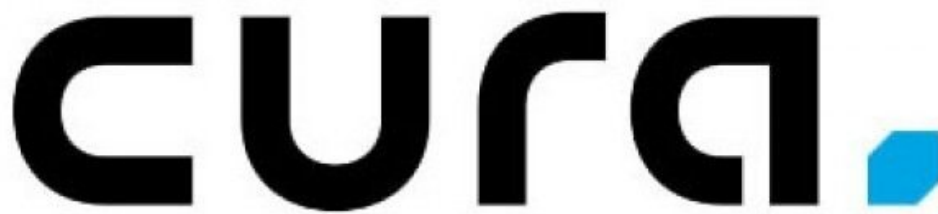




Ultimaker Cura: Getting Started

Learn how to install Ultimaker Cura and get started with slicing your first parts.

Written By: Chinmay Kandarp Sevak

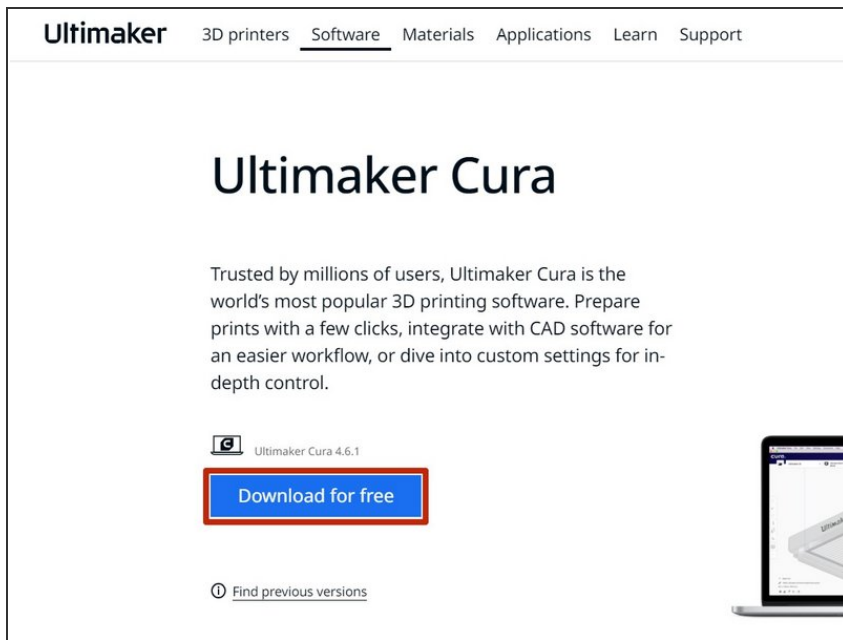


Introduction

Learn how to install Ultimaker Cura and get started with slicing your first parts. Ultimaker Cura can be used with a large variety of consumer 3D printers, but this guide uses the Ultimaker 2+ as an example. Learn more on the [Ultimaker website](#).

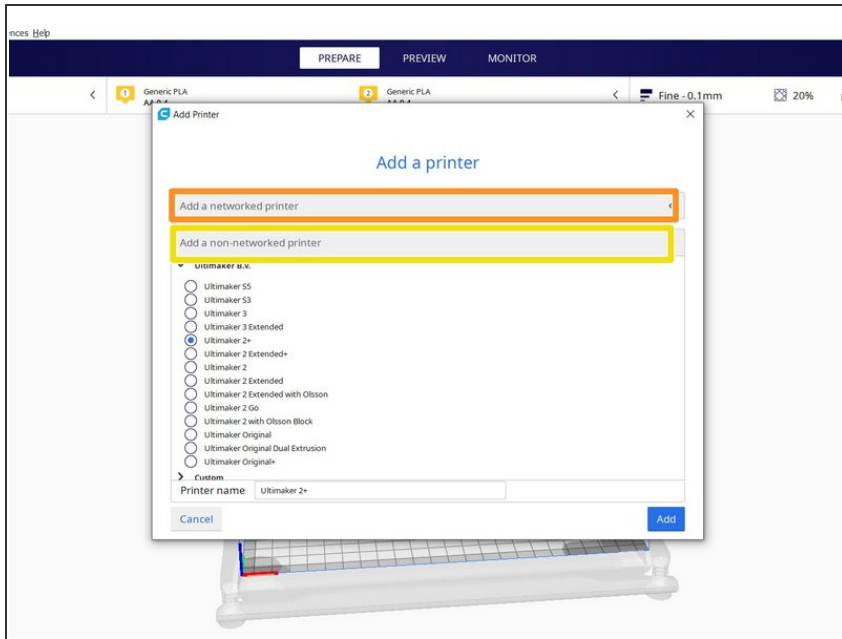
Images from: Kai Hirshon.

