

Cura (Documentation for version 15.04.06)

Getting Started

Installation

To start the installation of Cura, [download](#) it first. After downloading, open the installer and run the installation wizard to complete the installation. To make sure Cura can run on your computer, we recommend to check the system requirements described below.

Accepted platforms

- Windows Vista or newer
- Mac OSX 10.7 or newer
- Ubuntu 14.04 or newer

System requirements

- OpenGL 2 compatible graphics chip
- Intel Core 2 or AMD Athlon 64 or newer

Compatible file types

- STL (most common file format for 3D printing)
- OBJ

Configuration

When you open up Cura for the first time, you will be asked to select the 3D printer that you have. If you have an Ultimaker 2, Ultimaker 2 Go or Ultimaker 2 Extended no other configuration is required and you can directly start using Cura. For the Ultimaker Original and Ultimaker Original+ you will have to follow the complete “configuration wizard” to do the configuration.

Configuration Wizard

Select your machine

What kind of machine do you have:

- ☒ Ultimaker2
- ☐ Ultimaker2extended
- ☐ Ultimaker2go
- ☐ Ultimaker Original
- ☐ Ultimaker Original+
- ☐ Printrbot
- ☐ Lulzbot TAZ
- ☐ Lulzbot Mini
- ☐ Other (Ex: RepRap, MakerBot, Witbox)

The collection of anonymous usage information helps with the continued improvement of Cura. This does NOT submit your models online nor gathers any privacy related information.

Submit anonymous usage information: ☒

For full details see: <http://wiki.ultimaker.com/Cura:stats>

< Back Next > Cancel

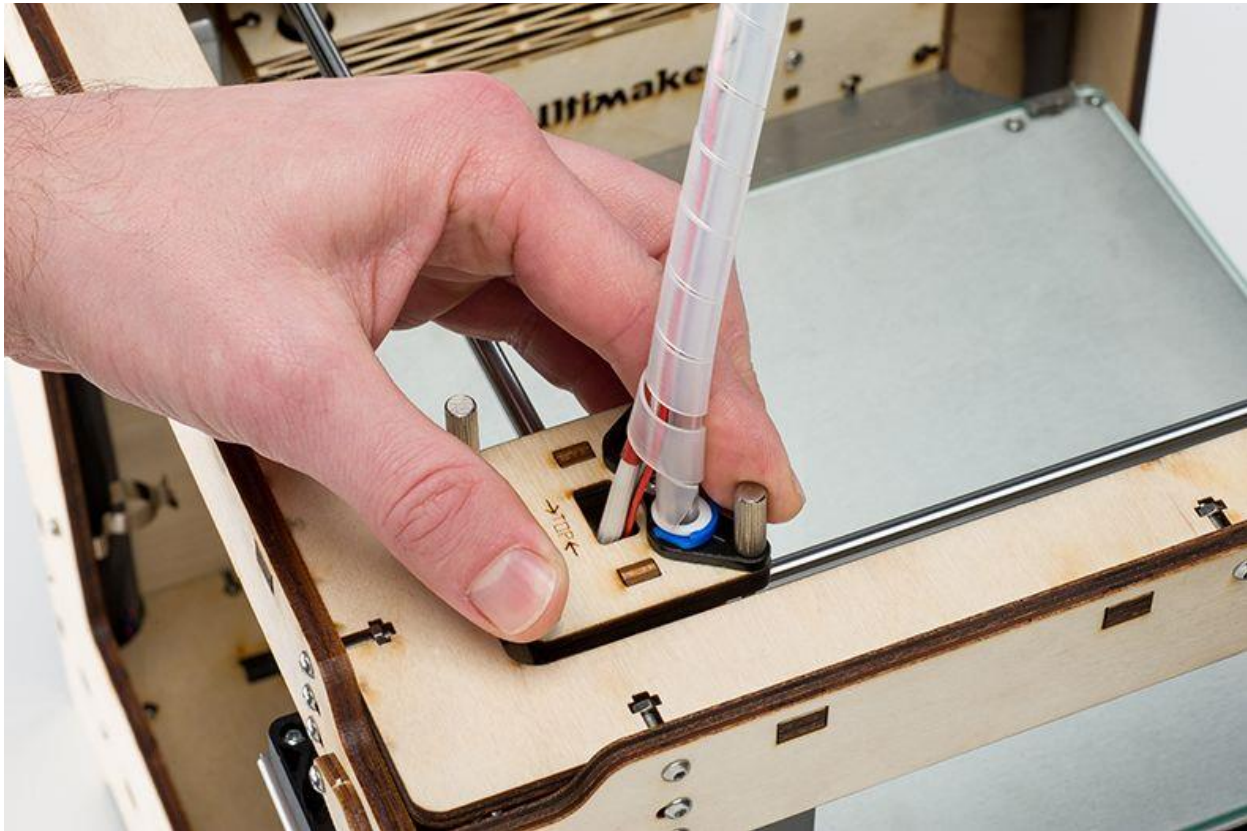
Select your Ultimaker for the correct Cura configuration

Configuration Ultimaker Original and Original+

The configuration wizard for the Ultimaker Original and Ultimaker Original+ will help you calibrating the printer and preparing it for the first use. Basically, it will do the following checks:

- The communication via USB

- Heating up the nozzle
- Checking the limit switches



Checking the limit switches

Furthermore you will be guided through the “bed leveling wizard” in order to properly set the height of the build plate. After setting the correct height you can let the Ultimaker print a square on the build plate to check the calibration.

For a more detailed explanation of the “configuration wizard” you can take a look at the Printer pages.

