**LABISTS X1 Tutorial: Get started with Cura 15.04.6**

This tutorial will explain how to get the Cura slicer setting right and how to create a G-code file for your printer. It's no doubt that proper 3D slicer settings dramatically affect your LABISTS X1 3D printer's performance, and the quality of the 3D models it prints.

Cura slices 3D models. It translates the 3D STL, OBJ or 3MF file into G-code file that the printer can understand. You can download 3D model in the STL file format from online platform such as https://www.thingiverse.com/ or you designed with CAD software

Cura Tutorial - Step 1: Installing Cura

1.In order to use the Cura Software on your computer and get it to work with the LABISTS X1 Printer, You can simply click following link and download version 15.04.6 of Cura software or copy the Cura program from the SD card.
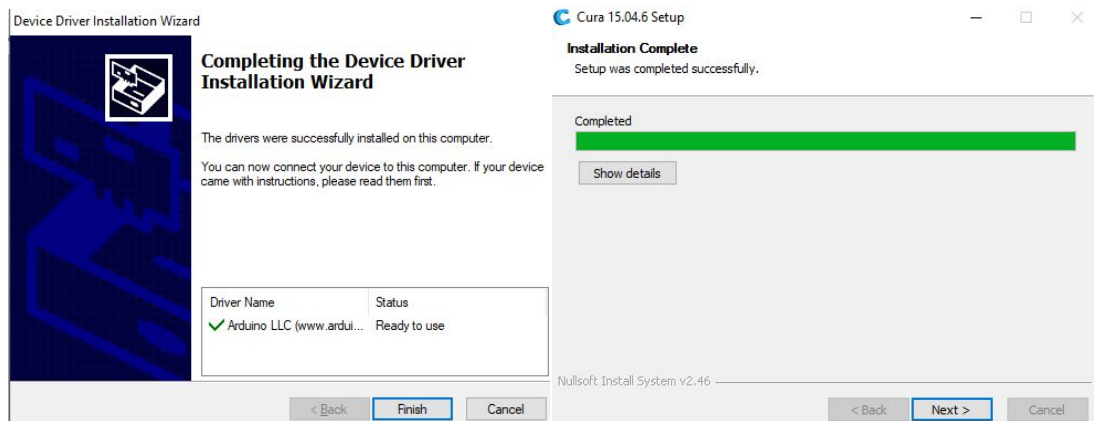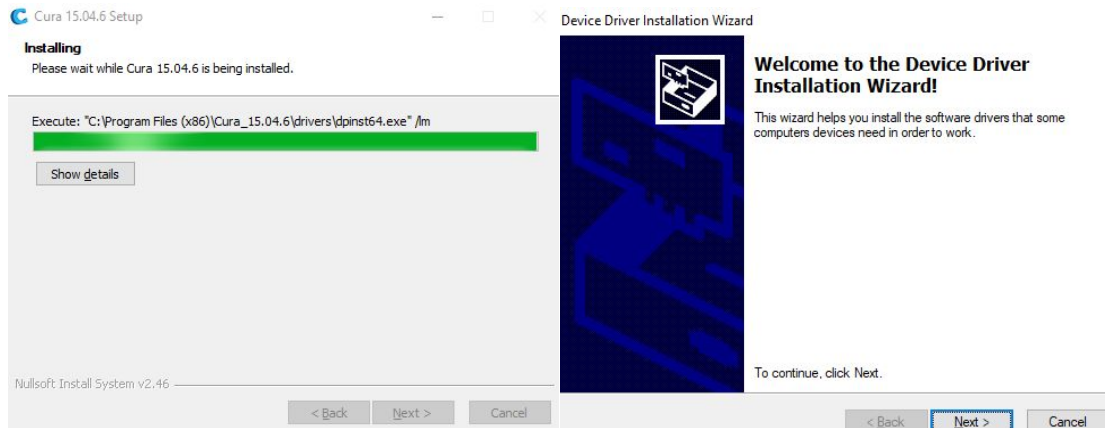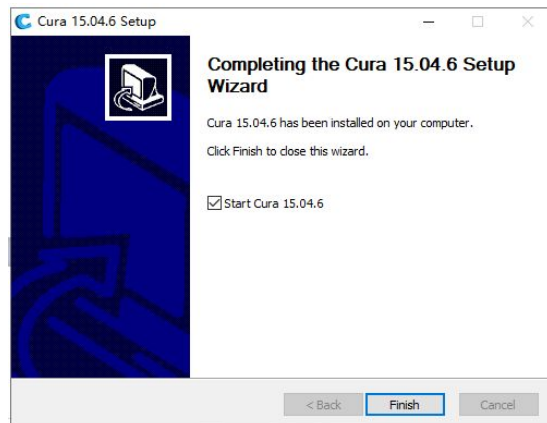
https://drive.google.com/open?id=16kgDBAJKtRGOcU8rKfLJTXp9PoBZuXxt
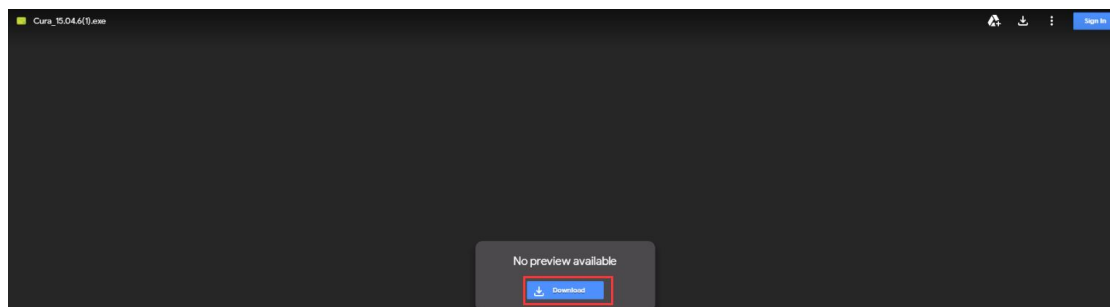
Note：DO NOT update the software after it is downloaded. This version works best with the X1 printer and newer versions will not work as smoothly.

