3.2 Installation of Cura.

Just like any other normal software installation. Find the "Cura.exe" in SD Card, and double click to cause the following window.









Configuration Wizard

Custom RepRap information

RepRap machines can be vastly different, so here you can set your own settings.
Be sure to review the default profile before running it on your machine.
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 name	RepRap
Machine width X (mm)	210
Machine depth Y (mm)	210
Machine height Z (mm)	225
Nozzle size (mm)	0.4
Heated bed	<input checked="" type="checkbox"/>
Bed center is 0,0,0 (RoStock)	<input type="checkbox"/>

RAISCUBE R2 Printing size is 210*210*225, and nozzle diameter is 0.4mm, with hotbed. So fill them with these parameters.

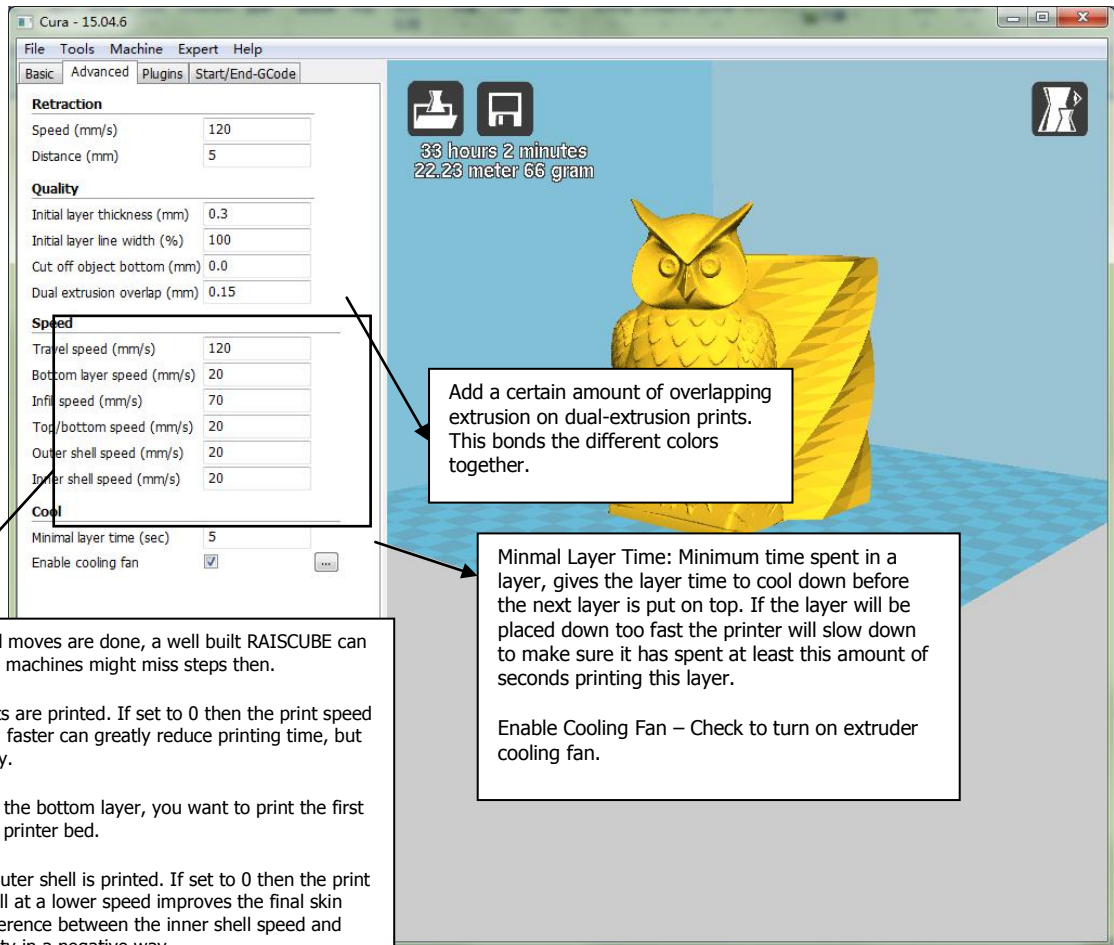
< Back Finish Cancel

Click "Finish" after configure the parameters as in pix, then Cura is installed successfully and show up.

