

Cura 2.3



Quick start guide

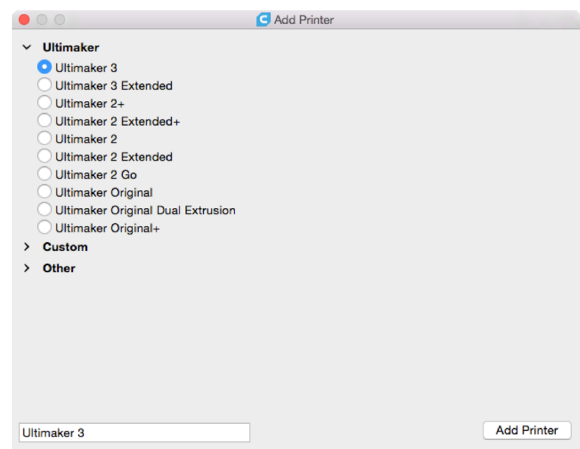
Initial Setup

When using Cura for the first time, you will automatically enter the **Add Printer Wizard**. To automatically configure Cura, select your printer from the list.

If your printer does not appear, you can create your own profile by selecting **Custom FDM Printer**. More information on custom FDM printers can be found [here](#).

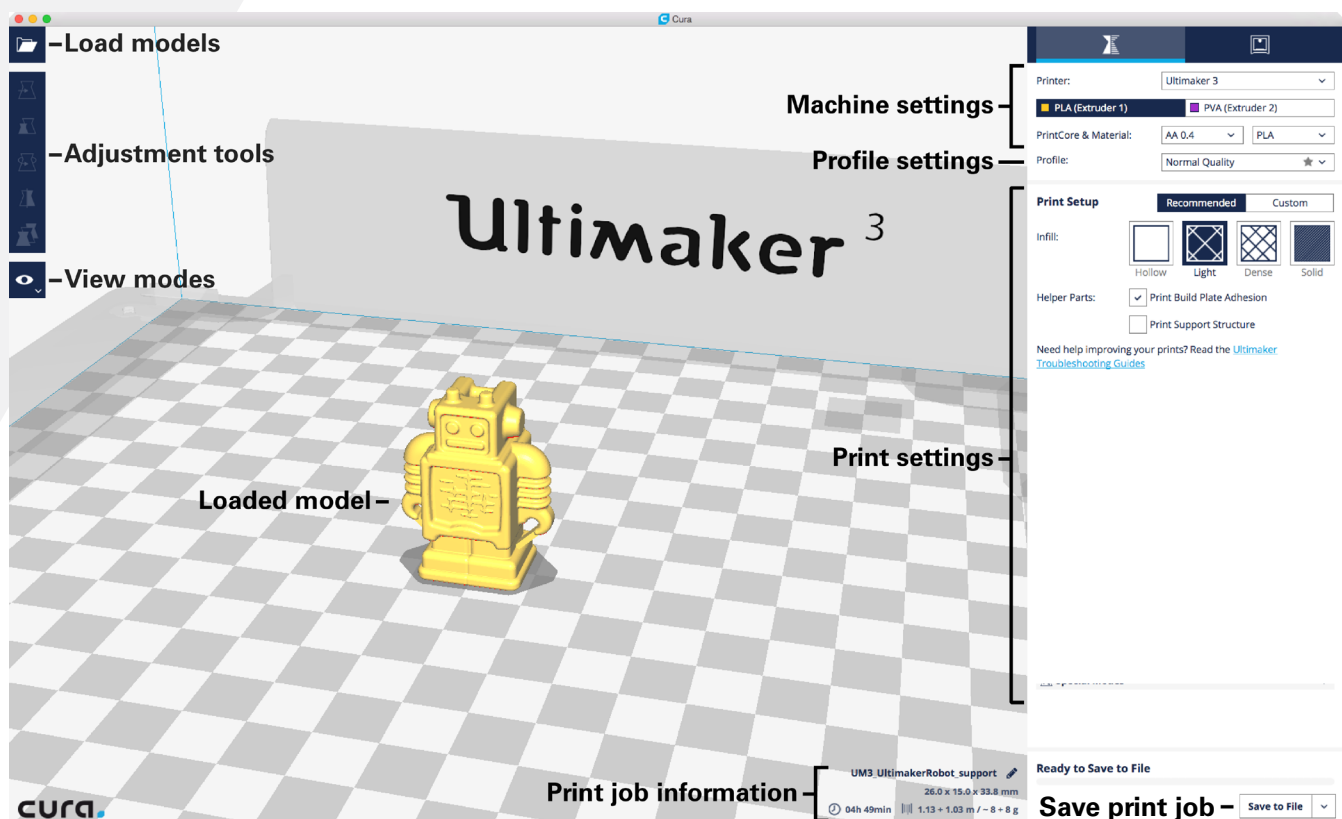
To add, edit or select a different printer, simply return to the **Add Printer Wizard**.

Go to Settings > Printer > Add Printer



Interface

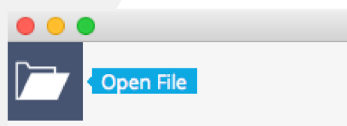
After you have added your first printer, the main interface of Cura will become visible. Here is an overview of the interface.