2.Launch the installer. The setup wizard will pop up, click "Next" to begin the installation .
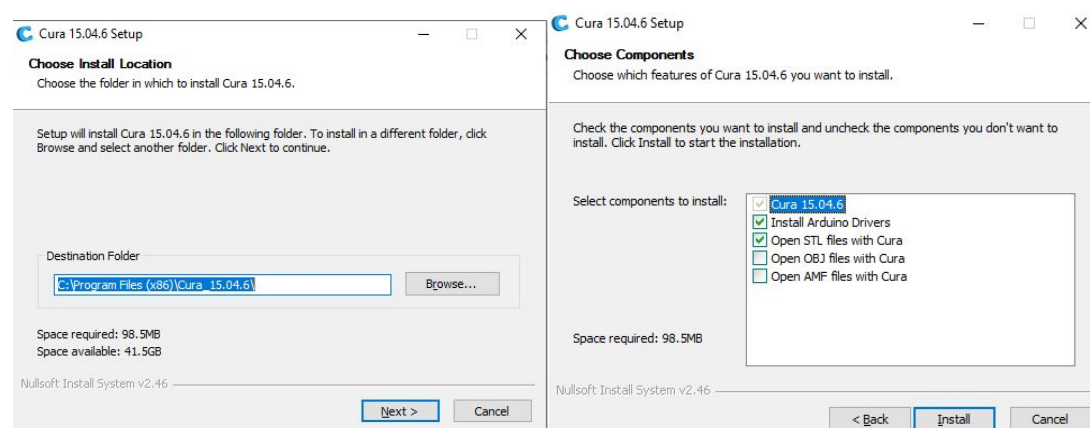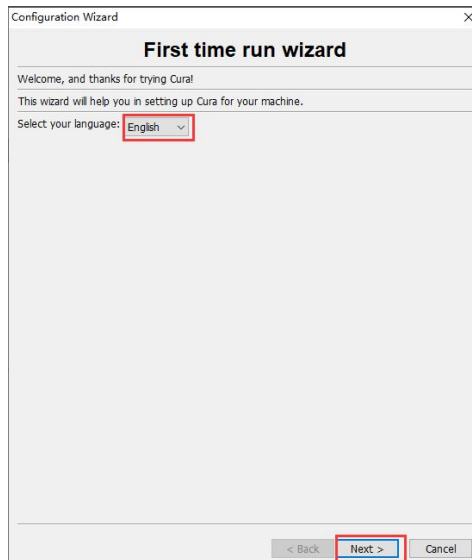
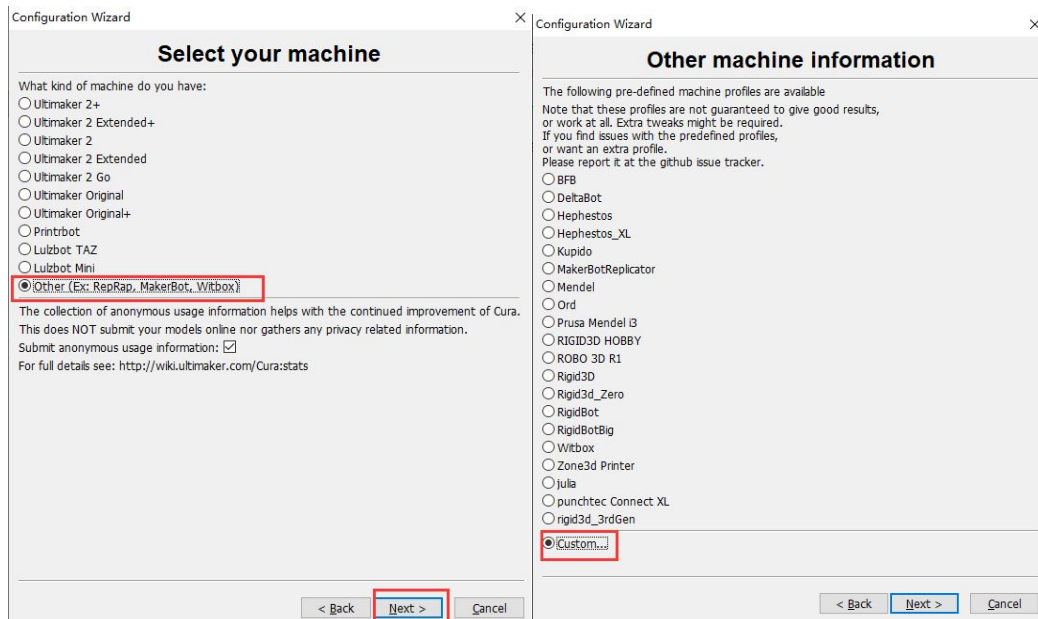**3. Click "Finish" and start Cura 15.04.6.**

