FOLLOW ALONG: ULTIMAKER CURA AS A BEGINNER'S GUIDE FOR SLICING

This practical session will explain how to download and install Ultimaker Cura, which is a popular slicer, and then define the printing settings, in order to have a global compression about defining parameters for the step 3 in additive manufacturing basic steps.

Even though there are a lot of softwares and even each technology has its own, this practical part can help to introduce us to the importance of the different printing parameters for the final quality of a part.

For this part, please download the ***ReginnaPart_NO_Supports*** and ***ReginnaPart_Supports*** files available in the platform.

Content

FOLLOW ALONG: ULTIMAKER CURA AS A BEGINNER'S GUIDE FOR SLICING 1

PART 1: DOWNLOAD & INSTALL ULTIMAKER CURA 2

PART 2: SETTING UP CURA SOFTWARE 4

PART 3: WORKSPACE OVERVIEW 5

PART 4: IMPORTING AND PREPARATION OF MODEL 7

PART 5: SETTING MATERIAL PARAMETERS 10

- **FOR CREATING AND MODIFING PARAMETERS OF MATERIAL (WHEN YOU NEED CUSTOM)** 11

PART 6: PRINT SETTINGS (PARAMETERS) 13

- **QUALITY SETTINGS** 15
- **WALL SETTINGS** 16
- **TOP/BOTTOM** 17
- **INFILL** 18

PART 7: SUPPORTS AND BUILD PLATE ADHESION 21

PART 7: SLICE AND PREVIEW 24

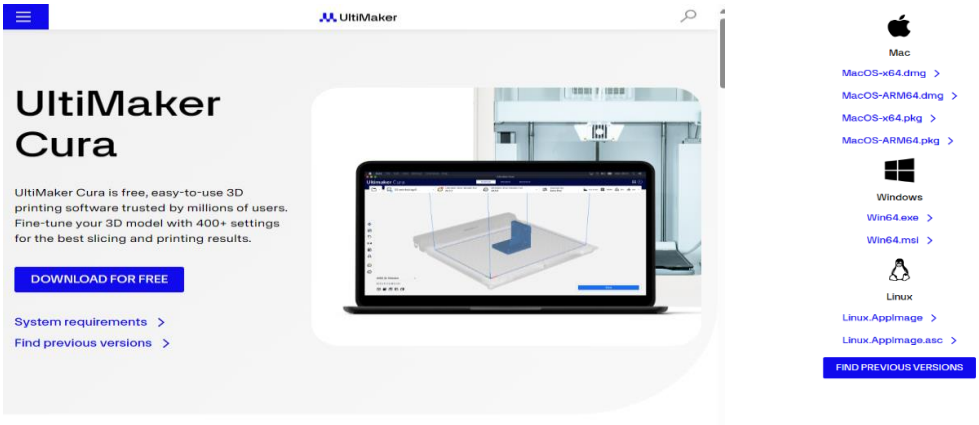
PART 1: DOWNLOAD & INSTALL ULTIMAKER CURA

Ultimaker Cura is free open source programme which allows you to select 3D models and it prepares them for a 3D printer. (Step 3 of Basic steps of AM process)

Step 1: Download the Cura Software.

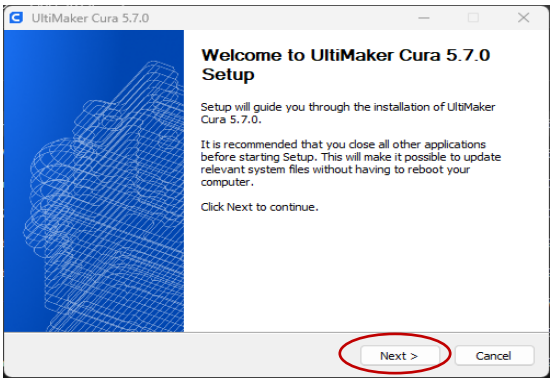
Go to the Ultimaker website, click Software and then Ultimaker Cura. You can also follow the next link: https://ultimaker.com/software/ultimaker-cura/

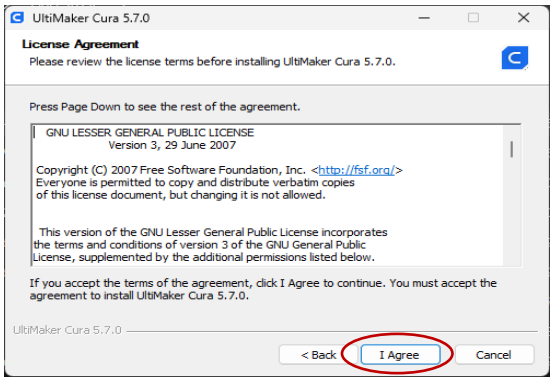
Click the “Download for free" button. Then it’s going to give you different options depending on your computer.

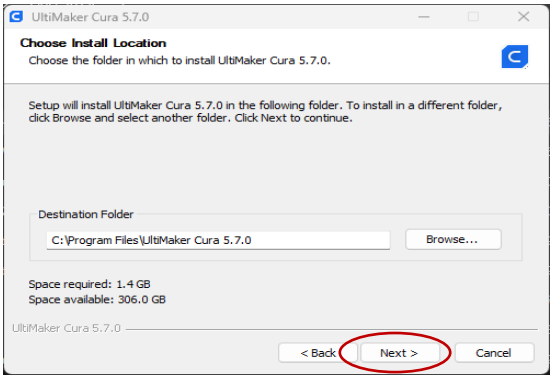


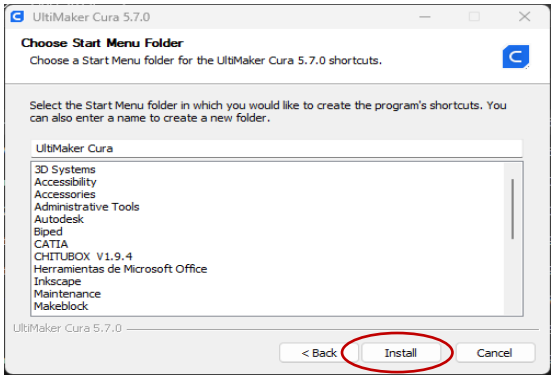
Step 2: After download the executable, install the software.

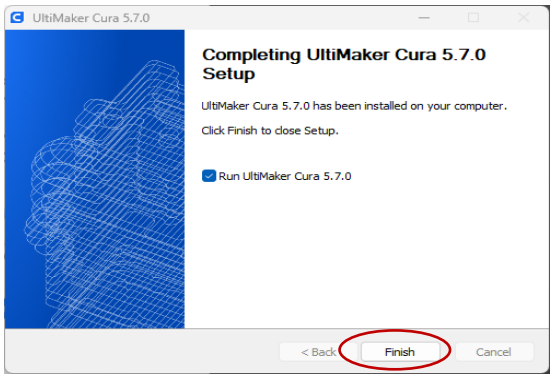
It might take some minutes.





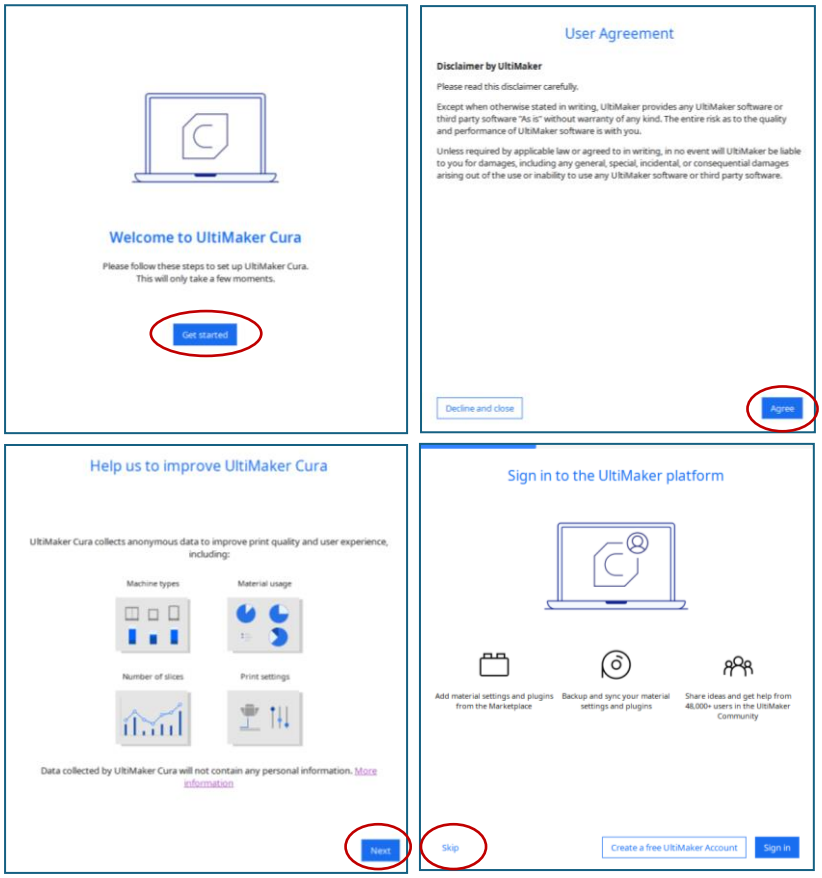






Step 3: After installing, open the software.

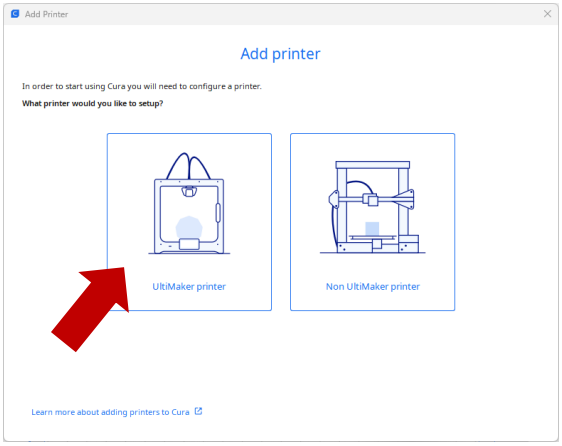
If it's the first time the next windows will pop up. Just **click on the blue buttoms**, except for **Sign in to Ultimaker platform** window, that for this lecture we are going to **skip**.