Step 1 — Download Ultimaker Cura



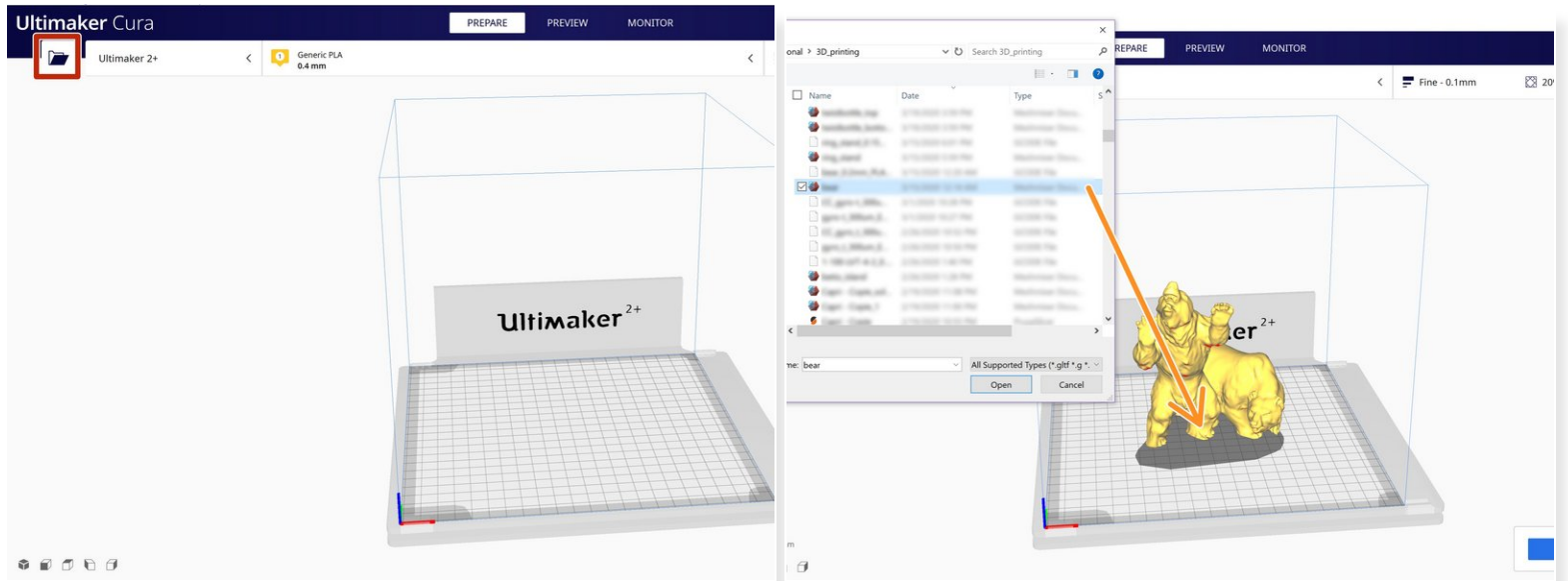
- Download the latest version of Ultimaker Cura from the [Ultimaker Cura webpage](#).
- ① Cura also comes in other flavors, such as Dremel Cura, so download whichever version is most appropriate for your application