Upon install or startup of the Cura software, you will see the window pictured below, which prompts you to update your Cura software – always click "No."

Cura Tutorial - Step2: Configuration Wizard

1. Select your language

2. Select your machine, choose "Other", then click "Next", follow by "Custom".

3. Add the setting for X1 printer. Enter and select the following values as the below picture showed, then click "Finish" .

Machine Name: X1

Machine width X(mm): 100

Machine width Y(mm): 100

Machine width Z(mm): 100

Nozzle size (mm): 1
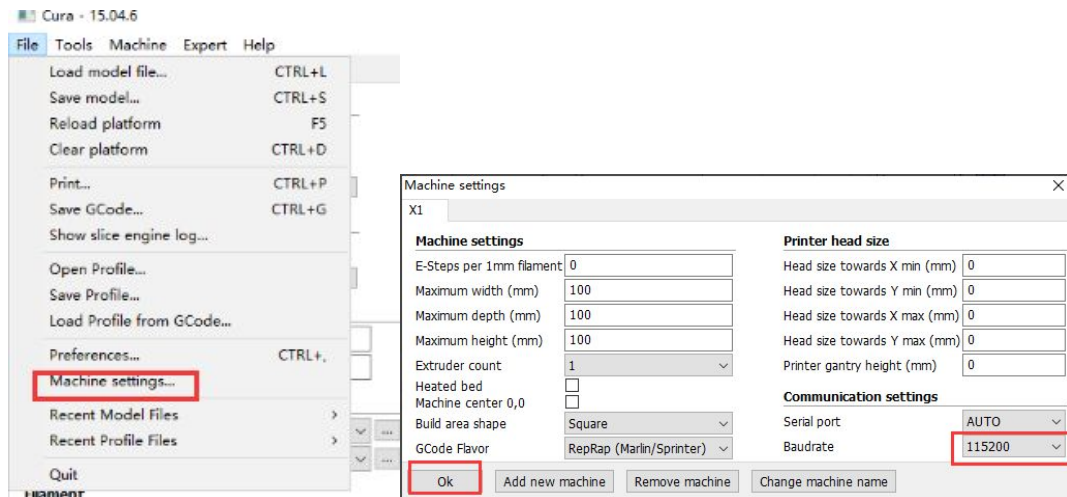
4. Congratulation! You have finished the installation!
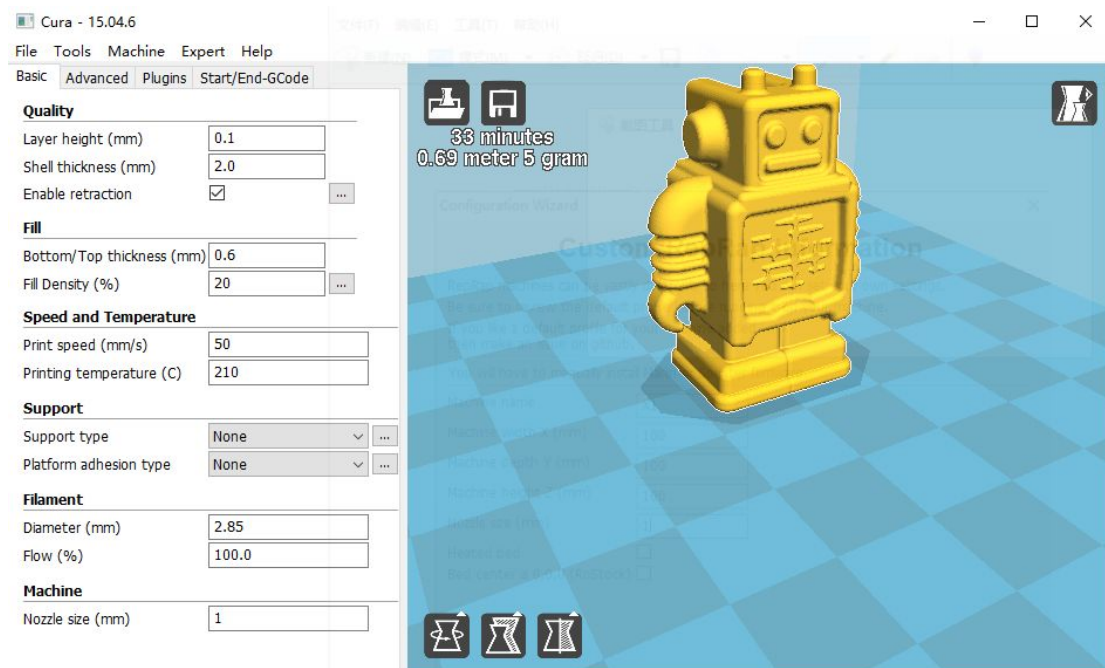
5. Machine Setting.   Go to File >Machine settings, change the port connection Baudrate to 115200, then click"OK".