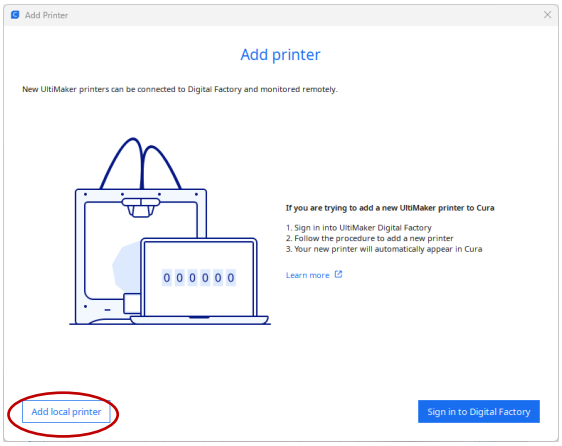


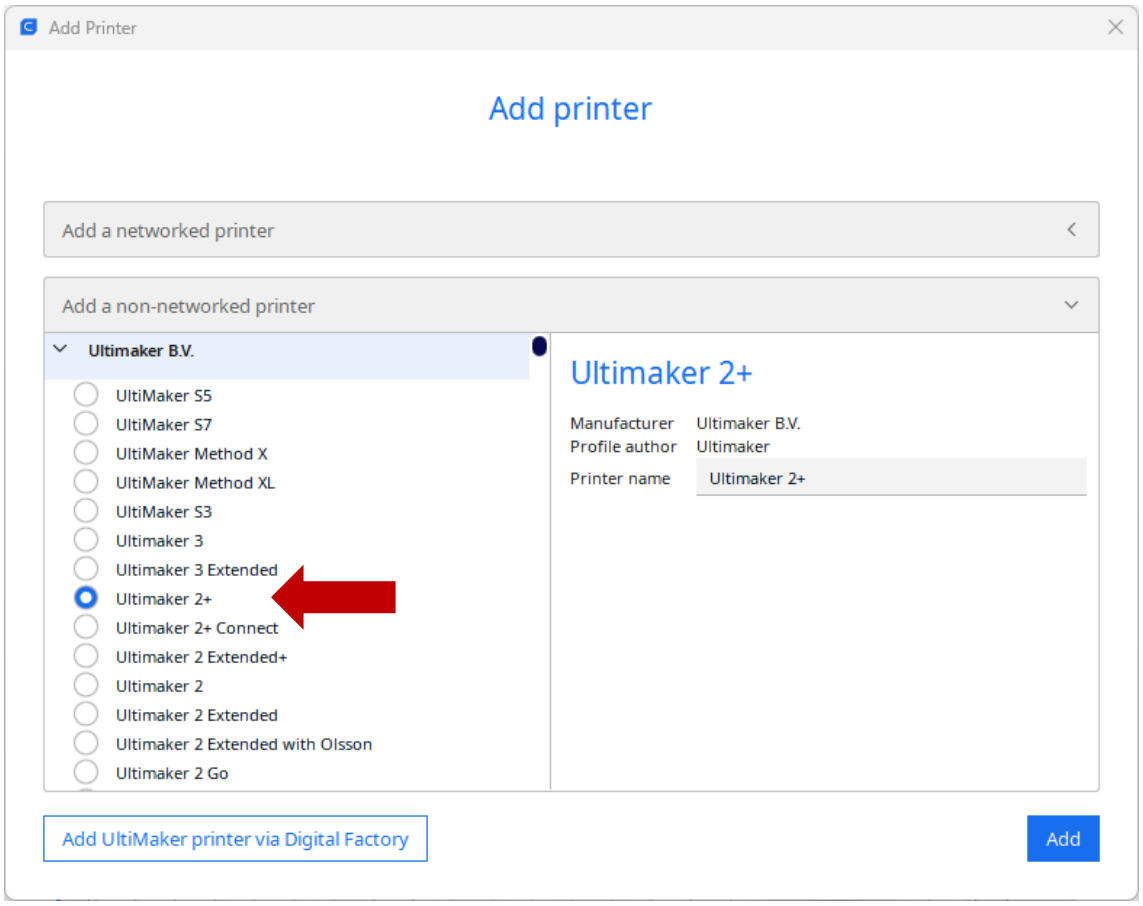
PART 2: SETTING UP CURA SOFTWARE

Step 1: Configure Cura Software with your printer.

In the **Add printer** window, we have two options depending on your printer. You can add *Ultimaker Printer* or a *printer for another brand*, and even though customized printer. For this lecture, we are going to select an Ultimaker printer, but later we will see how to add more printers as well.





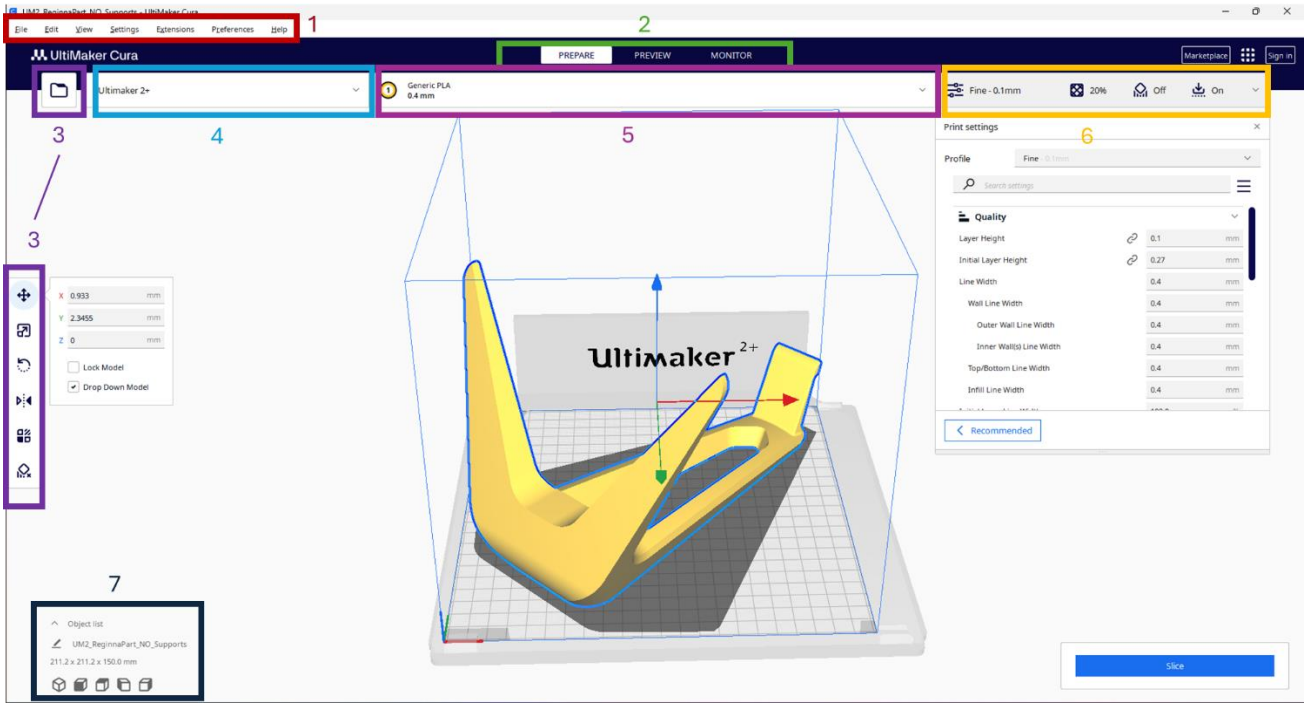


And we finish installing and setting up the software.

PART 3: WORKSPACE OVERVIEW

After selecting the printer, the following display it's going to appear. This is the representation of the printer, our **workspace**. As it can be seen in the middle we have a plane, which is surrounded by blue lines. This is our **build plate,** and the lines limit the **print area** of the selected printer.

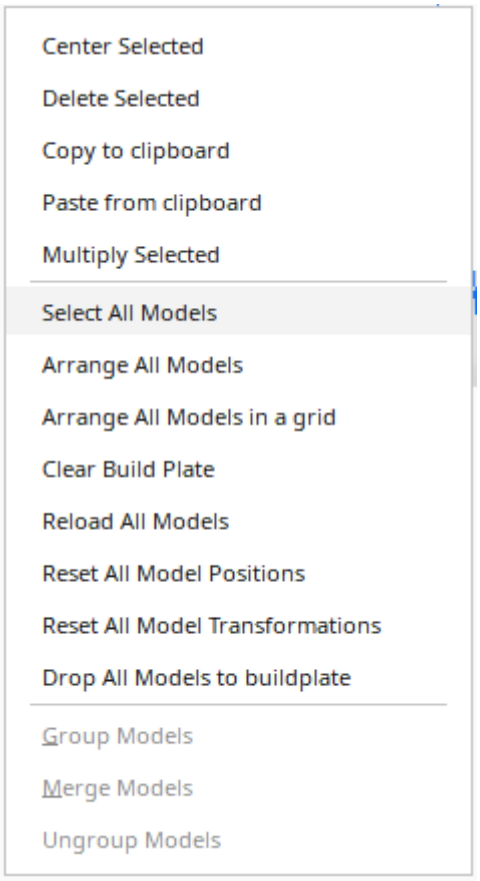
The interface has the following parts:



1. **Main menu**: We can access all the options that the 3D software has.
2. **Stages window**: now we are at the Prepare stage. This stage allows you to define the parameters, material, etc., in order to make a good quality part. In the Preview stage, we can see how it's going to be printed. The Control stage allows to monitoring the progress of you part, but since we need to have a printer connected, we are not going to use it for this lecture.
3. **Open file & Setting toolbar**: with this icon, we can open the 3D model file (.stl, .obj or .3mf) and when the model is loaded, the setting toolbar appears within and allows you to modify the object (move, rotate, scale, mirror effect and more).
4. **Printer menu**: printer defining and configuration.
5. **Material menu**: this menu allows you to select the material that can be predefined or customised and define parameters related to the material. Also we can select the nozzle.
6. **Print settings (parameters).**
7. **View menu and model information:** The ways of displaying parts. The cubes refer to which position the object is going to have from our view, isometric view, top, buttom, right or left side, and the drop-down will show 3 very useful will show 3 very useful visualisation forms. On the other hand, we also have the model information which contains the name of the 3D model. Here we will see the dimensions of the part, length, width and height

MOUSE SHORTCUTS FOR MOVEMENT

- Right click+hold: **rotate**
- Mouse wheel: **zoom**
- Mouse wheel+hold: **printer placement**
- Right click: **shortcut menu**: It offers several ways to organise your 3D models with a single click to improve workflow efficiency.