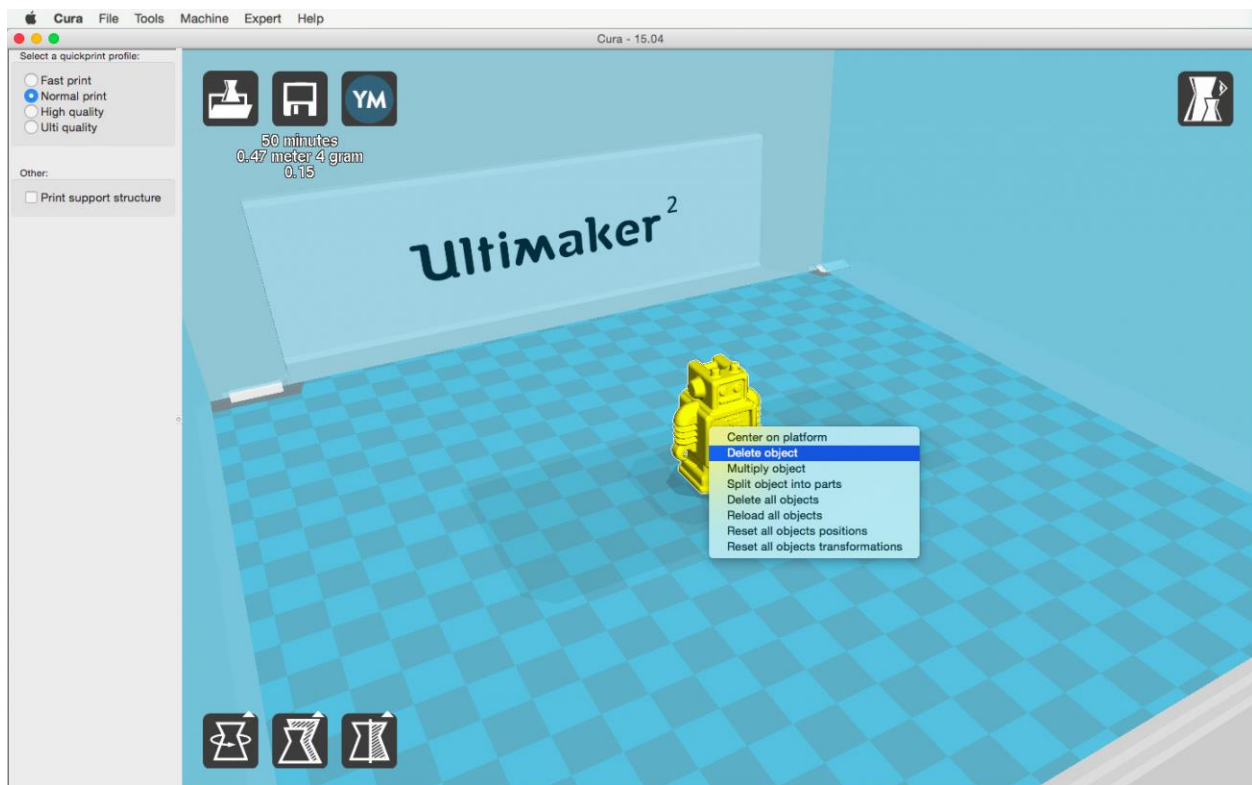
Interface

The below image shows you the main screen of Cura with the most important functionalities. Here you can load and adjust 3D models and select the preferred print profile, after which Cura will convert the 3D model to a 3D print file.

The Basics

Preparing a print file

When you use Cura for the first time you will see the Ultimaker robot model positioned on the platform. You can start converting this model into a print file, but if you want to use another model you can simply remove the Ultimaker robot model via “Preferences” > “Clear platform” or by a secondary click on the model and selecting “Delete object”.



Remove the model from the platform by selecting "Delete object"

Converting process

The basic process of converting a 3D model to a print file in Cura is described below.

1. Load a 3D model (STL, OBJ, DAE or AMF file) into Cura via the “Load” button.

2. Select one of the “quick print” profiles to use a default profile for your print, or switch to the “Full settings” via “Expert” > “Switch to full settings” if you want to have more control over the settings.
3. After you have done the settings and Cura has converted the file, you can save the print file (G-code file) via the “Save tool path” button to your computer or directly on the SD card when inserted.
4. At last, you can simply eject the SD card from your computer (make sure to safely remove it) and place the it in your Ultimaker to start the print.

Below the “Save tool path” button you will also see some information about the print. Cura will give an indication on the print time, required amount of material and weight of the print based on the settings that are used.

Note: you might notice a running progress bar just below the load button; this indicates that Cura is preparing the print file. This process will be restarted once a 3D model has been loaded or change in the print settings has been made.

Quick print profiles

Below you can find a short overview on the basic settings for each of the quick print profiles.