Cura Tutorial - Step 3: Overview
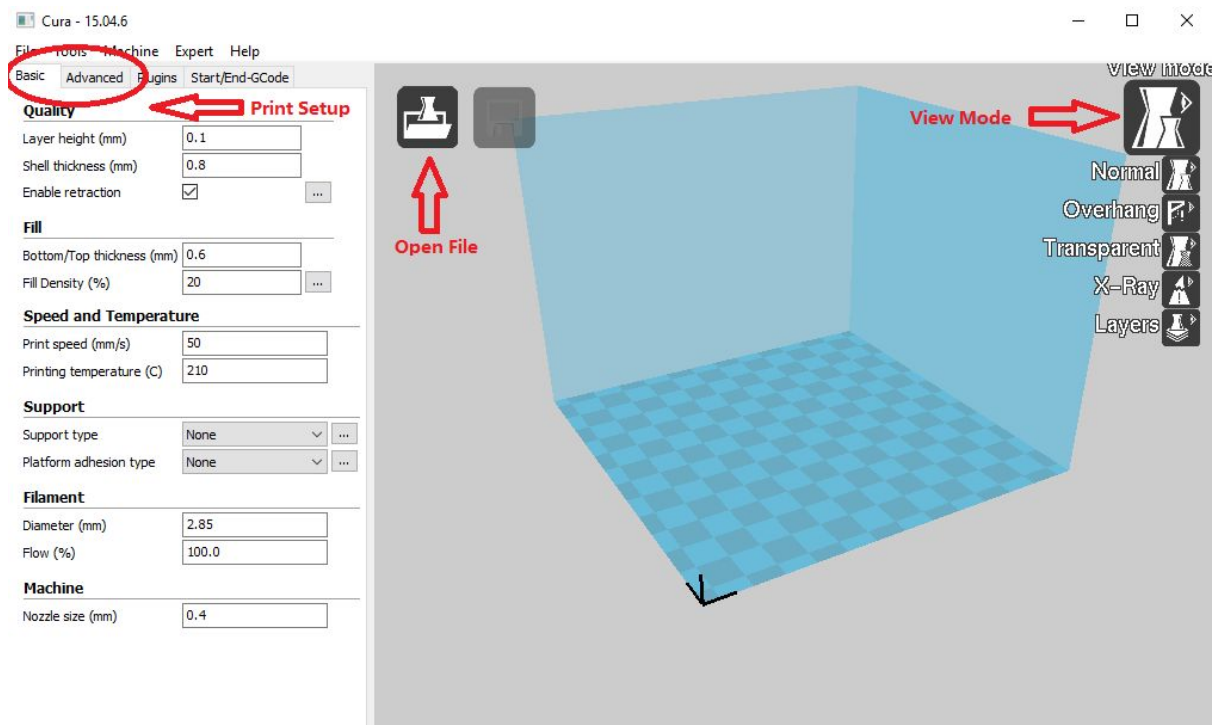
1. **Open file:** Use to open your STL or OBJ file, you can drag and drop the files here too.