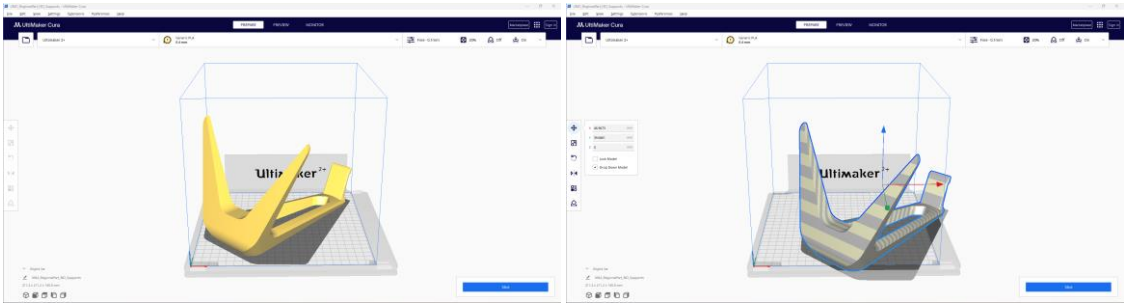
- **Centring the selected model** on the build plate
- **Remove the selected model** from the build plate
- **Multiply the selected model** to create a number of copies, defined in the pop-up menu.
- **Select all models** on the build plate, for simultaneous adjustment.
- **Arrange all models** automatically on the build plate. This is especially useful when several smaller 3D models are loaded simultaneously.
- **Clear build plate** for loading new 3D models.
- **Reload all updated models** on the build plate, if the 3D model has been updated or replaced by a new 3D model. been updated or replaced by a new version outside of Ultimaker Cura.
- **Reset all model positions** on the build plate
- **Reset all model transformations** on the build plate
- **Drop all models to buildplate** (in case they are above z=0)
- **Group, merge and ungroup models** to prepare them for dual extrusion or fit them simultaneously It must to be more than one part for this.

PART 4: IMPORTING AND PREPARATION OF MODEL

Step 1: Open your model (stl, obj o 3mf).

Click on the folder icon and select the part with the name of: *ReginnaPart_No_Supports*.

When the model is loaded, if it is on the **build plate** is displayed in the **colour** corresponding to the material selected for printing. Models that are **not completely within the printable area have a striped grey overlay.**

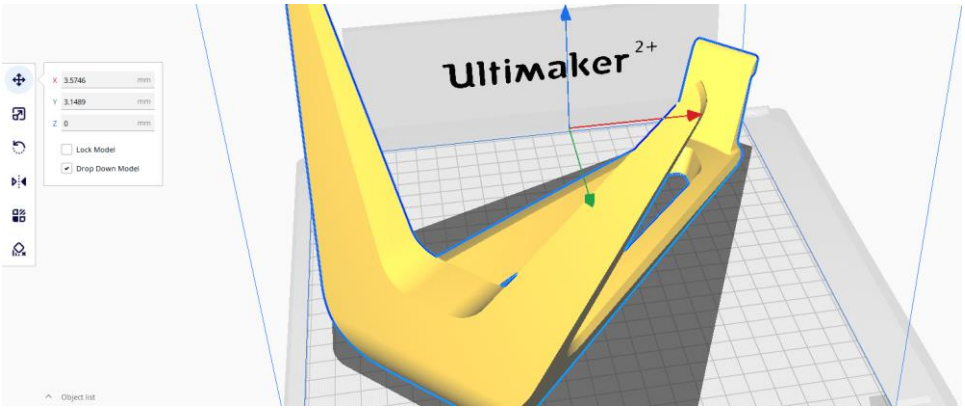


Step 2: Preparation of the model

After opening the file, we get a **setting toolbar** with the file modification options (**move, scale, rotate, mirror effect and more options**).

Move: You can use this setting to move and change your model's position on the build plate.

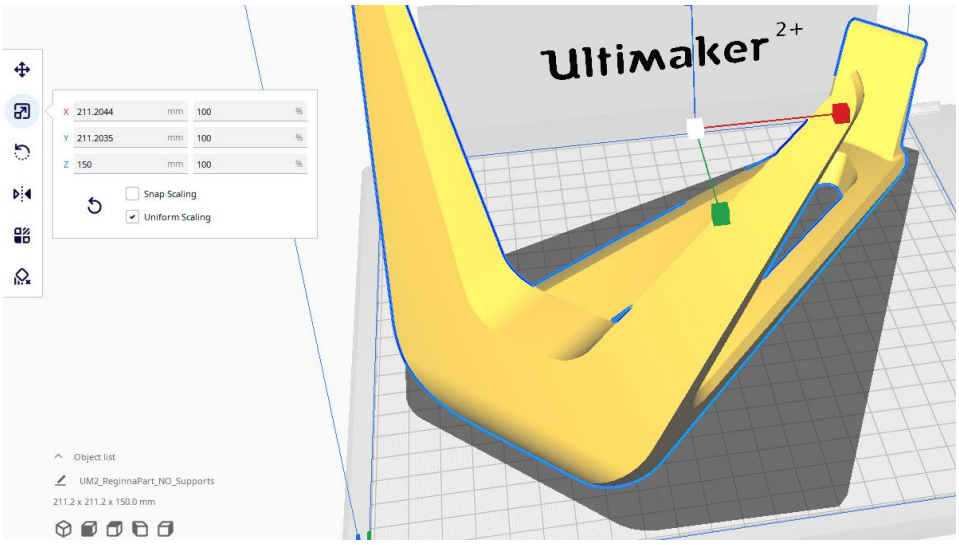
You can move the model in three ways. One involves using your **mouse to drag the model** to your desired spot. In the other method, you can **input your desired X, Y, and Z** coordinates in the box, and the model will automatically move to that position. And finally, you can **drag from the lines of the coordinate system** in your part.



Scale: if you want to increase or decrease the model's size, you can use the scale tool for that.

You can **drag each system's axis** to increase the model's size in that direction. You can also use the more accurate **percentage system** to scale your model or **numbers** in mm. If

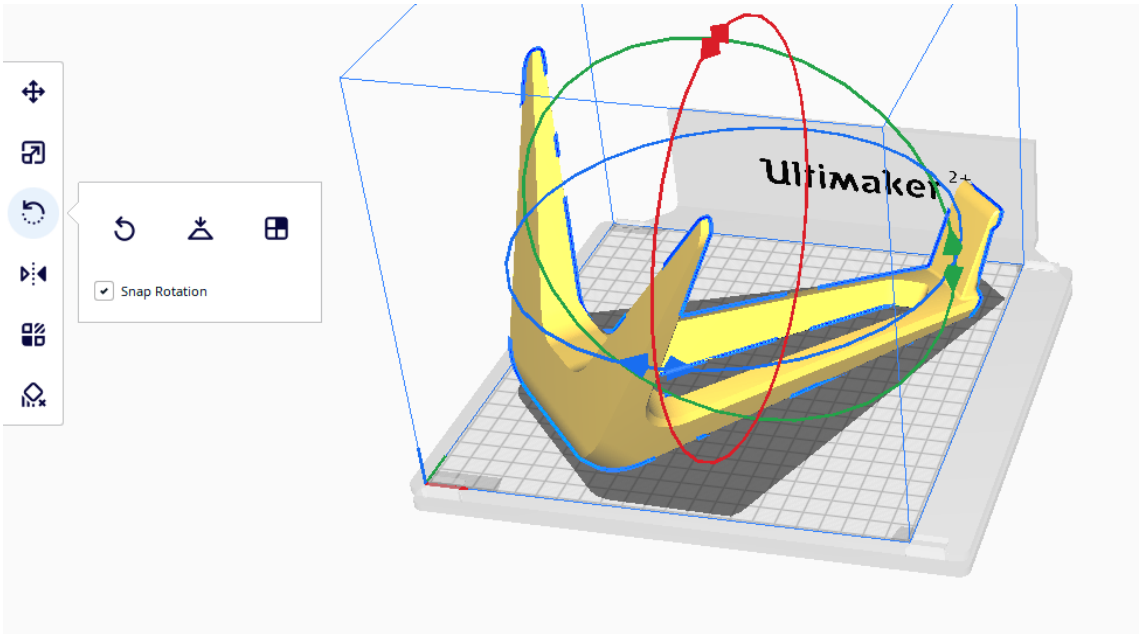
you're going to scale **all axes by that factor, tick the uniform scaling box**. However, If you want to scale a particular axis, untick the box.