Step 2 — Configuring Cura



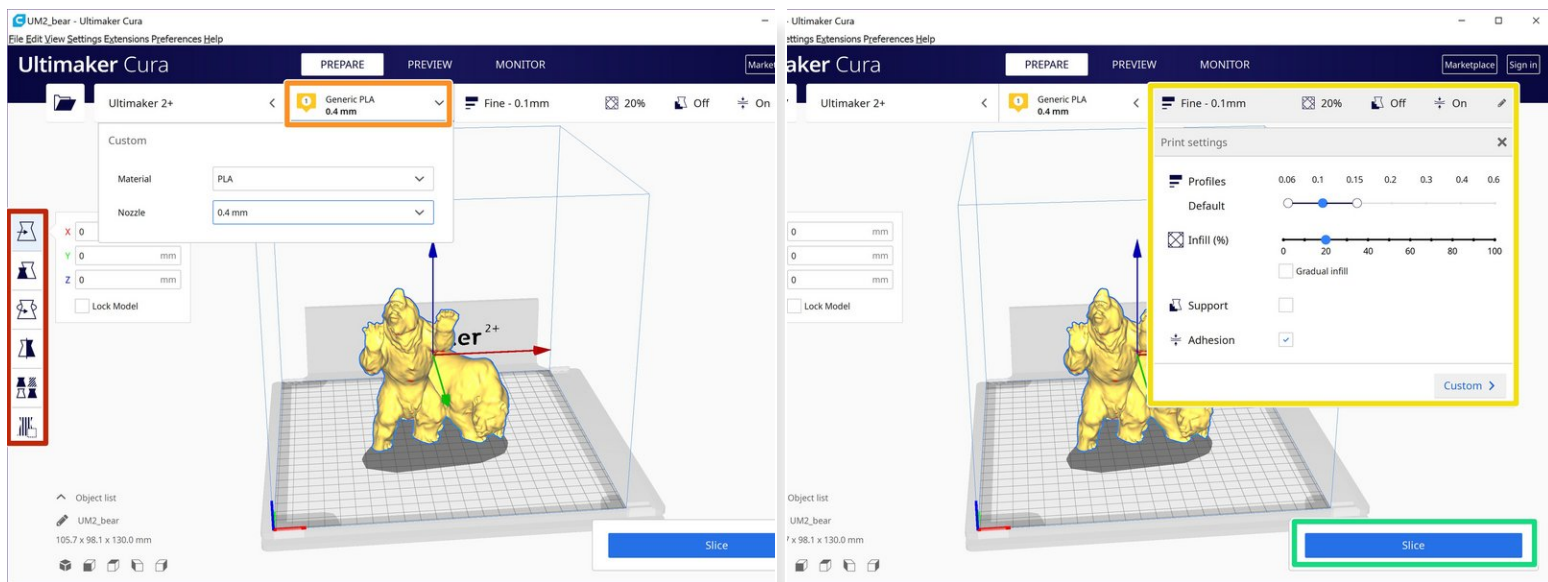
- Review then agree to the terms of use on the first page, then click next.
- Review the page that describes the changes from prior versions of Cura, then click next
- Open the appropriate menu to add a printer
 - If the printer is always connected to your network, then use **Add a networked printer**
 - For other printers, use **Add a non-networked printer**
- ① For this guide, we will be adding the Ultimaker 2+, which is not wifi-enabled

Step 3 — Importing the Model to Slice



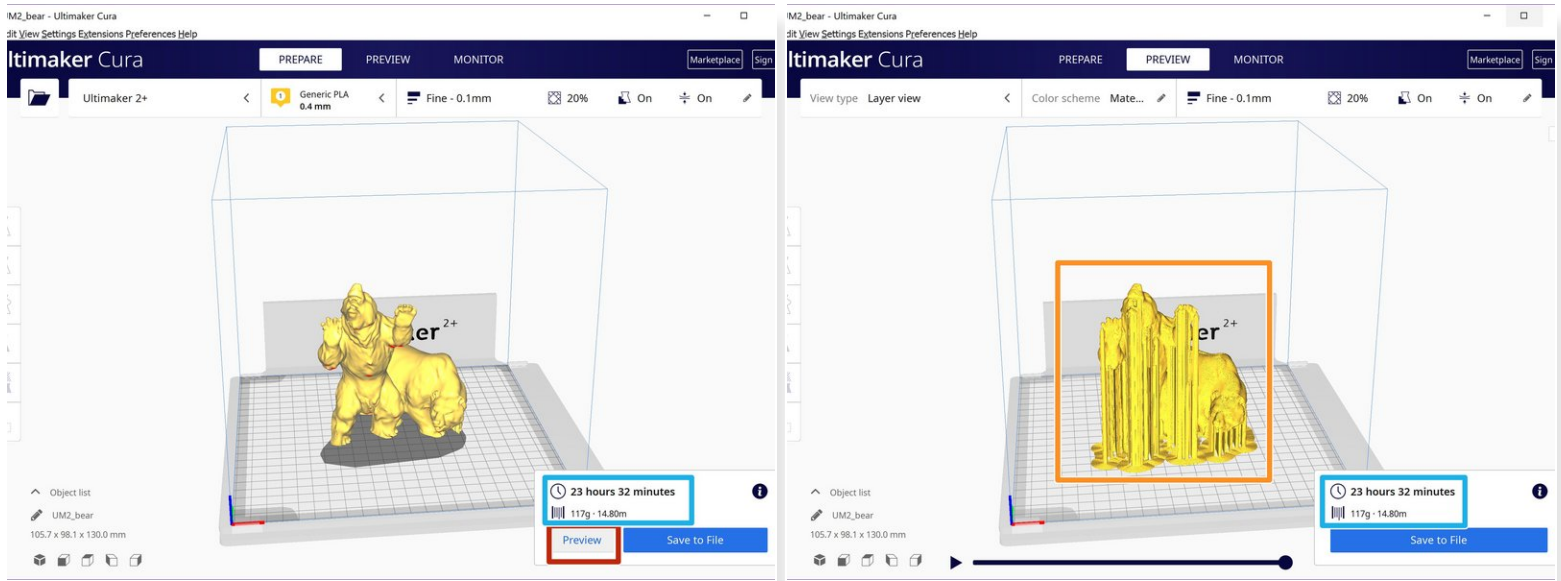
- There are two ways to import an STL in Cura:
 - Click on the file icon in the top left to open a file browser window
 - Drag and drop a file into the Cura window
- ❗ Multiple files can also be added to a single print by following the same process
- 📌 Be sure to check the imported units of the part to verify that it is the correct size