Fast print

Layer height: 0.15 mm

Shell thickness: 1 mm

Fill density: 10%

Normal quality

Layer height: 0.1 mm

Shell thickness: 0.8 mm

Fill density: 20%

High quality

Layer height: 0.06 mm

Shell thickness: 0.8 mm

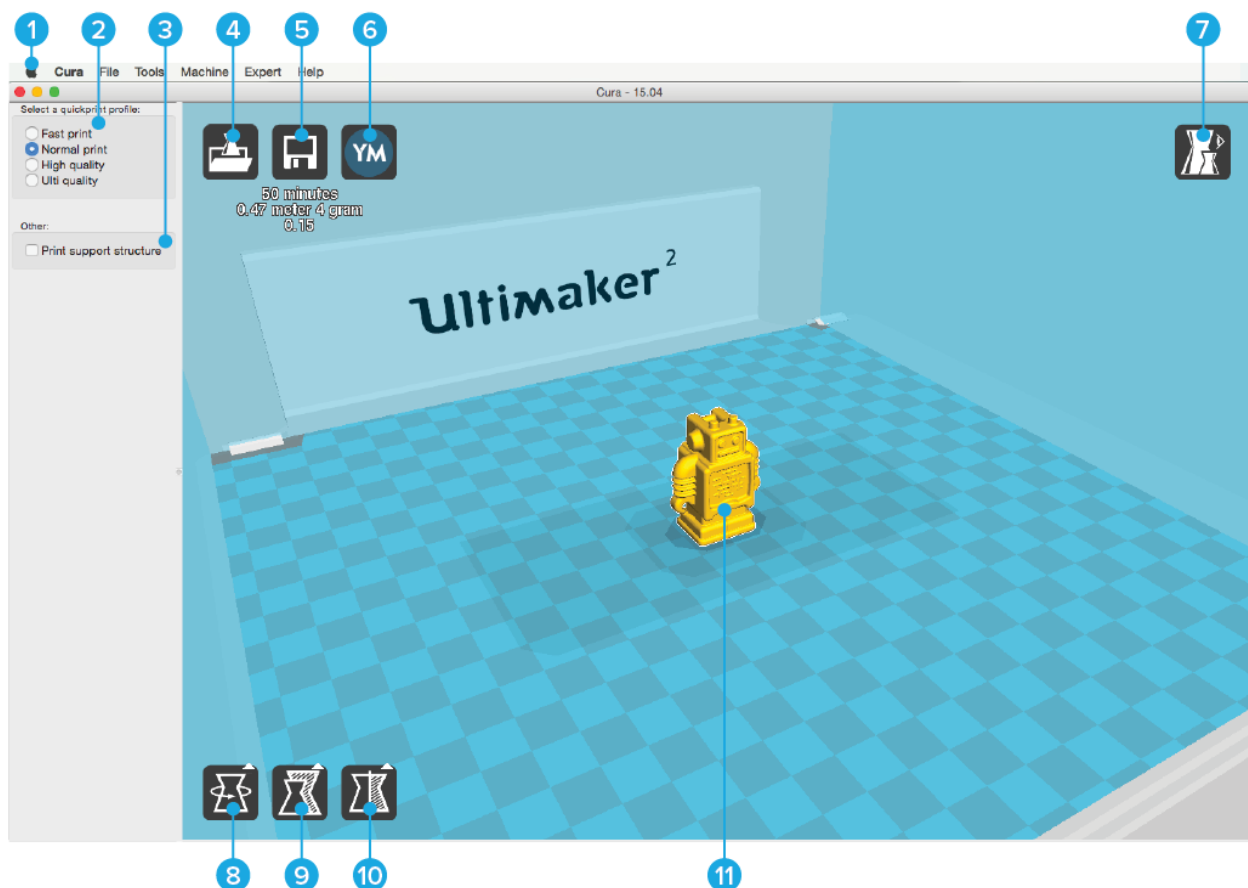
Fill density: 20%

Ulti quality

Layer height: 0.04 mm

Shell thickness: 0.8 mm

Fill density: 20%



The main screen

1. Menu bar: gives you access to preferences, machine settings and profiles.
2. Quick print profiles: default print profiles that can be used directly.
3. Support structure: offers the option to add support to the print
4. Load button: click this button to load a 3D model in Cura.

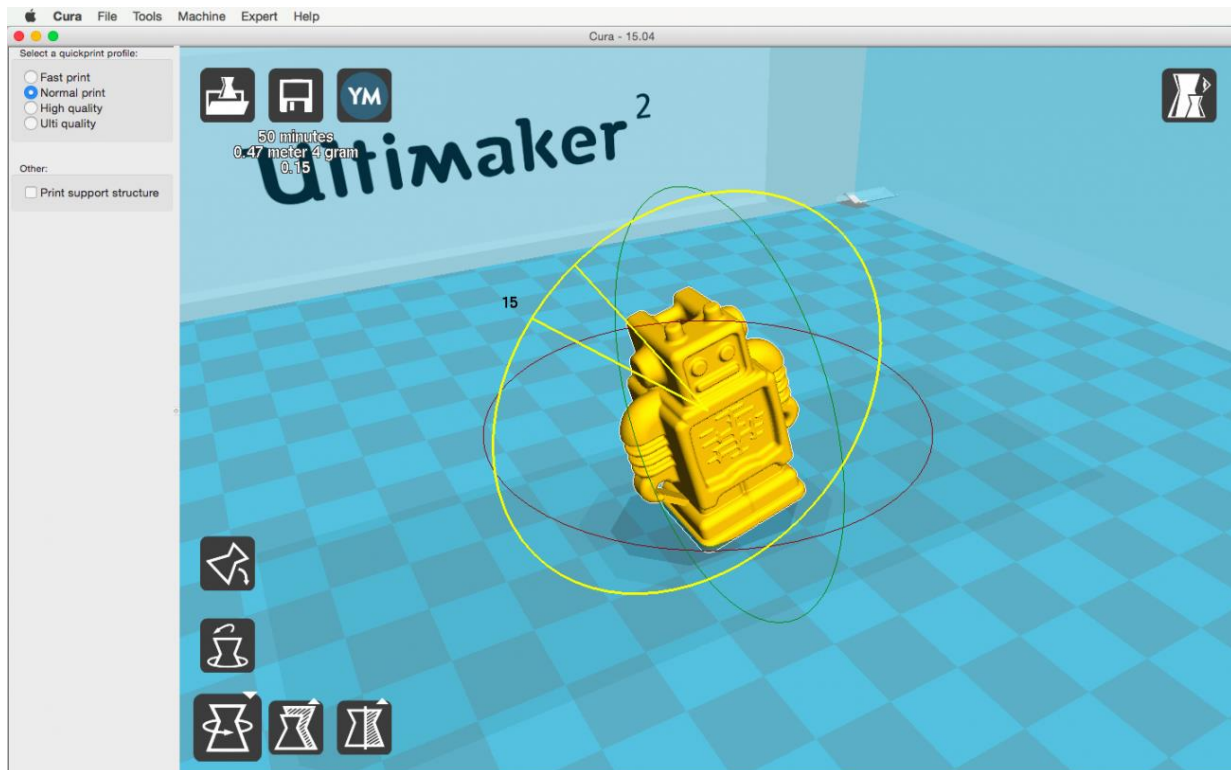
5. Save button: with this button 3D print files can be saved to your computer or directly on the SD card (if inserted).
6. YouImagine button: use this button to directly share 3D files on YouImagine.com.
7. View modes: use different view modes to check the printability of your model and to view the print path.
8. Rotate: rotate the model in the X, Y or Z direction.
9. Scale: change the (outer) dimensions of the 3D model.
10. Mirror: use this option to mirror the 3D model in X, Y or Z direction.
11. Visualization: shows the platform of the Ultimaker with the loaded 3D model(s).