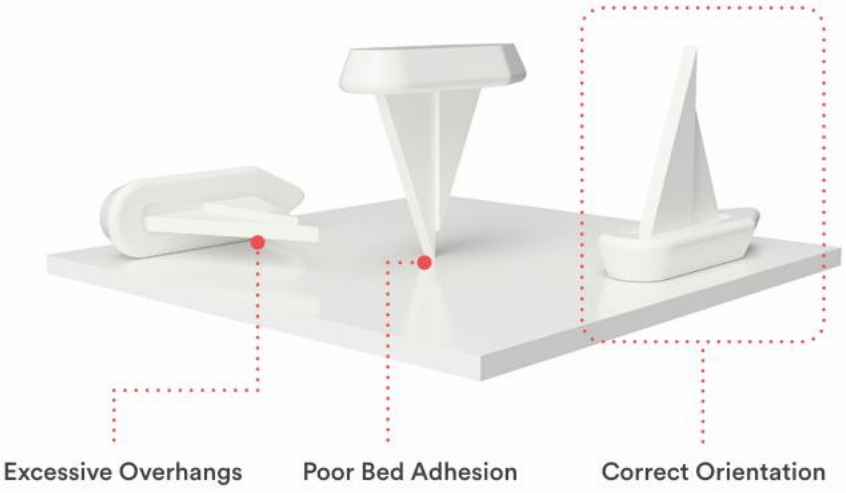
Rotate: You can use the rotate icon to change the model's orientation

Once you press the rotate icon a series of red, green, and blue bands will appear on the model. By **dragging these bands**, you can change the orientation of the model.

Also, we have the options **Lay flat** and **Select face to align with the build plate**. The first one will automatically select the flattest surface on your model and rotate it so it's lying down on the build plate. The second one will turn a face that you selected to the build plate.

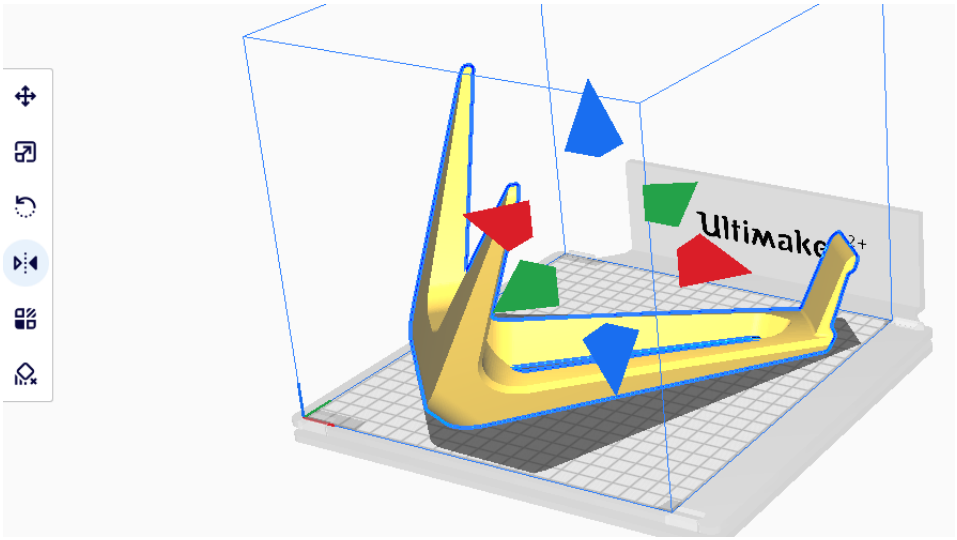


This step is so important because a bad orientation might cause problems in printing or excessive generation of supports. Take a look at the following example of orientation.



Mirror: You can quickly flip the model you're working on 180° in any direction with it.

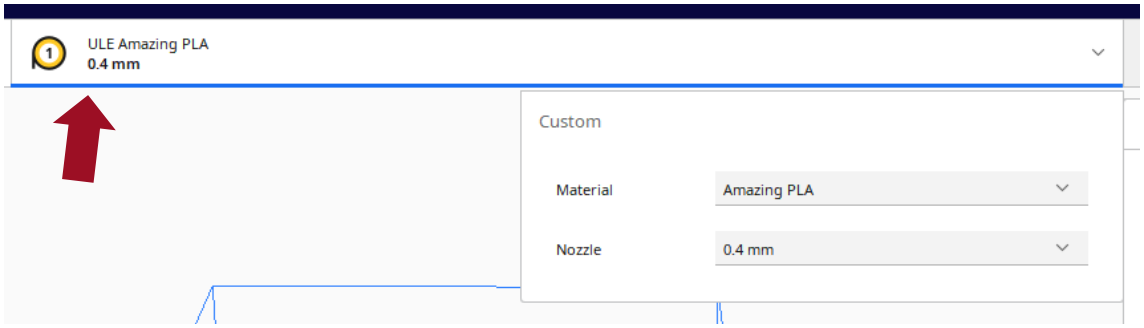
Click on Mirror and you will see several **arrows** on the model. Tap on the arrow pointing in the direction you want to flip the model, and voilà, you've turned it.



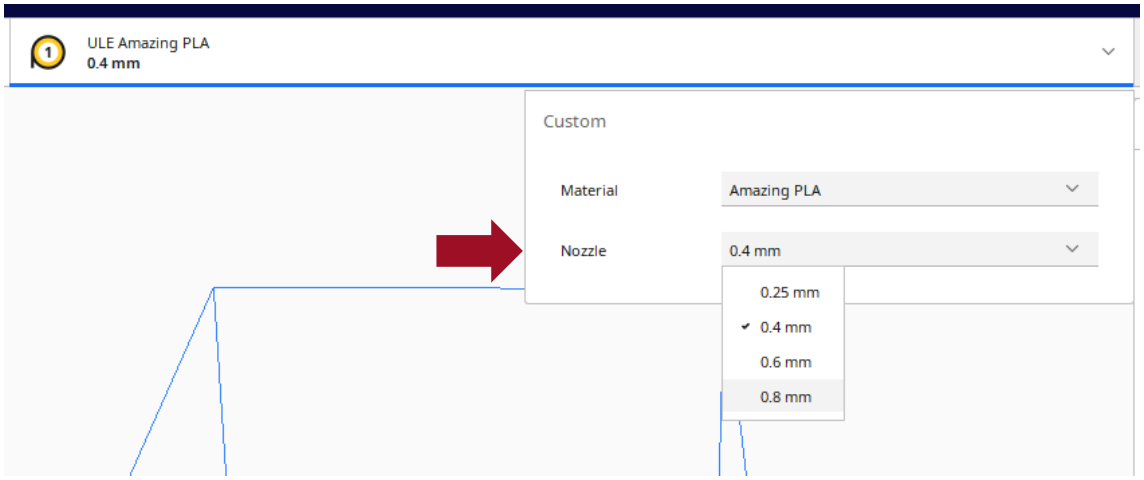
PART 5: SETTING MATERIAL PARAMETERS

It's crucial to select the exact type of material and nozzle you're using in Cura.

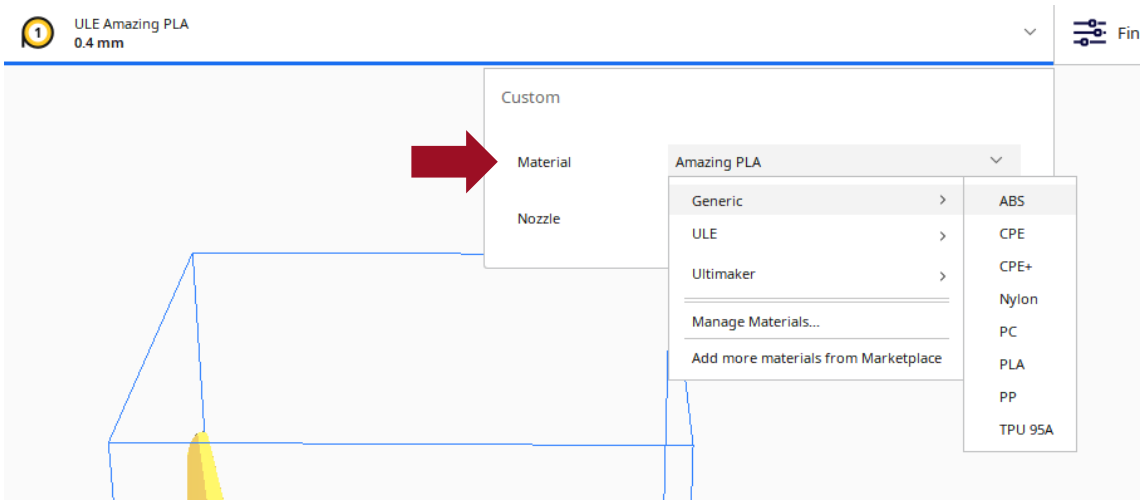
Step 1: **click on material menu**. A submenu is going to pop up, where you can see two sections: Nozzle and Material.



Step 2: **Select the Nozzle size** that you are using.

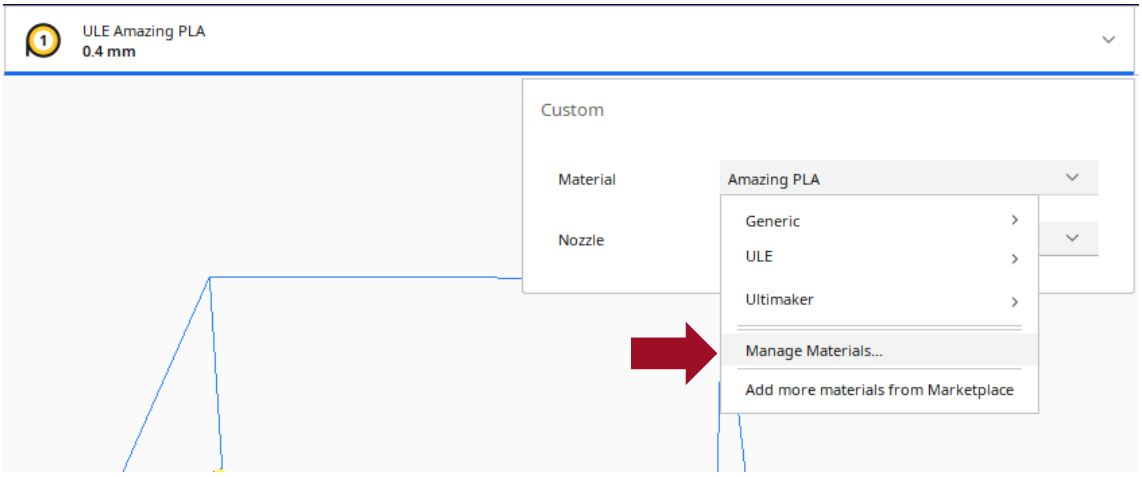


Step 3: **click on material and select** the brand of filament that you are using and the material.