1.4 Cura 3D Printing Slicing Configuration.

The image shows the Cura 15.04.6 software interface with several settings panels and callouts explaining their functions:

- Quality Panel:**
 - Layer height (mm):** 0.1. Callout: "Layer height" is the most important setting to determine the quality of your print. Normal quality prints are 0.2mm, high quality is 0.1 or 0.05mm.
 - Shell thickness (mm):** 0.8. Callout: Shell Thickness: Thickness of the outside shell in the horizontal direction. This is used in combination with the nozzle size to define the number of perimeter lines and the thickness of those perimeter lines.
 - Enable retraction:** . Callout: Retract the filament when the nozzle is moving over a none-printed area. Details about the retraction can be configured in the advanced tab.
- Fill Panel:**
 - Bottom/Top thickness (mm):** 0.6. Callout: This controls the thickness of the bottom and top layers, the amount of solid layers put down is calculated by the layer thickness and this value. Having this value a multiple of the layer.
 - Fill Density (%):** 20. Callout: This controls how densely filled the insides of your print will be. For a solid part use 100%, for an empty part use 0%. A value around 20% is usually enough. This won't affect the outside of the print and only adjusts how strong the part becomes.
- Speed and Temperature Panel:**
 - Print speed (mm/s):** 30. Callout: Speed at which printing happens. A well adjusted Ultimaker can reach 150mm/s, but for good quality prints you want to print slower. Printing speed depends on a lot of factors. So you will be experimenting with optimal settings for this.
 - Printing temperature (C):** 200. Callout: Printing Temperature used for printing. Set at 0 to pre-heat yourself. For PLA a value of 210°C is usually used. For ABS a value of 230°C or higher is required. Bed Temperature is used for the heated printer bed. Set at 0 to pre-heat yourself.
 - Bed temperature (C):** 50.
- Support Panel:**
 - Support type:** Touching buildplate. Callout: Type of support structure build. Touching buildplate is the most commonly used support setting. None does not do any support. Touching buildplate only creates support where the support structure will touch the build platform. Everywhere creates support even on top of parts of the model.
 - Platform adhesion type:** None.
- Filament Panel:**
 - Diameter (mm):** 1.75. Callout: Diameter – Filament Diameter (e.g 1.75mm); Flow--Flow compensation, the amount of material extruded is multiplied by this value.
 - Flow (%):** 100.0.
- Machine Panel:**
 - Nozzle size (mm):** 0.4. Callout: Nozzle Size(mm) – 0.4mm.
- Other Callouts:**
 - Open File.** (Icon)
 - Save file.** (Icon)
 - Est Printing Time.** 14 hours 55 minutes, 22.73 meter 68 gram.
 - Layer View.** (Icon)
 - Different options that help in preventing corners from lifting due to warping. Brim adds a single layer thick flat area around your object which is easy to cut off afterwards, and it is the recommended option. Raft adds a thick raster below the object and a thin interface between this and your object. (Note that enabling the brim or raft disables the skirt)**



Add a certain amount of overlapping extrusion on dual-extrusion prints. This bonds the different colors together.

Minimal Layer Time: Minimum time spent in a layer, gives the layer time to cool down before the next layer is put on top. If the layer will be placed down too fast the printer will slow down to make sure it has spent at least this amount of seconds printing this layer.

Enable Cooling Fan – Check to turn on extruder cooling fan.