Adjusting the model

Cura offers several ways of adjusting the model before printing it. This doesn't mean that you can change the actual shape of the model, but you can for example change its position and dimensions.

Rotate

The icon on the left side shows the Rotate button; it allows you to rotate the model in the X, Y or Z direction. By selecting one of the axes and moving your mouse you can rotate the object with an angle of 15 degrees. You can also rotate the object with an angle of 1 degree by holding "Shift".



Rotate the model with an angle of 15 degrees

Once the Rotate button has been selected a few more options will pop up just above the Rotate button. The “Reset” button offers the possibility to resets the model to its original position. By using the “Lay flat” option your model will be placed flat on the surface of the build plate. This can become handy when your model doesn’t have a completely flat surface or when it doesn’t load correctly in Cura. The model will be placed on the “flat” surface closest to the build plate.

Scale

The middle icon is the Scale button. This option can be used to change the dimensions of the model in X, Y and Z direction. Scaling can be done by clicking and dragging the (square) scaling icons that appear on the model, or by changing the numeric values that appear in the box just above the Scale button. You can either use a scale factor to change the dimensions or enter the exact size. A model will be scaled in all directions simultaneously by default, but you can unlock “Uniform scale” in order to scale in each direction independently.



Scale the model by dragging it

Furthermore a “Reset” and “Scale to max” button will be shown above the Scale button. The Reset button allows you to reset the model to its original dimensions. By using the Scale to max option the model will be scaled to the maximum size that fits in the machine.

Mirror

The most right icon at the bottom is the “Mirror” button. This can be used to flip the model in either the X, Y or Z direction.

View modes

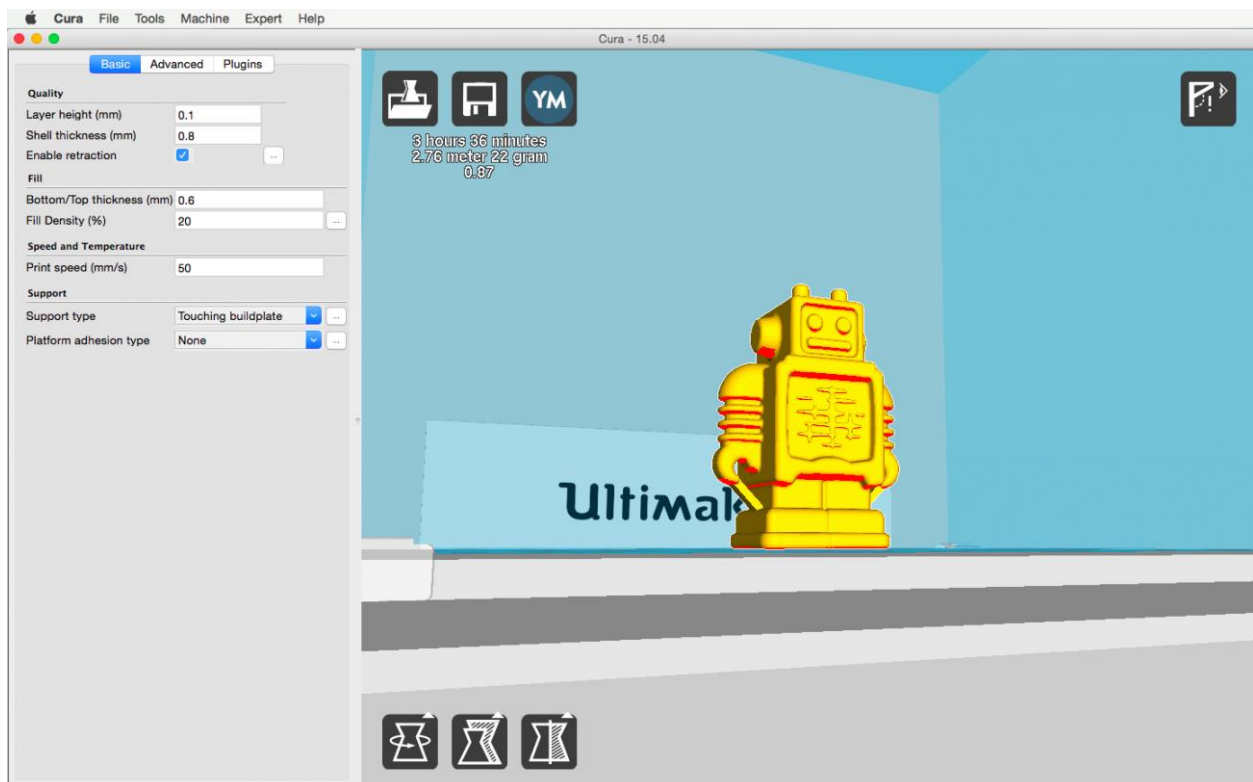
In Cura there are different ways of viewing the model on the build plate. You will find these different views by clicking the “View mode” button in the top right corner.

Normal

This is the default view mode and it will basically show the model in the way it was designed, which means that you can see the shape and size of the model.

Overhang

If you have a model with parts floating in the air or slanting parts it can be handy to use the Overhang view mode. Cura will show the overhanging parts in red, so that you can easily see if support is needed. The overhanging parts that are shown here are based on the “Overhang angle for support” in the Expert settings, which is set to an angle of 60 degrees by default.