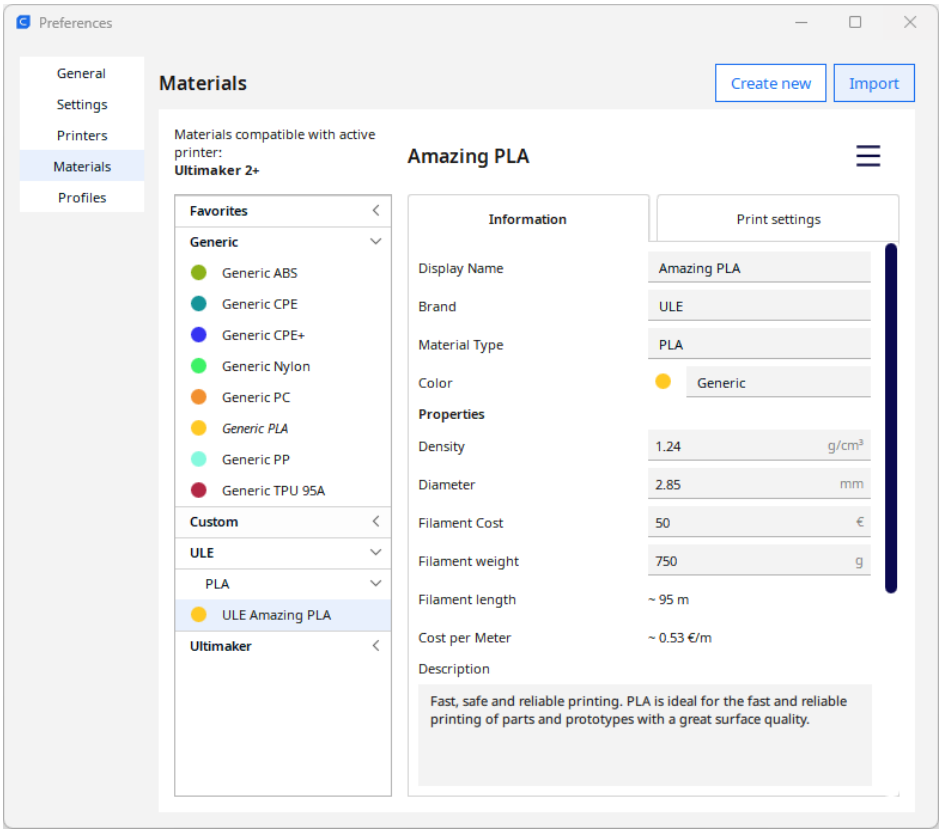


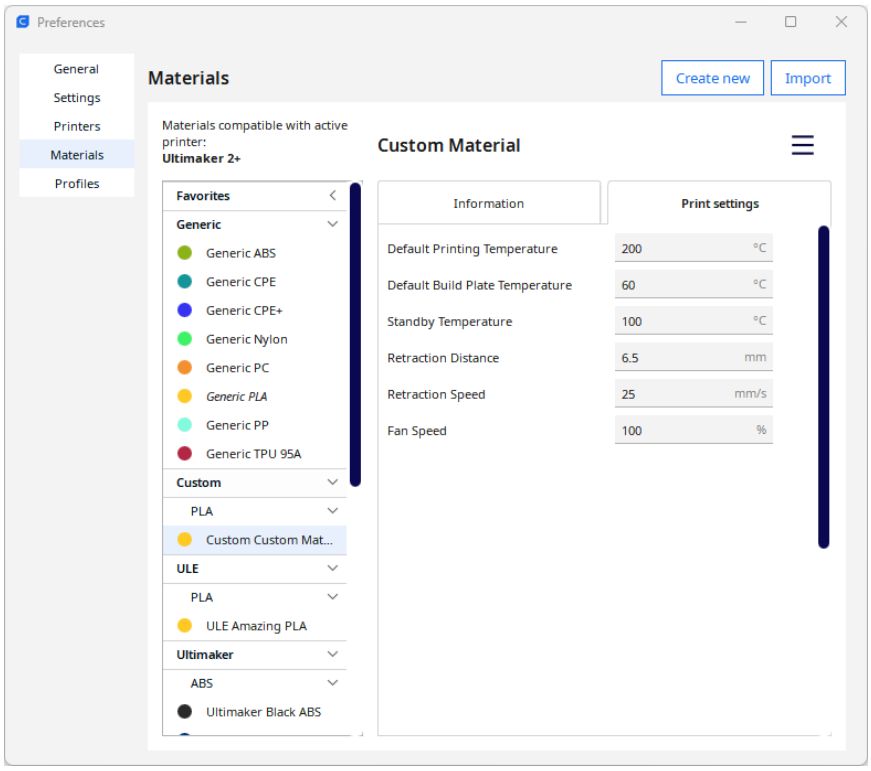
FOR CREATING AND MODIFING PARAMETERS OF MATERIAL (WHEN YOU NEED CUSTOM)

Step 1: click on Manage materials. A new windows will pop up. There we have the materials defined.



Step 2: **click on Create new**. This allows you to modify not only information about the filament but also the print settings such as printing temperature, build plate temperature, etc.



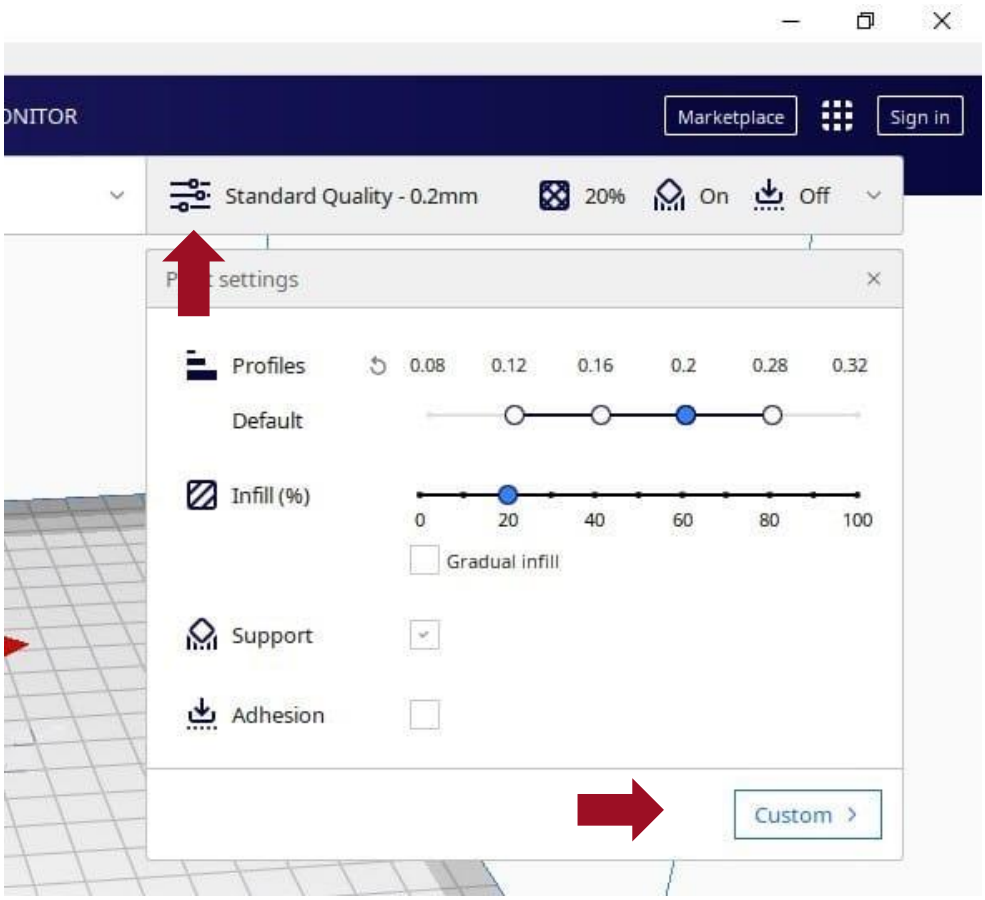


Step 3: Depending on your material, **modify these parameters**.

PART 6: PRINT SETTINGS (PARAMETERS)