Step 4 — Preparing to Slice the Part



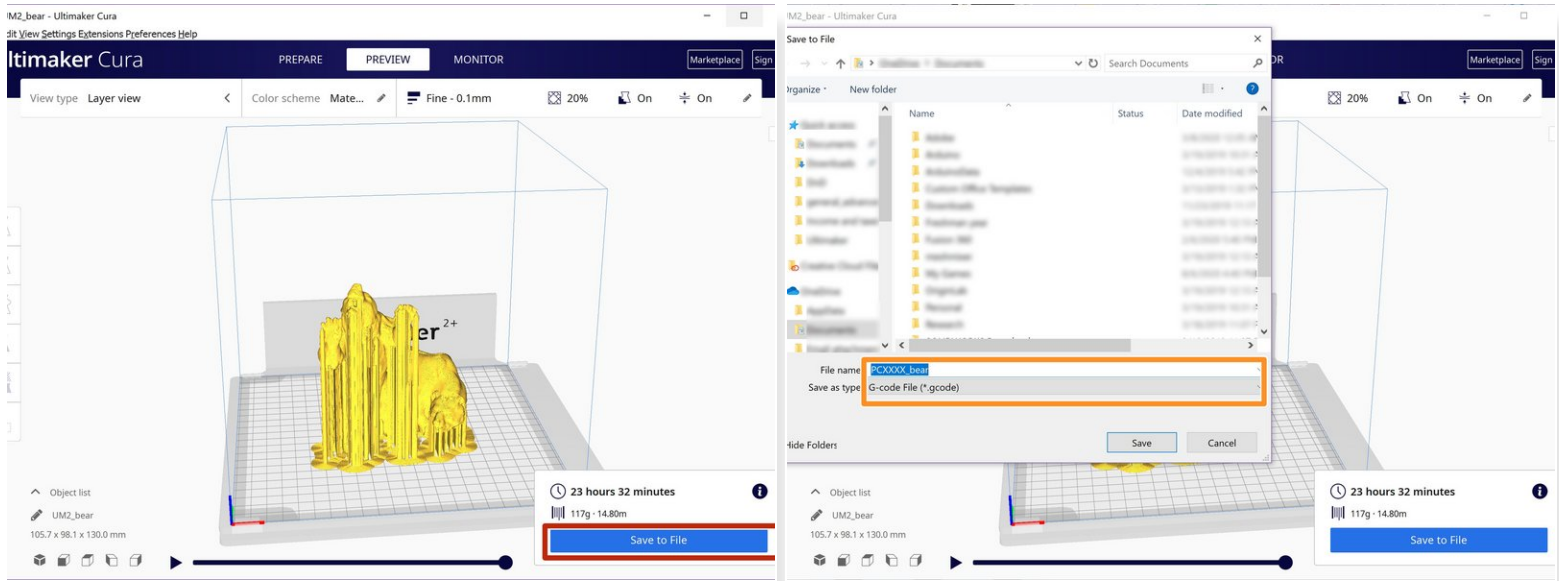
- The tools on the left enable manipulation of the file: move, scale, rotate, mirror or duplicate
- Select the material and nozzle size with which to print the part
 - ① The top section of the screen may be slightly different depending on which printer is configured, so it may be necessary to look for the desired settings
- Choose the default profile most appropriate to the part:
 - ① Most parts will be fine with a 'normal' print profile, which balances speed and quality
- Press the Slice button after choosing the correct profile and material

Step 5 — Print Preview



- After slicing, confirm the part looks as intended by clicking **Preview**
- Examine the preview to insure no error in placement of the automatic supports or build-plate adhesion aids
- Estimates of material used and time to print the part are also displayed here
 - If necessary, remember these values for keeping track of part cost or comparison against other print settings

Step 6 — Exporting the .gcode



- Click on the **Save to File** button if the print matches expectations
- Save the print to the desired location
- ❗ For parts that you may want to reference the settings used, it is also recommended to save the Cura project file
- ❗ By default, the file title will be the name of the first file imported to Cura for that session
 - Change the title if necessary for your desired workflow