2. **View Mode:** Let you switch between Layers and Normal view.

3. **Print Setup:** Printer-specific settings which user can access to all the settings (Basic Mode and Advanced Mode) and can be changed.

4. **Save to Disk:** When you are finished, save the G-code to your hard disk or SD card for the printer.
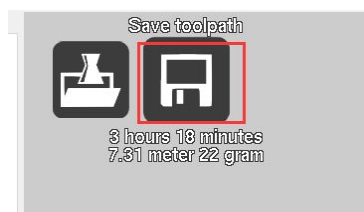
Cura Tutorial - Step 4: Handling 3D Model in Cura

1. Load a 3D model into Cura using the "Load" button **or clicking the File > Load model file.**

2. Explore different view mode.
In Normal View, you see the entire object (the way it will look when printed).
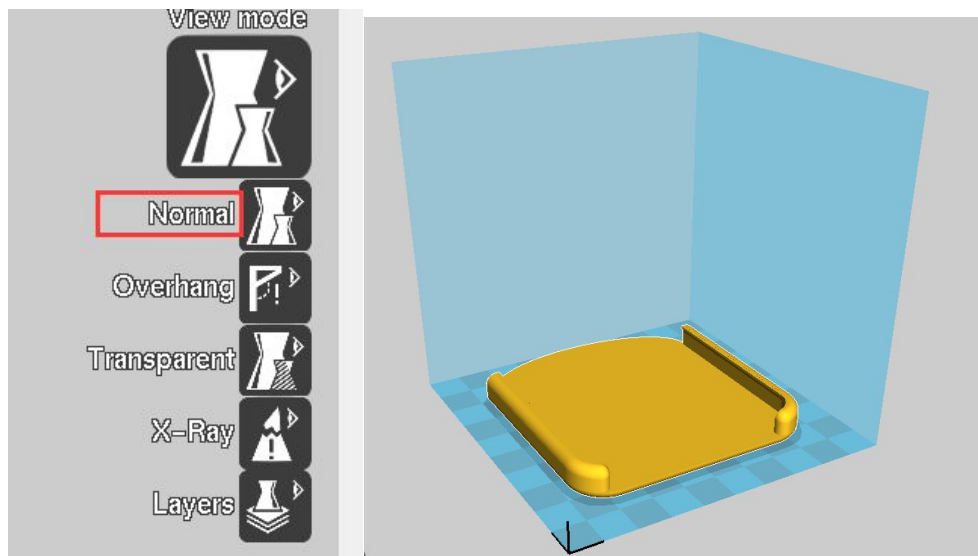
When you click the model, it will pop Rotate, Scale, Mirror icon. You can scale, rotate or mirror it on the build platform. Just play with these functions, you can undo the changes by clicking Reset button or clicking the icon again in Mirror option.