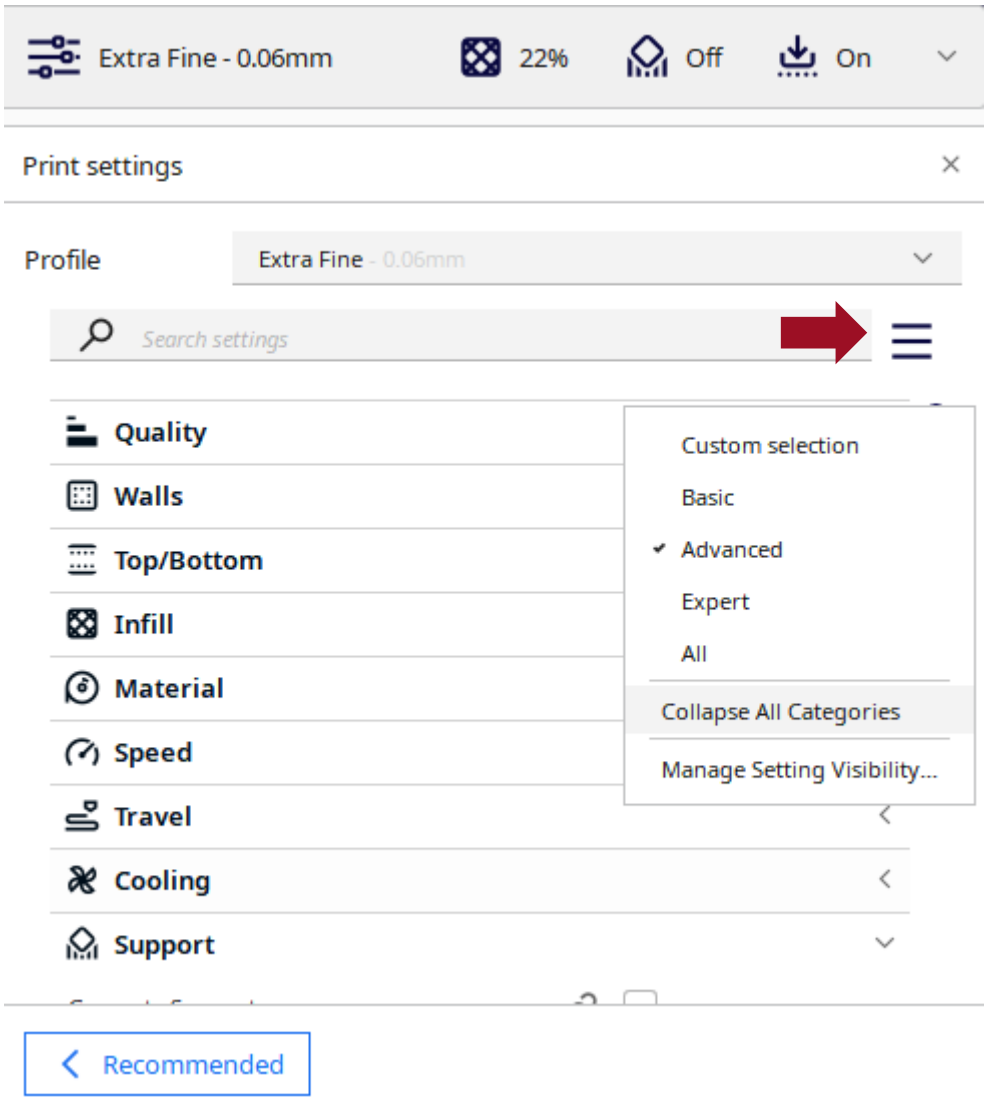
You can customize every aspect of your 3D printing experience in Cura through print settings. However, using all of them at once can be somewhat overwhelming for a beginner. So, for this part we are going to focus on the most used settings and explain their function and how might affect to the printing process.

Step 1: Click on the **print settings menu**. If it's the first time you use it, its going to a appear a submenu like in the following picture. Click the Custom button on the lower right to access more settings and functionality.



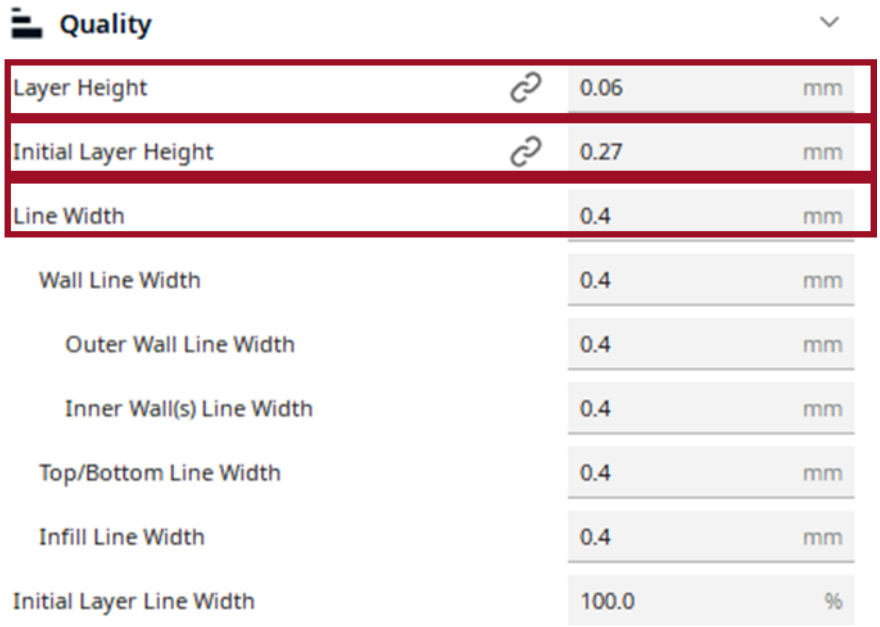
Now we have full access to all the printing options that Cura offers.

Step 2: **Adjust the view of the settings** we want for Cura to show. Click on the three horizontal lines. For this part we are going to select **Advanced**.



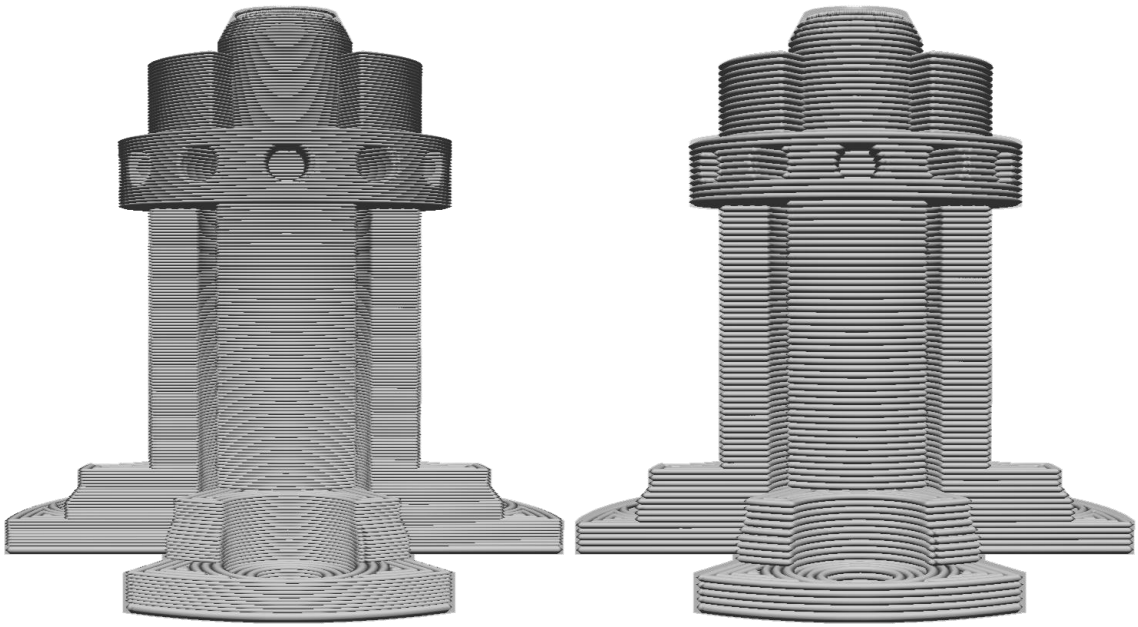
As it can be seen, the settings are divided into categories, so we are going to explain the settings through these categories.

QUALITY SETTINGS



LAYER HEIGHT

The layer height is one of the most frequently changed settings. It is the **thickness** of one **printed layer** in millimeters. Usually, **thinner layers** increase visual quality, give a **smoother surface, and allow more detail**. On the other hand, **thicker layers decrease the print time** substantially. Thicker layers may also give better-quality overhangs.



The profiles in the ***Recommended* mode** are based on a resolution, or **layer height**. The number of available profiles depends on the configuration, meaning the printer, nozzle size, and material(s).

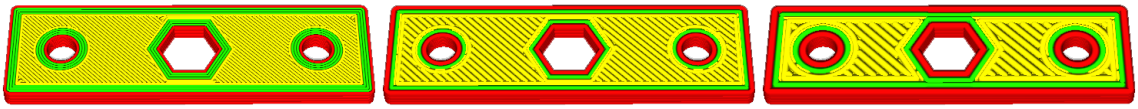
INITIAL LAYER HEIGHT

This setting defines the **height of the first layer** of your print. The initial layer height is **usually thicker than the layer height** to create **a stronger adhesion** with the build plate.

LINE WIDTH

The line width defines the **width of a single printed line**, which should be close to the **nozzle size**. Based on the line width, the extrusion rate is adjusted, which means that it will automatically calculate how much material must be extruded.

Tip: The line width is generally equal to the nozzle size. When customizing this value, it is advised to keep the line width between **75% and 125%** of the nozzle size. Below 60% or above 150% is not advised as this may cause extrusion problems.



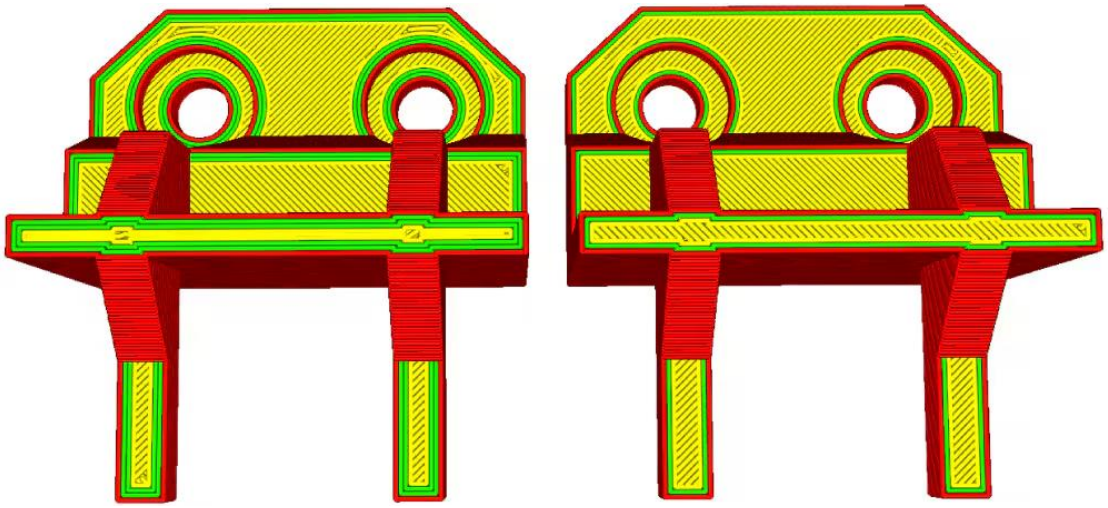
Above, you see the exact same model with line widths of 0.25, 0.4, and 0.8 nozzles.