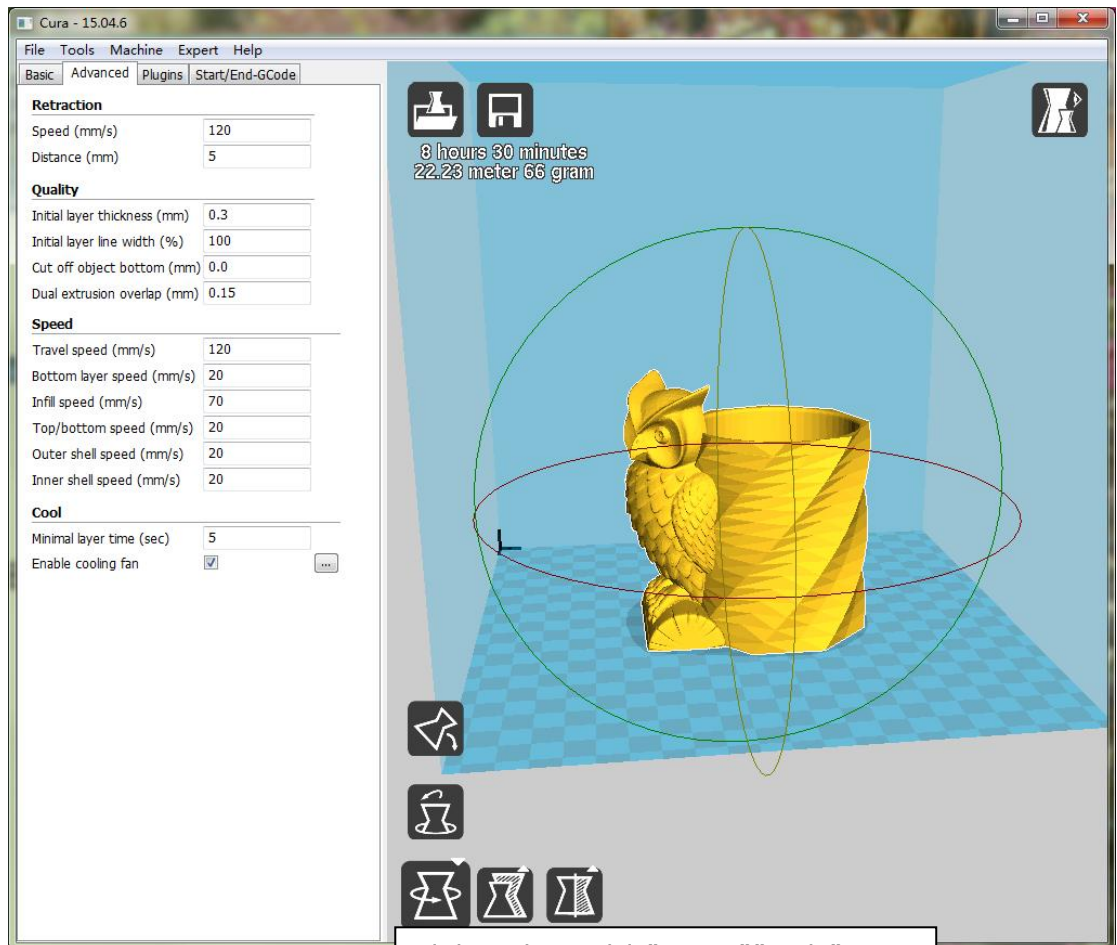
TravelSpeed -- Speed at which travel moves are done, a well built RAISCUBE can reach speeds of 250mm/s. But some machines might miss steps then.

InfillSpeed: Speed at which infill parts are printed. If set to 0 then the print speed is used for the infill. Printing the infill faster can greatly reduce printing time, but this can negatively affect print quality.

Bottom Layer Speed: Print speed for the bottom layer, you want to print the first layer slower so it sticks better to the printer bed.

Outer Shell Speed: Speed at which outer shell is printed. If set to 0 then the print speed is used. Printing the outer shell at a lower speed improves the final skin quality. However, having a large difference between the inner shell speed and the outer shell speed will effect quality in a negative way.

Inner Shell Speed: Speed at which inner shells are printed. If set to 0 then the print speed is used. Printing the inner shell faster then the outer shell will reduce printing time. It is good to set this somewhere in between the outer shell speed and the infill/printing speed.

“Rotate” “Scale” “Mirror”

Click on the model, “Rotate” “Scale” “Mirror” 3 buttons appear on left bottom coner.

Speed: Speed at which the filament is retracted, a higher retraction speed works better. But a very high retraction speed can lead to filament grinding. 120 is a common value for it.

Distance: Amount of retraction, set at 0 for no retraction at all. A value of 4.5mm seems to generate good results. Normally set to be 3~5mm.

Cut off object bottom: Sinks the object into the platform, this can be used for objects that do not have a flat bottom and thus create a too small first layer. Also can be used to printing

Retraction: Retract the filament when the nozzle is moving over a none-printed area. Details about the retraction can be configured in the advanced tab.

