# Cura:
Basic 3D Printing Guide

Introduction

Cura is an open-source slicer, developed by Ultimaker, for use with FFF (Fused Filament Fabrication) printers such as the PrintrBot Play. A slicer analyzes your 3D model and cuts it into a series of cross-sectional layers. It then writes this information to a text document of commands for the printer to follow, in a language called GCode.

This guide will provide you with the basic information you'll need in order to turn your 3D model into an instruction file for the Printrbot.

Printer Operation

MLibrary PrintrBot Play 3d printers are loaded with PLA plastic. PLA (polylactic acid) is a biodegradable polyester made from plant matter. You are responsible for generating your GCode. A 3d printing consultant will tend to the print server, part removal, and material needs.

User Interface
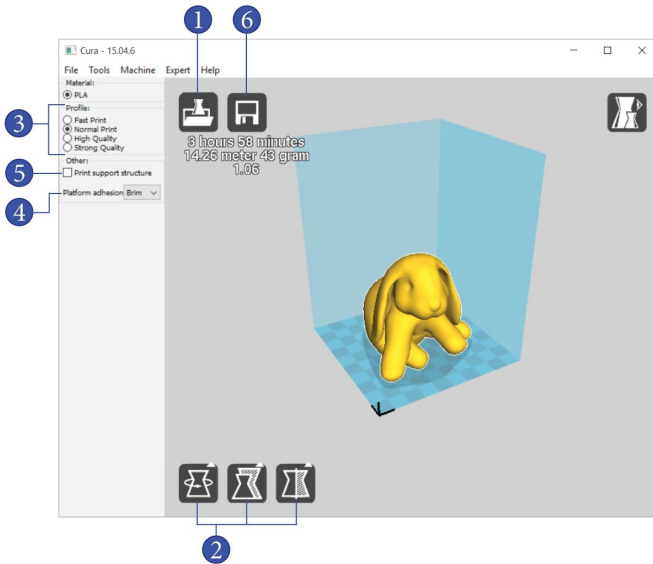
1. Load your model file; see "Load File"
2. Move, Scale, or Rotate your model; see "Transform"
3. Choose a print quality setting; see "Quality Settings" and "Examples"
4. Select platform adhesion; see "Platform Adhesion"
5. Select support structure; see "Support Structure"
6. Export your GCode file. If there is a loading bar beneath this button, wait until it disappears before clicking 'export'. See "Export"
7. Done! Put your GCode file on a flash drive and bring it to the PrintrBot station to have it printed.

Expert Modes

Cura 15 offers 2 modes of operation: QuickPrint, and Full Settings.

Quickprint is the recommended option as it offers quick simple and safe operation. Simply select your print quality and choose whether your 3d part requires support structure and or brim.

Full Settings offers more control over the 3d printer especially in the area of Supports. If you 3d model requires more advanced support structures first select a predefined profile in QuickPrint and bring it into Full Settings for adjustment. Be sure to work with a 3d printing consultant as incorrect settings can cause damage to the 3d printer.

Load File

Load your STL or OBJ 3D model file by either:

1. Dragging it into the build platform viewport
2. Clicking the "Load" button in the top left-hand corner of the viewport
3. Clicking "File" > "Load"

Once done, the 3D model should appear on the virtual build platform. If there is room in the build volume you may load multiple parts.

Transform

Once you have your model loaded, you may want to move, rotate, or scale it. The tools for this can be found in the lower left-hand corner of the viewport. The part(s) you want to transform must be selected first. Click on a part to select or deselect it.

When transforming your part, it's important to make sure it fits inside the cube that represents the printer's build volume. If the model is not fully inside this area, the part will turn grey, and Cura will not slice it.

Quality Settings

Fast Quality

This is the quickest of the presets, with a layer height of 0.3mm and a fill density of 5%. Choose this setting if your part doesn't have highly-detailed features, and/or if you don't want to wait several hours for your print to complete.