WALL SETTINGS

WALL THICKNESS & WALL LINE COUNT

This setting adjusts the **thickness of the walls of the model**. Ultimaker Cura rounds the wall thickness to a multiplication of the line width. In general, a wall thickness of two or three times the line width is sufficient. A higher value will create a sturdier model and decreases the chance of leaks, while a lower value can significantly decrease the print time and filament costs.

Instead of setting a thickness in millimeters, you can also set a number of walls. When you set the wall line count, the wall thickness is calculated and will grey out.



Example: The model on the left has three walls, and the model on the right has two walls.

TOP/BOTTOM

TOP/BOTTOM THICKNESS

With the top/bottom thickness you can set **the thickness of the solidly printed top and bottom layers of the print**. A higher value ensures all gaps on the top and bottom layers are closed completely. However, this can also increase the print time and amount of filament used.

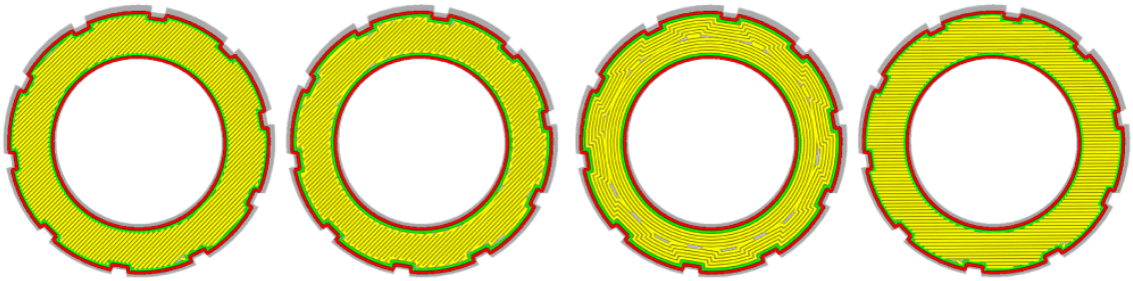
Top/bottom pattern (Expert view set up)

Ultimaker Cura allows you to choose from **different printing patterns** for the top and bottom layers. These are the available patterns:

- **Concentric**: The pattern is printed from the outside to the center of the print.

- **Lines**: A diagonally printed pattern with travel moves on the shell of the model.

- Zig-zag: A diagonally printed pattern with connections on the shell of the model.



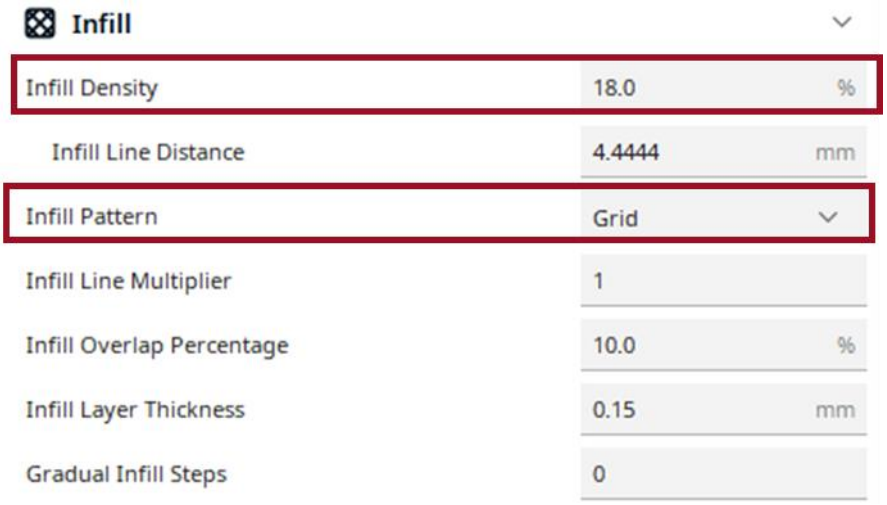
MONOTONIC TOP/BOTTOM ORDER

It is a top/bottom **infill with modified trajectory planning**. The fill lines are extruded from left to right monotonically. This avoids the ridges that form when one printing direction meets the other. This takes slightly more time to print, but makes flat surfaces look more consistent.

ENABLE IRONING

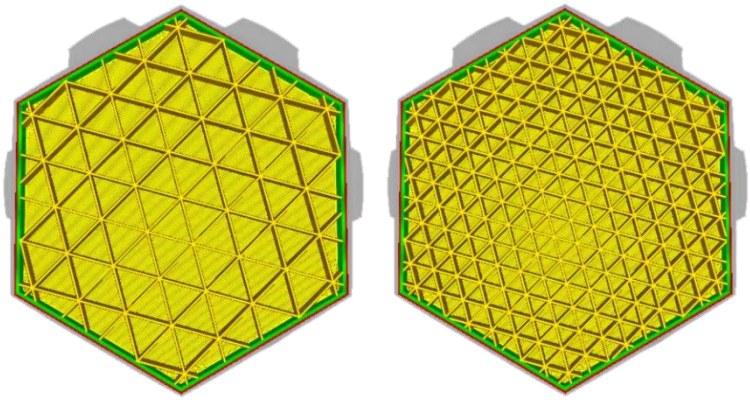
Ironing is a technique where the nozzle travels over the top-most layer after printing it, to iron the top layers to a smooth surface.

INFILL



INFILL DENSITY

The infill density defines the **amount of plastic used on the inside of the print**. A higher infill density means that there is more plastic on the inside of your print, leading to a stronger object. An infill density around 20% is used for models with a visual purpose, higher densities can be used for end-use parts.



The model on the right has a higher infill density than the model on the left

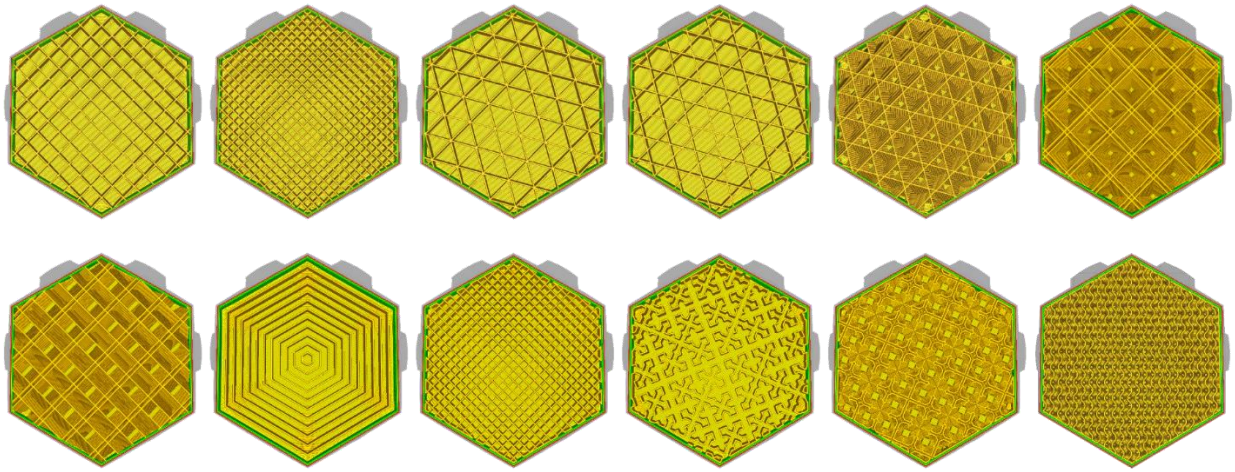
INFILL PATTERN

Ultimaker Cura allows you to change the pattern of the printed infill structure, which is beneficial in some use cases. For example:

- **Strong 2D** infills are used for everyday prints
- **Quick 2D** infills are used for quick, but weak models
- **3D infill**s are used to make the object equally strong in all directions
- **3D concentric** infills are used for flexible materials

The following pattern options are available:

- **Grid**: Strong 2D infill
- **Lines**: Quick 2D infill
- **Triangles**: Strong 2D infill
- **Tri-hexagon**: Strong 2D infill
- **Cubic**: Strong 3D infill
- **Cubic (subdivision)**: Strong 3D infill (this saves material compared to Cubic)
- **Octet**: Strong 3D infill
- **Quarter cubic**: Strong 3D infill
- **Concentric**: Flexible 3D infill
- **Zig-zag**: A grid shaped infill, printing continuously in one diagonal direction
- **Cross**: Flexible 3D infill
- **Cross 3D**: Flexible 3D infill
- **Gyroid infill**: Infill with increased strength for the lowest weight
- **Lightning**: Infill that is extremely fast to print and only supports top surfaces