Start printing in a few simple steps

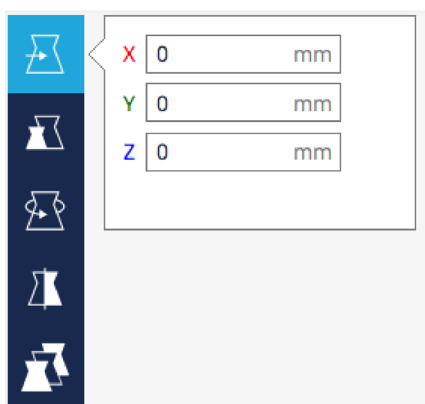
1. Load model

Firstly, load one or multiple models by clicking the **Load Models** button at the top right corner of the screen. Cura supports all major 3D file types: stl, obj, x3d and 3mf.



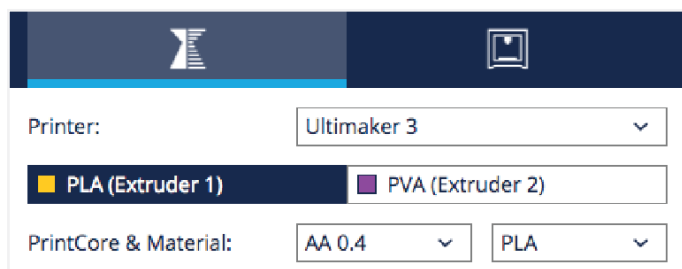
2. Adjust the model

Left-click the model to highlight and use the **Adjustment Tools**. These tools can be used to position, scale and rotate the object. The following tools are available: **Move**, **Scale**, **Rotate** and **Mirror**. For more information on the **Adjustment Tools**, visit [this page](#).



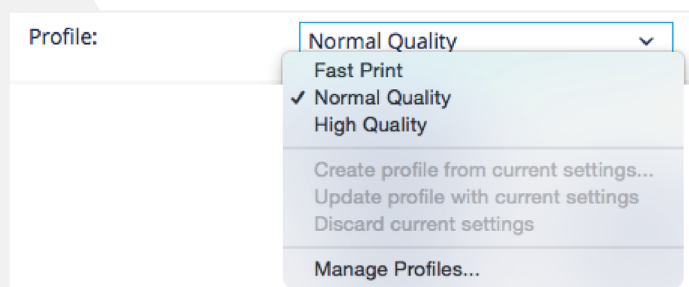
3. Check printer settings

Printer Settings are displayed in the top right corner of the sidebar. Depending on the printer, you can set **Nozzle Size**, **Print Cores** and **Materials**. Make sure these settings match your physical setup.



4. Choose profile quality

Cura comes with preloaded **Printing Profiles** that are tested throughout, to create the perfect print every time.







5. Recommended mode, helper options

Recommended mode is ideal for those using Cura for the first time.

It has 3 simple options to complete the **Printing Profile**:

- **Infill density**, to choose the model strength
- **Build plate adhesion**, to prevent warping
- **Support structure**, to print bridges and sharp overhangs

Infill:



Hollow Light Dense Solid

Helper Parts:

☒ Print Build Plate Adhesion

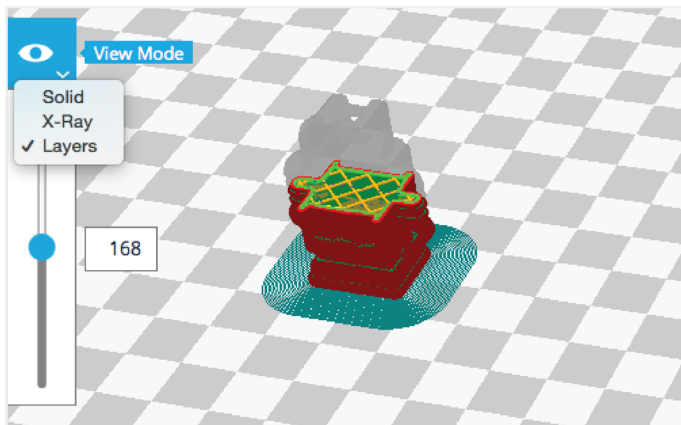
☐ Print Support Structure

6. Preview

Wait for Cura to finish slicing the object, which is visible in the bottom right corner of the screen. All print settings are set and you can now preview the result. Click on the **View Mode** icon and select Layers.


The **Layer Number box** can be used to jump to a specific layer. To go one layer up / down at a time, select the box and use the arrow keys on the keyboard.

The image below shows the result of the slice. All colors represent a certain model part, making it easier to determine the proper printing strategy. More information about the layers and their colors can be found [here](#).





7. Print

When you are happy with the preview, save the file or print directly from Cura. Click **Save To File**, **Save to Removable Disk** or **Print Over Network** in the bottom right corner of the screen.

UM2_UltimakerRobot 

26.0 x 15.0 x 33.8 mm

 00h 58min  0.55 m / ~ 4 g

Ready to Save to File

Save to File

Tip: Change the Name of the Print job by clicking on it. You can also view the Estimated Print Time and Cost.

Thank you for choosing Ultimaker, happy printing!