Normal Quality

This is the standard preset, with a layer height of 0.2mm and a fill density of 10%. This setting is a compromise between quality and speed. A "Normal" print will take about twice as long as a "Fast" print.
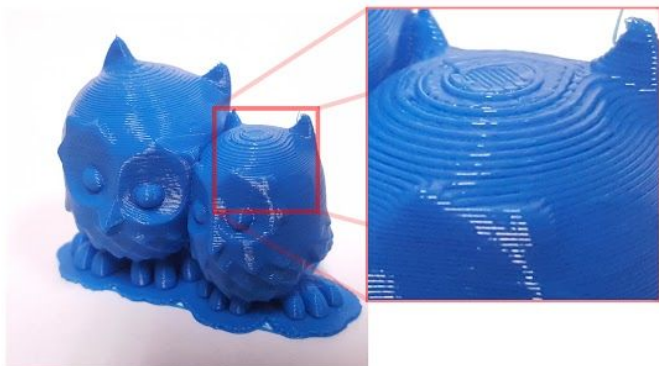
High Quality

This is the slowest of the presets, but gives the best quality result, with a layer height of 0.1mm and a fill density of 20%. A "High Quality" print will take about three times as long as a "Normal" print, and five to six times as long as a "Fast" print.
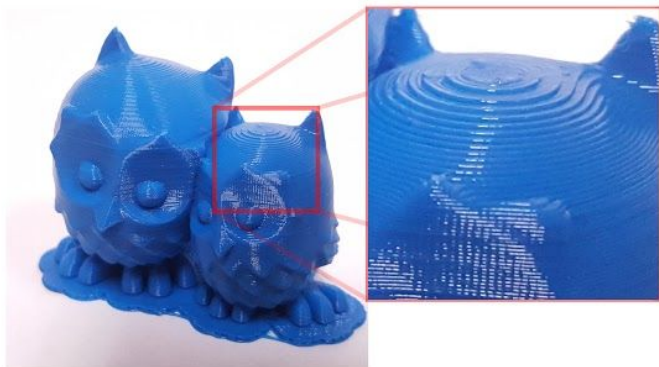
'Strong' Quality

This preset has the external appearance of a "Normal" print, with a layer height of 0.2mm, but has 60% infill, making for a far stronger and heavier (more dense) part. "Strong" prints will take a long time; about as long as a "High Quality" print.
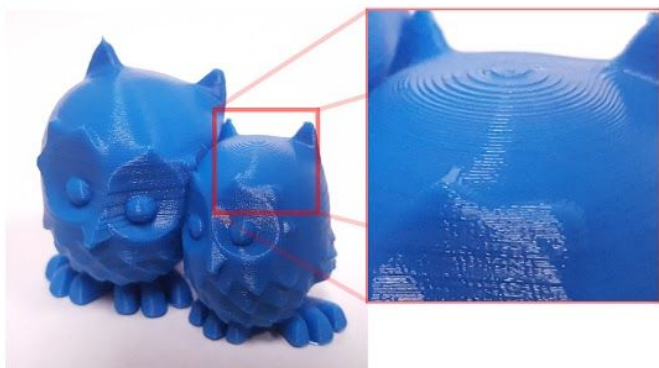
Fast Quality
0.3mm Layers

Normal Quality
&
'Strong' Quality
0.2mm Layers

High Quality
0.1mm Layers

Platform Adhesion

Platform adhesion tools are used to increase the surface area at the bottom layer of a part. This will help your part stick to the build platform and reduce warping.

A brim adds a single layer of easily removable material around the base of your part. Unless your part has sever overhangs this is the recommended option.

A raft adds a thick removable raster below the part. Rafts are recommended for parts with sever overhangs or parts that are very large as it will be easier to remove the part from the build tray.

Support Structure

If your model has overhangs that exceed 60 degrees, support structure may be required to ensure your part 3d prints correctly.

To activate support in QuickPrint check the box labeled "Print Support Structure" to have Cura automatically generate support structure beneath your model's overhangs.

The Full Setting mode allows for more support structure setting; None, Touching Build Plate, and Everywhere.

Touching Build Plate creates support structures that will contact the build plate.

Everywhere will create support structures that contact the build plate and top surfaces of the 3d model where needed. Everywhere is intended for parts with the most demanding overhangs.

Export

Cura slices 3d models in real time. Whenever you make a valid change to your 3d model a loading bar will appear in the upper left-hand corner of the viewport. When the loading bar completes Cura is ready to export your GCode.

To save your GCode click on the floppy disc icon at the upper left-hand corner, or go to "File" > "Save GCode". A dialog box will appear, prompting you to choose where to save the file. Save the file to a flash drive and bring it to the 3d printing consultant for review. If the part looks good and a 3d printer is available the consultant will send your part to print!