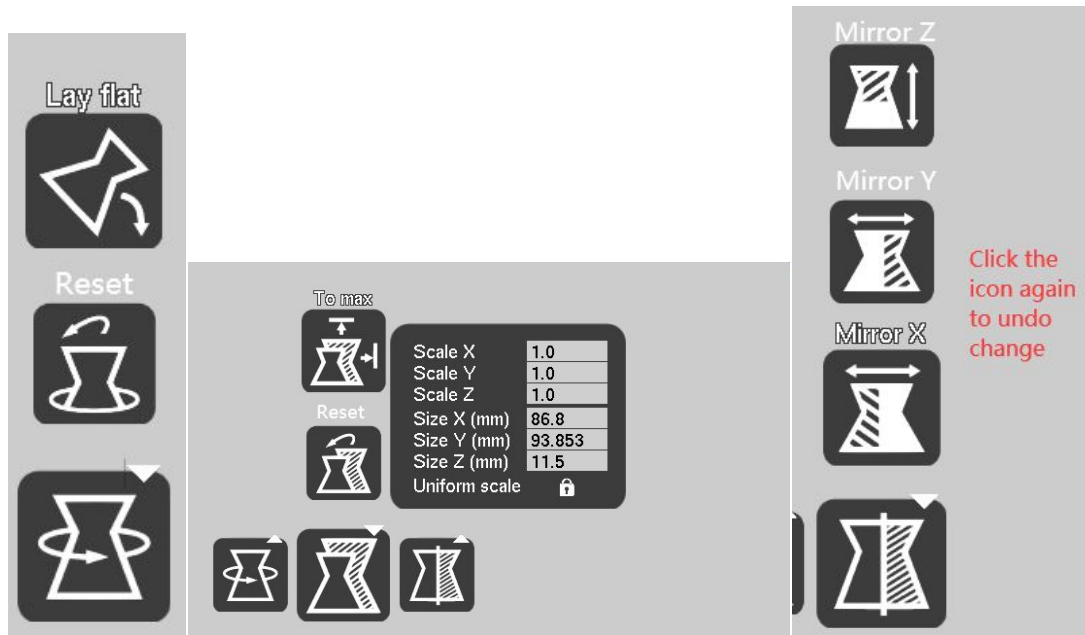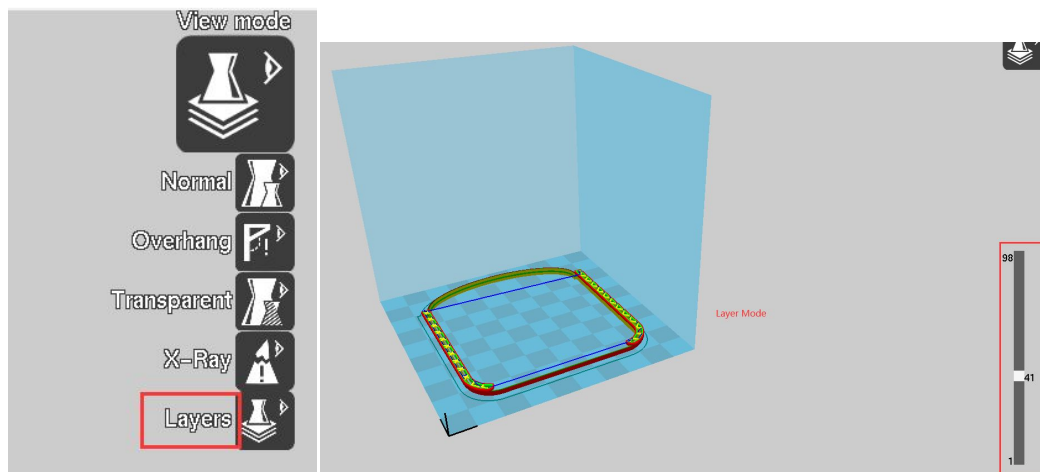In Layer View, you can go through layer by layer with the scrollbar at the bottom right. When switching to Layer View, it may take a short time before the layers are calculated and displayed (depending on the model and on your computer hardware).
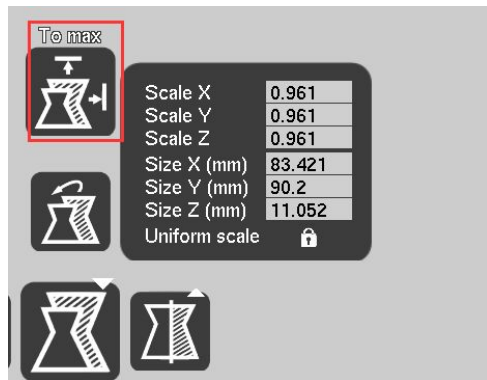

Attention

**If one of dimension is larger than the printing size of the printer (X,Y,Z=100mm), you can't transfer the STL file into G-code file . You need to click Scale button to change the scale.**