Overhang is shown in red

Transparent

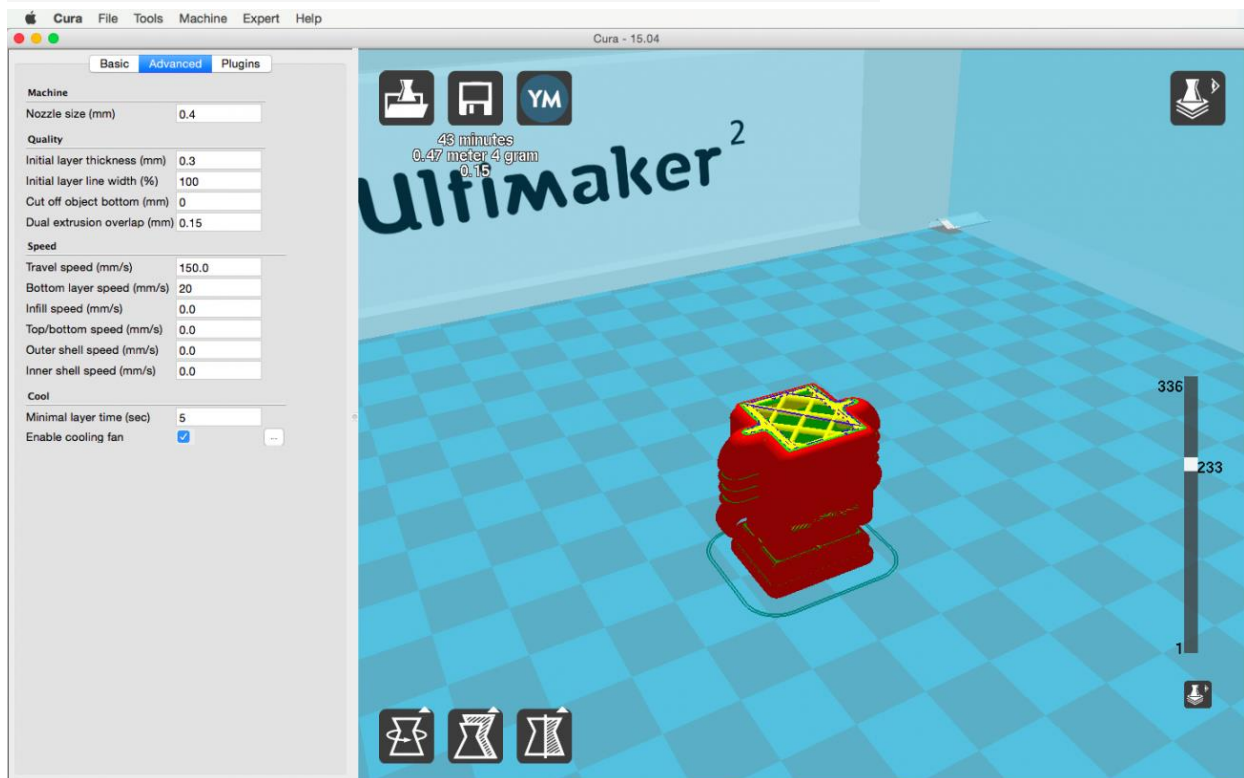
If you want to look through the model you can use the Transparent view. You can use this view mode if you have a model with internal structures.

X-ray

The X-ray view helps you to identify if there are gaps or intersections in the model (which could happen if the model is not designed properly). Red areas tell you that these parts will potentially cause problems while printing.

Layers

One of the often used view modes is the Layers view. Here you can exactly see how the model is going to be printed by scrolling through the layers. It's advised to use this view mode before printing the model because you can easily see if there are faults in the model/print or not.



By scrolling the bar on the right you can go through the model

Updating firmware

Once in a while a new Cura version and firmware version are released. In order to stay up to date it is therefore recommended to download the latest version of Cura once available and update your Ultimaker with the latest

firmware as well. The latest version of Cura (including latest firmware version) can always be found on the Software page.

In order to install the latest firmware to your Ultimaker, follow the steps described below.

1. Connect it

Connect the Ultimaker to your computer with the USB cable. For the Ultimaker 2 Family and Ultimaker Original+ you also need to attach the power supply and turn the Ultimaker on.

2. Uploading firmware

Start Cura and go to “Machine” > “Install default firmware” (make sure the correct Ultimaker is selected in the “Machine menu”). Cura will now automatically upload the latest firmware to your Ultimaker.

Full Settings

Basic

Under the Basic tab you can find the print settings that are changed most often. These settings will mostly define the quality and strength of your print.

Quality

The layer height is probably the most often changed setting. By default the layer height is set to 0.1 mm, but you can of course use thinner layers to increase the quality or use thicker layers for faster prints.

You will also see that retraction is enabled. Retraction means that the filament will be pulled back when the nozzle is moving over an area where it doesn't have to print - for example when you have 2 pillars on a certain distance from each other. By retracting the filament we make sure that no filament is coming out of the nozzle while the print head is just moving. This

way “stringing” (thin threads of plastic in between the printed objects) is prevented. The below image shows an example of what happens when retraction is disabled.