The screenshot shows the Cura 15.04.6 interface. On the left, the 'Advanced' tab is selected, showing settings for Retraction, Quality, Speed, and Cool. The Retraction section has Speed (mm/s) set to 120 and Distance (mm) set to 5. The Quality section includes Initial layer thickness (mm) at 0.3, Initial layer line width (%) at 100, Cut off object bottom (mm) at 0.0, and Dual extrusion overlap (mm) at 0.15. The Speed section lists Travel speed (mm/s) at 120, Bottom layer speed (mm/s) at 20, Infill speed (mm/s) at 70, Top/bottom speed (mm/s) at 20, Outer shell speed (mm/s) at 20, and Inner shell speed (mm/s) at 20. The Cool section has Minimal layer time (sec) at 5 and Enable cooling fan checked.

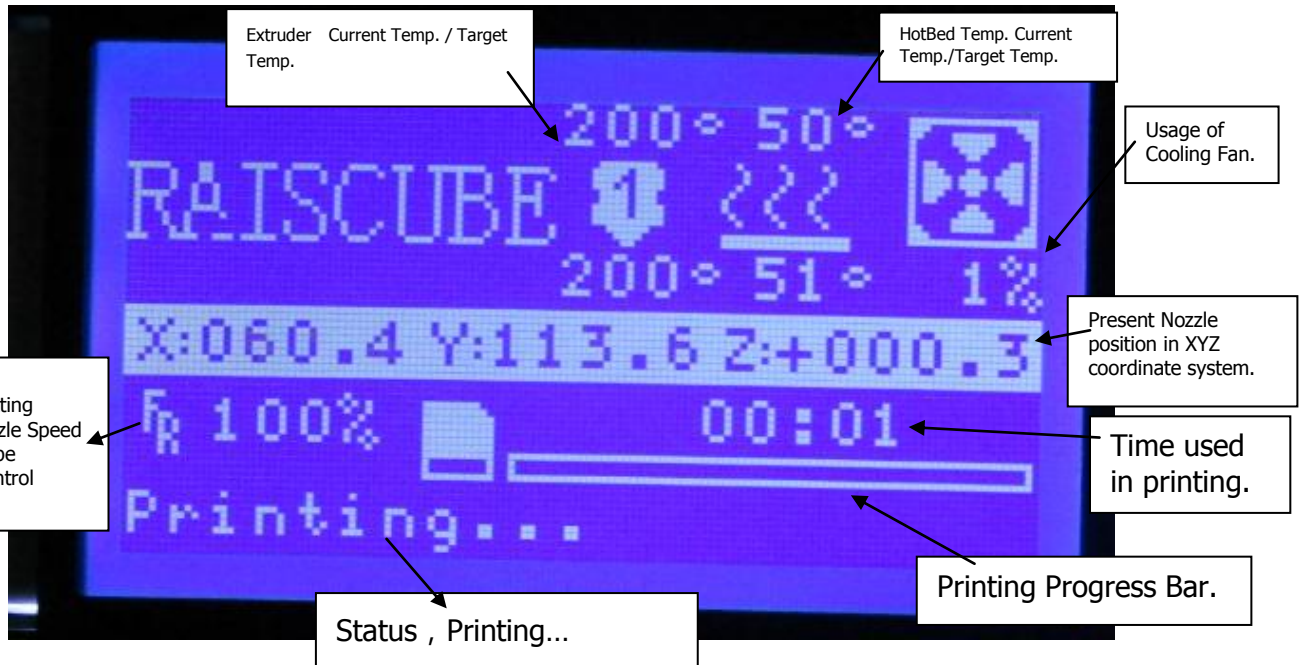
On the right, a 3D model of a yellow owl is shown on a blue checkered platform. Above the model, the dimensions are listed as W, D, H: 71.4, 93.7, 98.5 mm. Below the model, a scale panel shows Scale X, Y, and Z all set to 1.0, and Size X, Y, and Z in mm as 71.438, 93.716, and 98.479 respectively. A lock icon is present next to 'Uniform scale'. Above the model, a status bar indicates a print time of 8 hours 30 minutes, a length of 22.23 meter, and a weight of 66 gram.

At the bottom of the interface, there are three icons for scaling: a middle icon for adjusting model size, and two side icons for zooming. A text box explains that the middle icon is used to adjust model size for printing, and another text box notes that the model is locked by default to keep the whole object at the same zoom rate, with a lock icon to zoom XYZ separately.

After Configuration of Parameters, remember to save files to SD Card. Pleaes note to name file in English, or will fail to printing.

4. LCD Control Panel Menu

4.1 Screen Display Explanation.



4.2 Menu Map