**You can scale the model to the appropriate dimension quickly by click the To max button **

3. Remember to keep the "Uniform Scaling" always locked, otherwise the printed model will be distorted. To close the Scale box, click the Scale button a second time.
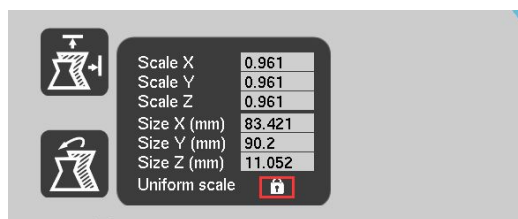
4. Right-click the model to open the context menu. Here, you can undo the changes to the model and center it on the platform again. You may also duplicate the object, if you wish to print several copies of the model.
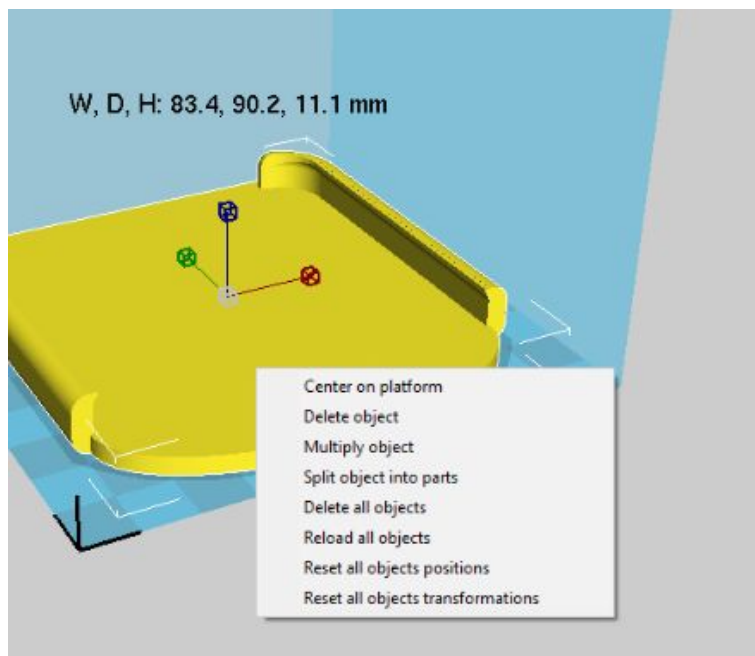
Cura Tutorial - Step 5: Cura Settings

For now, we work in Basic Mode. The values you can manipulate it to obtain the best printing quality:

- **Layer Height (recommended 0.1 or 0.2mm). The smaller the layer height value, the better**

printing quality - that will increase print time, however, probably result in failure of printing as well.

•The Shell Thickness setting (recommended 0.8mm) of Cura determines the thickness of the

object's wall. It has to be an integer multiple (1x, 2x, 3x, etc) of the nozzle diameter.   A thickness setting of 0.8mm means that the walls will be 2 lines wide (as the nozzle of X1 printer has a diameter of 0.4mm).

•The Bottom/Top Thickness (recommended 0.8mm). If you print an object with a large flat top,

you may want to print more layers in order to close the top surface completely. This avoids the unwanted "pillowing" effect.   (Again, the value must be integer multiples of the nozzle diameter).

•Infill Density(recommended 20%) determines how much plastic is printed inside the object. A

higher value means that more plastic will be printed. Typically, 10% to 20% are sufficient to build strong objects. In case, you wish to print the object completely hollow, set the density to 0%.
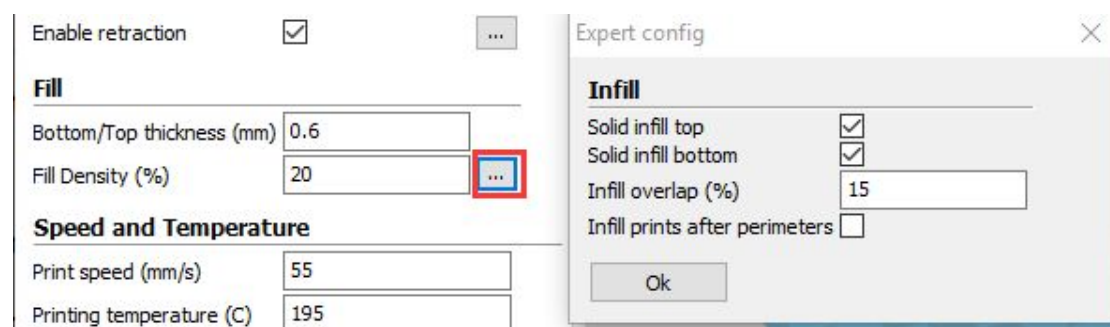
•Print Speed (recommended 10- 40mm/s). The larger the print speed value, the faster print

speeds.

•Printing Temperature (recommended 180 - 230°C). The larger the printing temperature value,

the higher the nozzle temperature.

•Select the "Support Type" option. Supports are needed when your model has overhanging

parts or parts floating in the air, if you select "None", overhanging part may collapse, and even failed to print. So, it is better to select "Touching buildplate" or "Everywhere". 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.