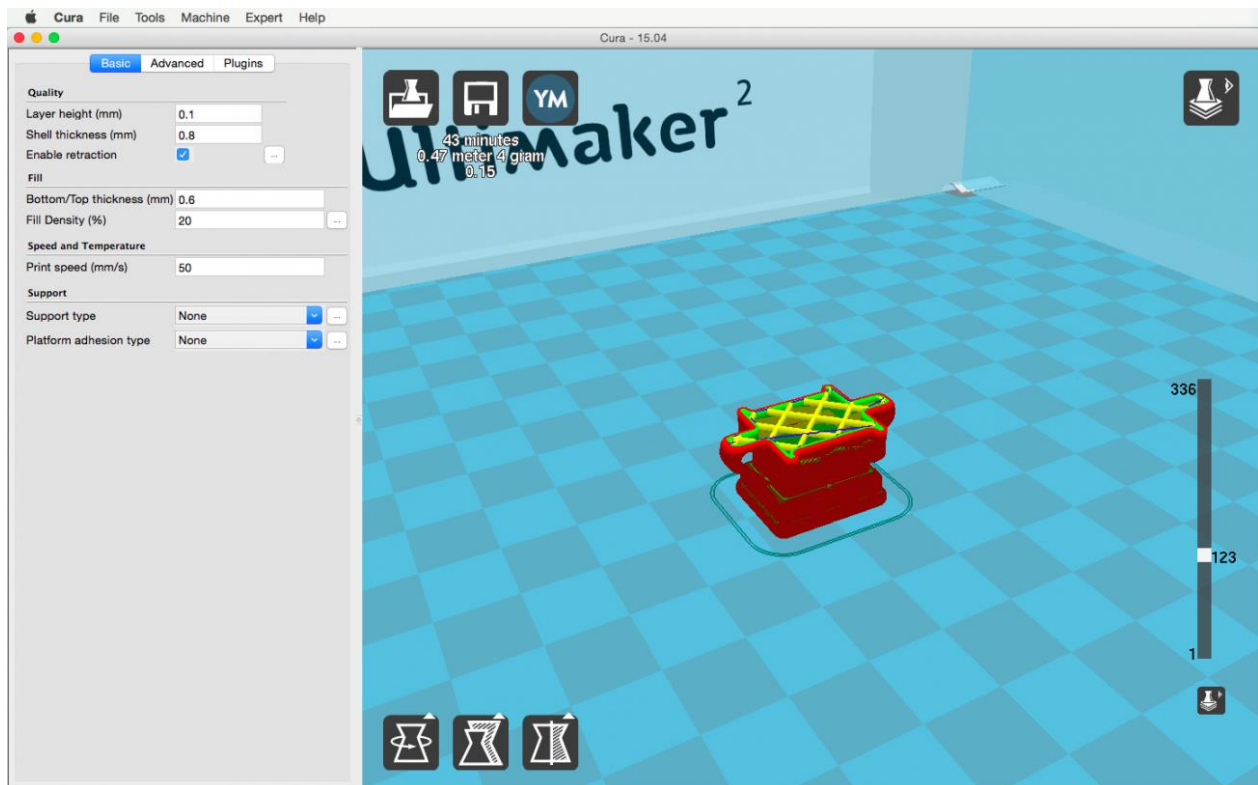
An example of stringing

Although retraction usually works very well it is possible that you still see some stringing, even though retraction is enabled. In this case you can try lowering the nozzle temperature, so that the plastic is a little less liquid when coming out of the nozzle, thus decreasing the amount of stringing. Keep in mind that lowering the printing temperature means that you (probably) need to decrease the print speed as well to ensure the plastic will completely melt before it leaves the nozzle.

Fill

These settings very much relate to the strength of the print. A higher fill density of course means that there’s more plastic on the inside of your print,

leading to a stronger object. You can also decide to print the object completely hollow, which is desired in some cases.



In the Layers view you can already see the fill density

The bottom/top thickness defines the amount of solid layers of the bottom and top of the print. Usually this is set to 0.6 mm, but it might be necessary to increase this in some cases. For example if you have an object with a big flat top surface or want to print with a low fill density more layers may be required to completely close the top of the print.

Speed and temperature

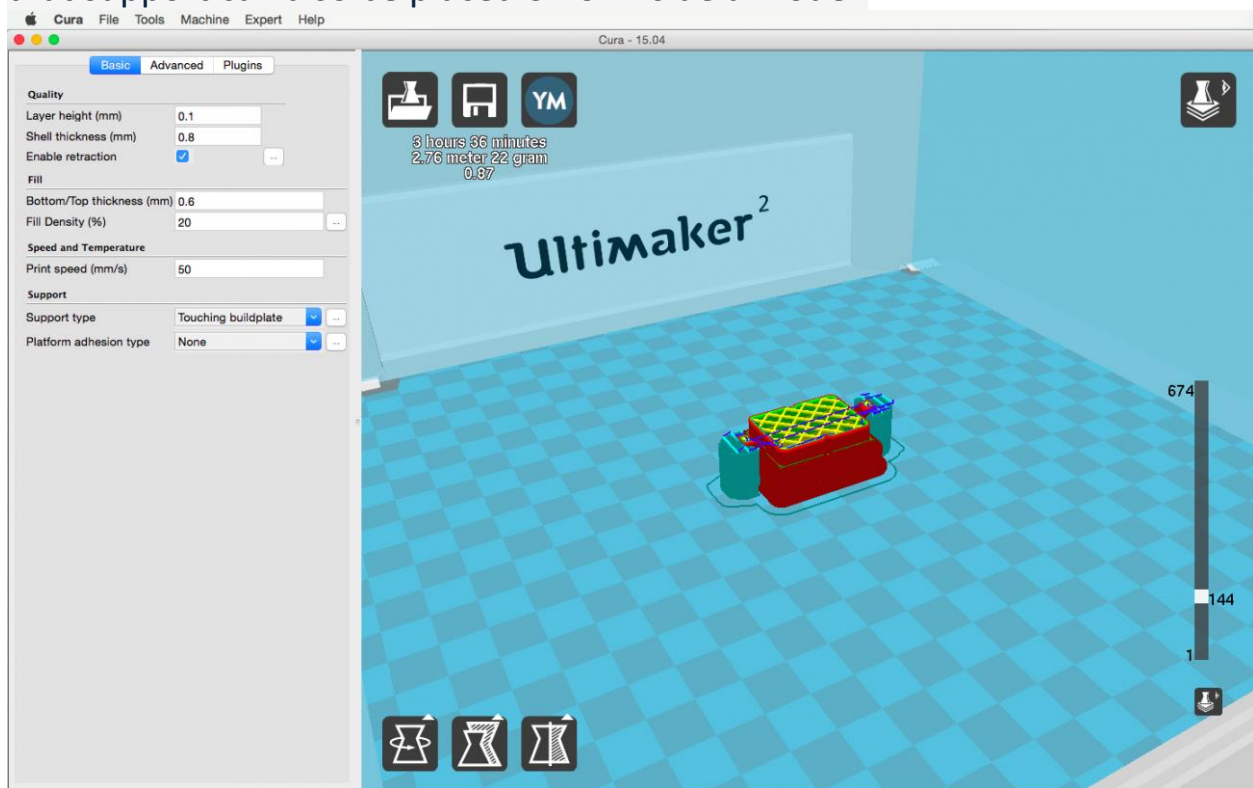
The speed and temperature that are used for printing have a big influence on how the plastic comes out of the nozzle and depend a lot on the material that is being used. The print speed basically tells with which speed the print head is moving while it's printing; based on this speed the amount of plastic that needs to be extruded is calculated. The print speed is set to 50 mm/s by default, but this can easily be changed if you want to print faster. Keep in

mind that increasing the speed means that you might have to increase the temperature as well to ensure the plastic is properly melted.

If you have selected the Ultimaker Original or Ultimaker Original+ in Cura you will also see the printing temperature. Depending on the material that you want to use for the print you might need to change this value. The Ultimaker 2 Family will have all material settings in the machine, where you can simply select a material profile.

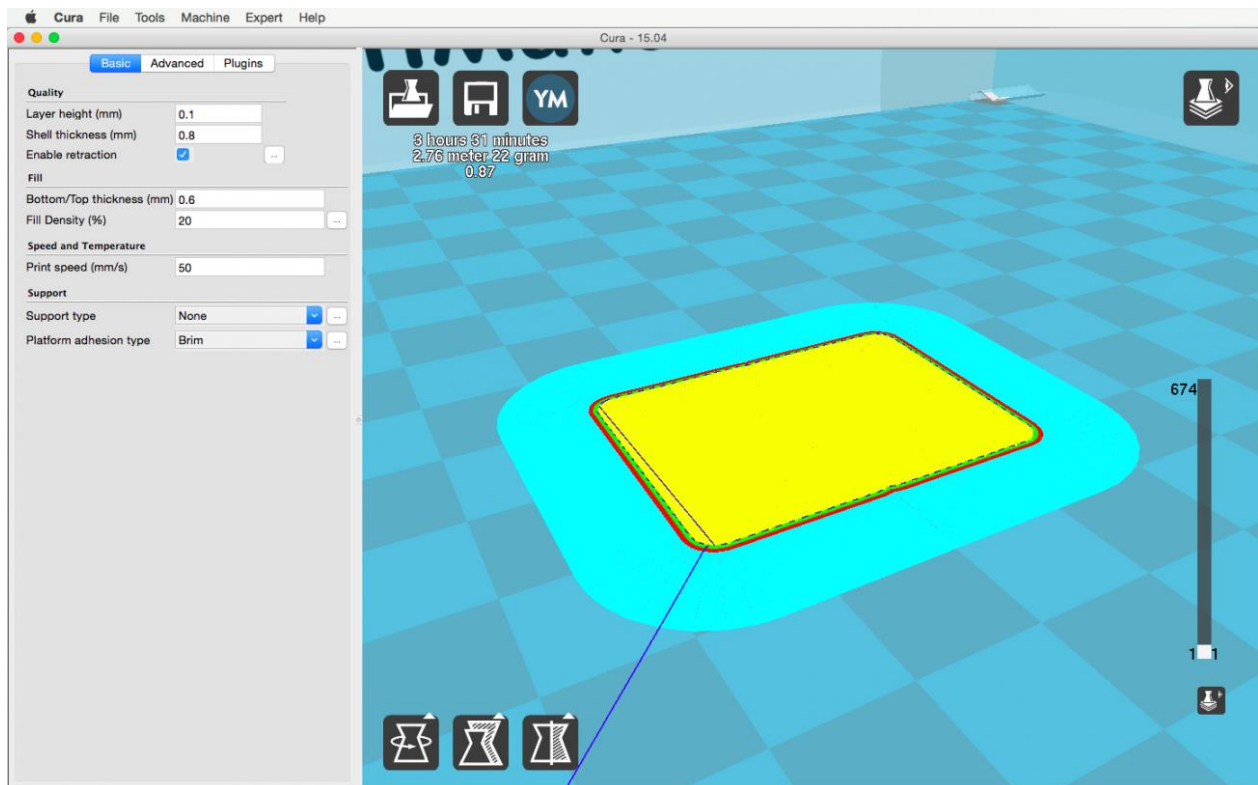
Support

Some models have overhanging parts, which means that part of the model floats in the air when you would print it. In this case you will need support to prevent the plastic from falling down. In Cura you can either select “Touching buildplate” support or support “everywhere”. With touching buildplate support Cura will generate support at the places where it can reach the floating parts from the build plate. Support everywhere means that support can also be placed on or inside a model.



Support is shown in light blue in the Layers view

When your model doesn't have a completely flat bottom or if you want to improve the bed adhesion you can use one of the platform adhesion types. A raft adds a thick grid between the model and the build plate and with a brim some extra lines of plastic are placed around the object on the first layer of the print. Especially brim is used a lot and works very well if you want to decrease the amount of warping on your print.



A brim is shown as light blue lines around the model in the Layers view

Filament (only UMO/UMO+)

For the Ultimaker Original and Ultimaker Original+ you can also change filament settings. The filament diameter of Ultimaker filament is always 2.85 mm, but if you use filament from other suppliers you might need to change this diameter setting in Cura.

Furthermore the flow can be changed, which defines the amount of plastic that is extruded based on the print speed and filament diameter. We

however advise to keep the flow on 100%, unless you are experimenting a lot with all the settings.

Advanced

Advanced settings are usually only changed if you have specific needs for which the default settings are not sufficient.