•Select the "Platform adhesion type" option(recommend Brim option). Different options that

help in preventing corners from lifting due to warping. If you select "None" for the model which contact areas between the bottom and the print platform is small, the edge will be lifted, even failed to print. Brim adds a single layer thick flat area around your object which is easy to cut off afterwards. Raft adds a thick raster below the object and a thin interface between this
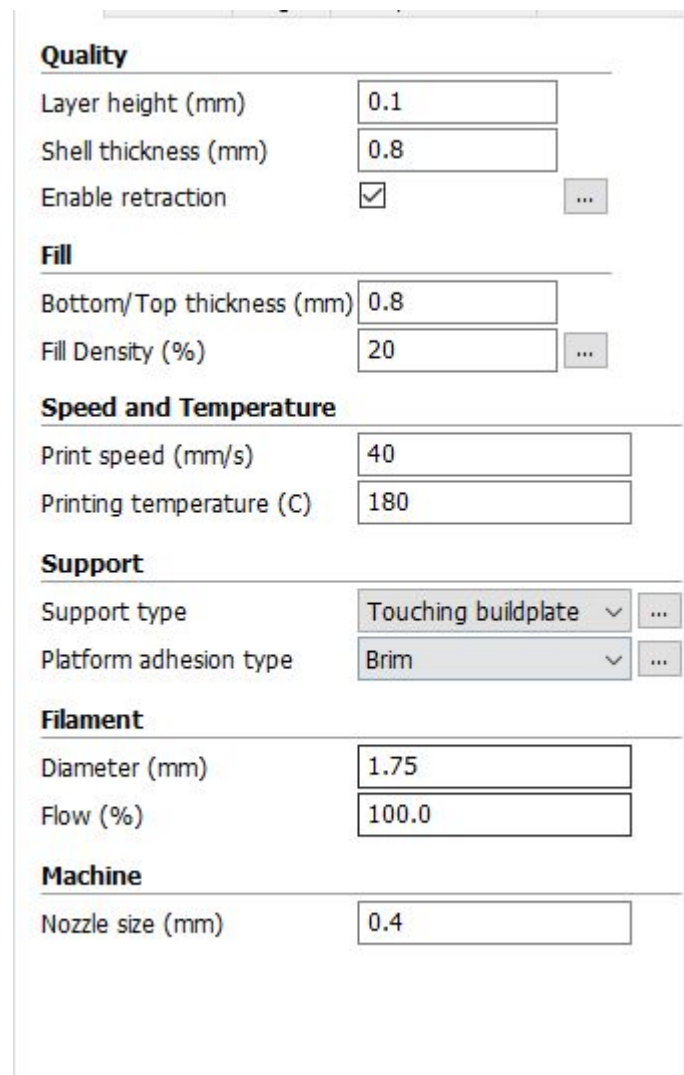
and your object.

•Diameter of the filament is 1,75mm

•Flow (%) of filament is the entire amount of the material that needs to be extruded for your

model. The Flow value is usually set to 100%, so the extruded amount equals the amount of material required. You only need to increase this setting if you use very soft materials. ATTENTION!

•The machine nozzle size is 0.4mm.

The below is the recommended X1 3D printer setting. (*Printer settings vary in shape, orientation and complexity of the 3D model.)

Quality

Layer height (mm)	0.1
Shell thickness (mm)	0.8
Enable retraction	☑ ...

Fill

Bottom/Top thickness (mm)	0.8
Fill Density (%)	20 ...

Speed and Temperature

Print speed (mm/s)	40
Printing temperature (C)	180

Support

Support type	Touching buildplate ˅ ...
Platform adhesion type	Brim ˅ ...

Filament

Diameter (mm)	1.75
Flow (%)	100.0

Machine

Nozzle size (mm)	0.4

Cura will calculates layer height, print duration and other settings according to the quality you selected.

Cura Tutorial - Step 6: Generate a G-code file

1.Click the Save to Disk button or select File > Save Gcode.
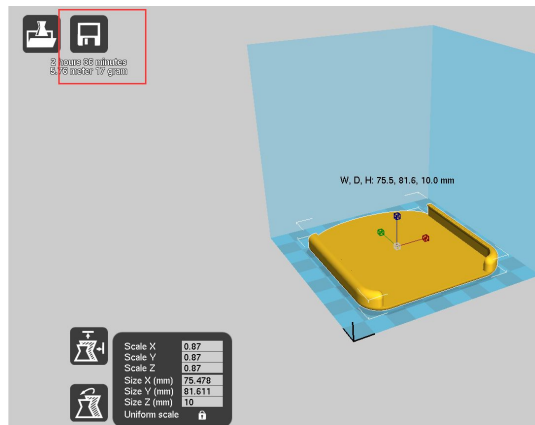
2.Enter a file name and the folder where the file is stored. You may store the file on the SD card you use to transfer the file to your 3D printer.

3.Reminder: Select Gcode File (*.gcode) and click Save.