Machine

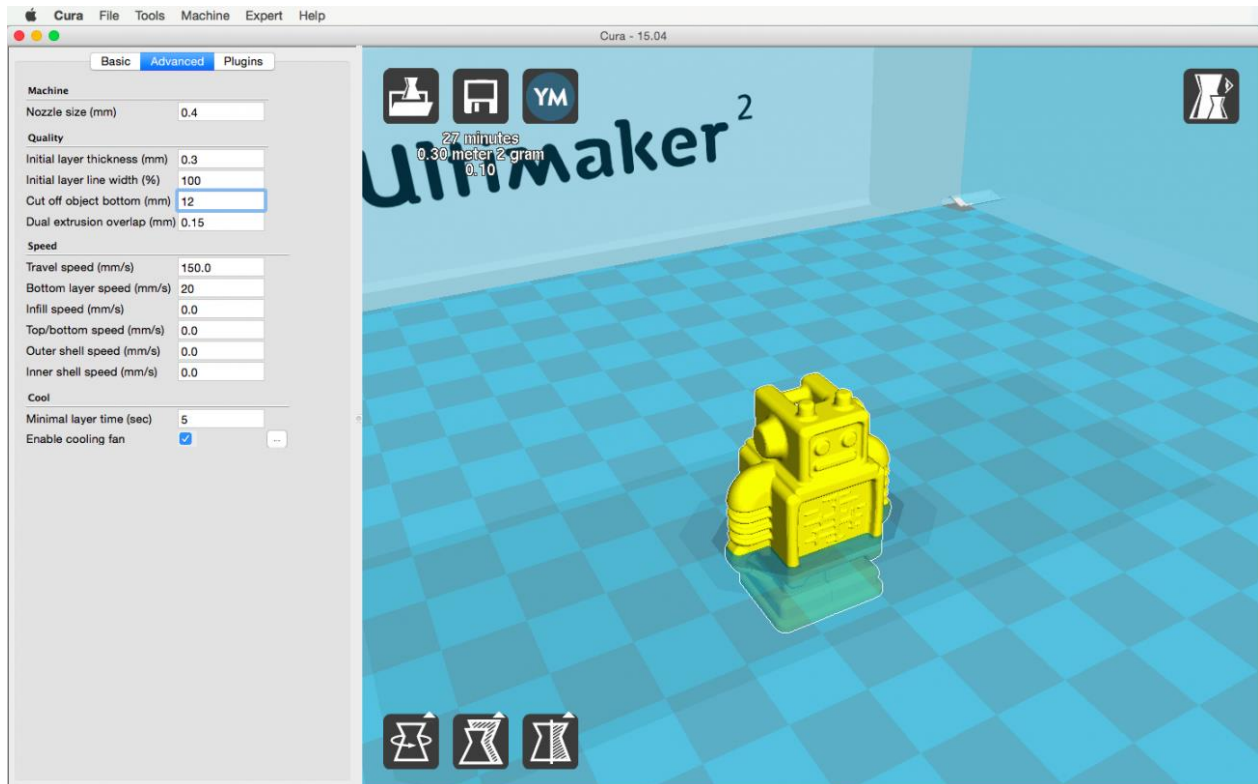
The nozzle size is a machine setting that you can change in the Advanced settings. All Ultimakers have a nozzle with a diameter of 0.4 mm, but if you're experimenting with a different nozzle you can set the correct diameter here.

Retraction (only UMO/UMO+)

Here you can see the speed and distance with which the filament is retracted. The default settings will work fine for most prints, but if you're using other materials or have prints with many retractions you might need to change these settings for the best results.

Quality

There are some more settings related to the layers and quality of the print. The initial layer thickness defines the thickness of the first layer of the print. By default this is set to 0.3 mm, which might be rather thick, but it will make it easier to print the first layer and make sure it sticks well. Also the "cut off object bottom" setting is an option that can be very handy. It allows you to lower the position of the model so it sinks through the platform. This can be desired if the bottom of the model is not completely flat or if you just want to print part of the model.



The model is sunk 12 mm into the platform

Speed

Besides a default settings for print speed in the Basic tab you can also set the speed for specific parts in the print. You can change the speed with which for example the infill or bottom layer are printed separately.

Cool

As cooling of the plastic is very important there are also some cooling parameters that you can set to your needs. The cooling fan is enabled by default and you can change the minimal layer time. The minimal layer time defines the minimum time it has to take for printing one layer. This way we ensure that the plastic has enough time to cool down, before starting the next layer.

Expert settings

For more experienced users there are also 'Expert' settings. The average user won't really use these settings, but if you're an advanced user and want to experiment a lot it could be interesting to play with these settings.

You can also access the Expert settings of the specific setting by clicking on the "..." box displayed next to the setting in the Basic and Advanced tabs.

Retraction

Here you can change settings that relate to retraction of the filament. Overall, the default settings will work fine, but if you're using different materials you might require different retraction settings as well to achieve the best results.

Skirt

The skirt is a line that is printed around the object on the first layer. It helps to get the extrusion running and it's a good way for checking the bed leveling just before your model is being printed. In some cases you might want to change the Skirt settings; for example if you are printing an object that covers the complete surface of the build plate and want to double-check the bed leveling.

Cool

Although the fans on the Ultimaker will always be full on after the first couple of layers you can also change the cooling parameters here. Especially if you are working with different materials that require different temperatures and cooling it might be helpful to adjust these settings.

Infill

Besides just setting the fill density and thickness of the bottom and top of the print there are a few more infill settings that can be changed. Especially the "Solid infill top" and "Solid infill bottom" are used quite often. If you disable one of these settings it means that either the top or bottom layer of

the print won't be printed. For example when printing a vase you can turn the "Solid infill top" off and set the fill density to 0%.

Support

In some cases it's better to not use the default Support settings, but to play a bit around with it. A different structure, overhang angle or fill amount can for example result in more or stronger support. Depending on the model this can lead to a better end result.

Black magic

These are extra options that can be useful in specific cases. When selecting the "Spiralize the outer contour" option, you can get rid of the seam in the print. Usually the height of the Z-stage will drop a bit after each layer, but with this option it will gradually move down (like a spiral). For printing single-walled objects (e.g. a vase) this option can be very useful. The "Only follow mesh surface" option allows you to print only the outside of the model (but without bottom and top).

Brim

Here you can change the size of the brim. More lines means a bigger brim and even better platform adhesion. In most cases the default of 20 lines will be sufficient.

Raft

For the Raft there are a lot of parameters that can be changed. We won't go in to detail here, but if you're working a lot with rafts it can be helpful to experiment with these settings.

Fix horrible

If a model is not correctly designed for 3D printing it can occur that the Ultimaker won't print the model as desired. There could for example be gaps or intersecting parts in the model. The "Fix horrible" settings often help

fixing these faults in models. You might need to try several combinations and check the result in the Layers view.

Plugins

More info coming soon.