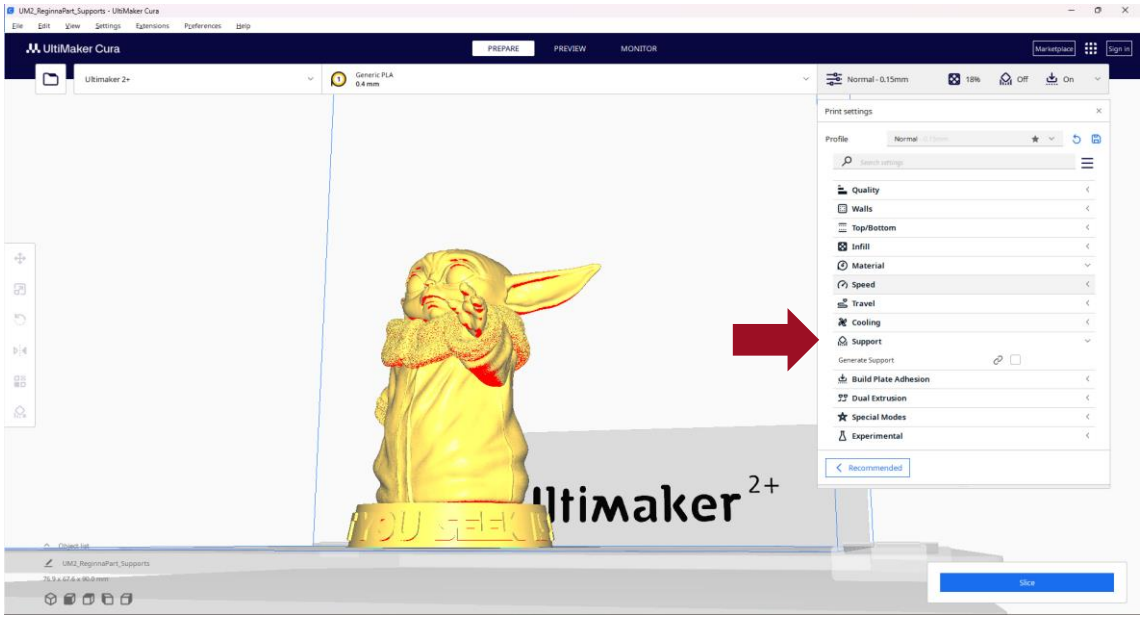


PART 7: SUPPORTS AND BUILD PLATE ADHESION

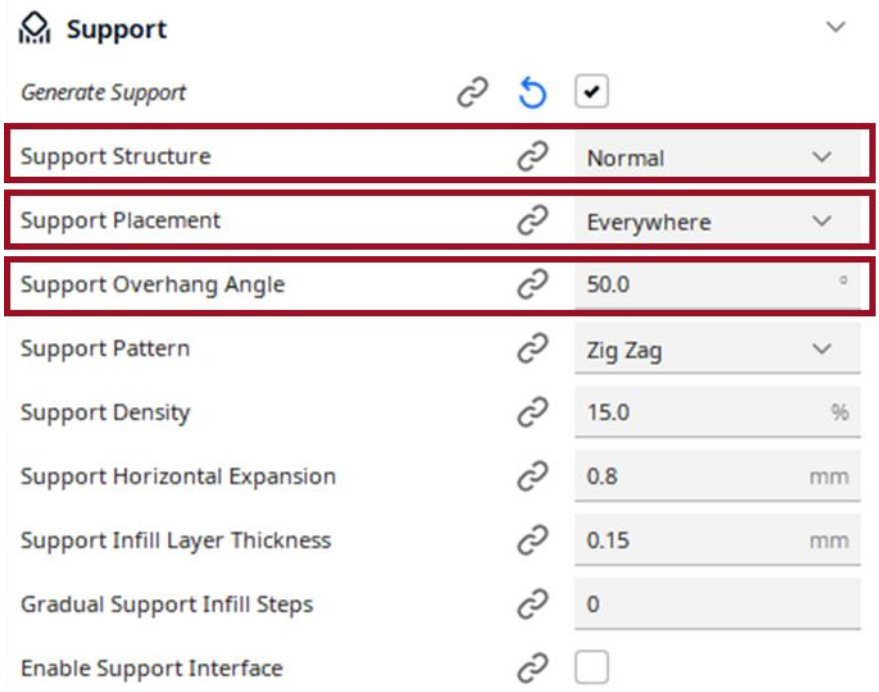
For this part we are going to need the other file so, click on the folder icon and select the part with the name of: *ReginnaPart_Supports*.

When the file is loaded we are going to see that some parts of the model are red. That indicates that will need supports to be printed correctly.

Step 1: Go to Print Settings Menu and click on supports.



Step 2: Click on Generate Supports. This allows you to generate the support structures.

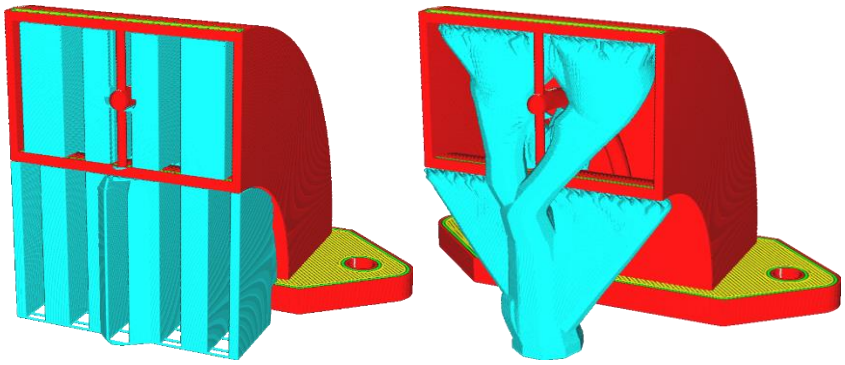


SUPPORT STRUCTURE

UltiMaker Cura offers two techniques to generate support: **Normal and Tree**. Both of these structure types have their own advantages and disadvantages.

Normal support generates a straight structure underneath the areas of the model that need support. This is quite sturdy and has large contact areas, to offer maximum support. However, this can cause more scarring on the model (unless a soluble support material is used). This technique uses a lot of material and takes a long time to print.

As an alternative, **Tree** support creates branches that grow around your 3D model. The benefits of this alternative include reduced print time and less scarring of the model. However, due to its complex structure, it takes significantly longer to slice and it is not suitable for all materials.



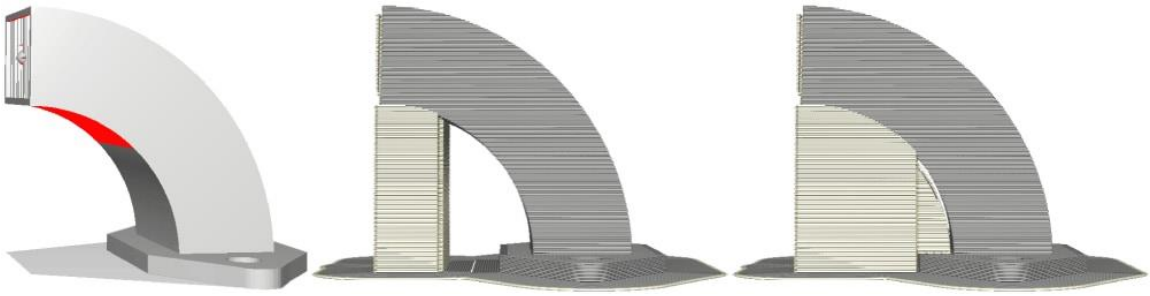
SUPPORT PLACEMENT

This setting **defines where the support structure is printed**. It contains the following options:

- **Touching build plate**: Support material is only printed from the build plate up.
- **Everywhere**: Support material is printed below every part that needs support, which means that it can also be placed on or inside a model.

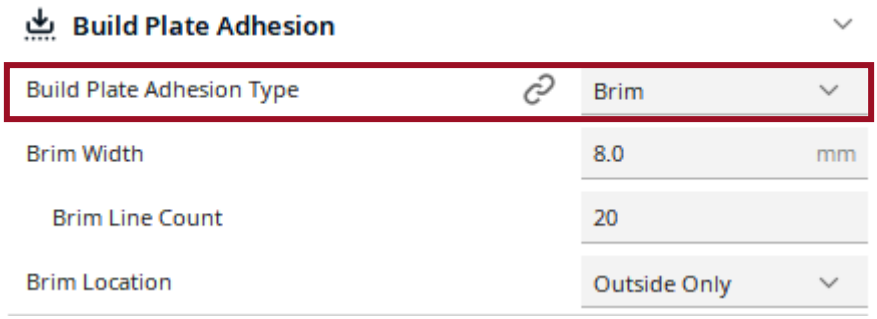
SUPPORT OVERHANG ANGLE

The overhang angle influences **how much support material is added and where**. A smaller angle leads to more support. For example, at a value of 0° all overhangs are supported, while at 90° no support material is added.



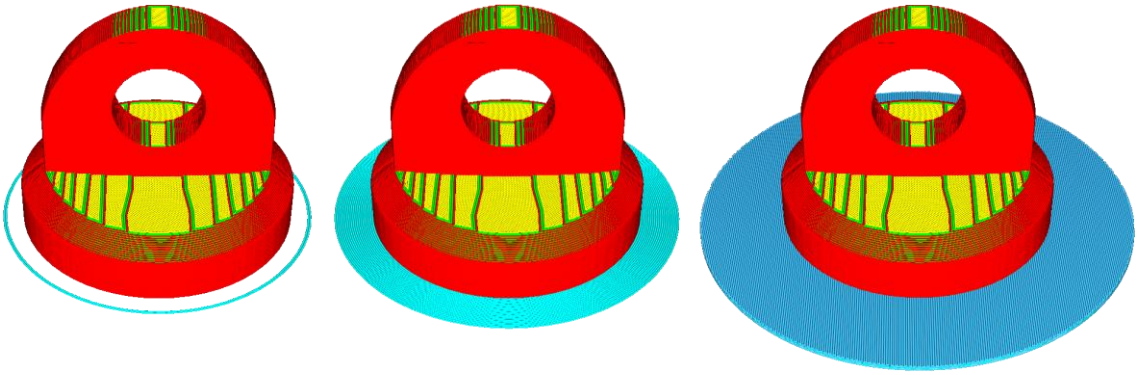
The image on the left is a model shown in the solid mode. The red areas indicate the overhang area that needs support. The model in the middle has an overhang angle of 70°, while the model on the right has an overhang angle of 45°.

Step 3: Define how it is going to be adhered to the printing bed.



There are three types of build plate adhesion: *Skirt*, *Brim*, and *Raft*. You can also easily disable the adhesion types by selecting *None* (not recommended)

- **Skirts** are great for simply priming your nozzle and leveling your bed for larger models.
- **Brims** are great for adding some adhesion to your models without using too much material.
- **Rafts** are great for adding a lot of adhesion to your models, reducing warping on your models.

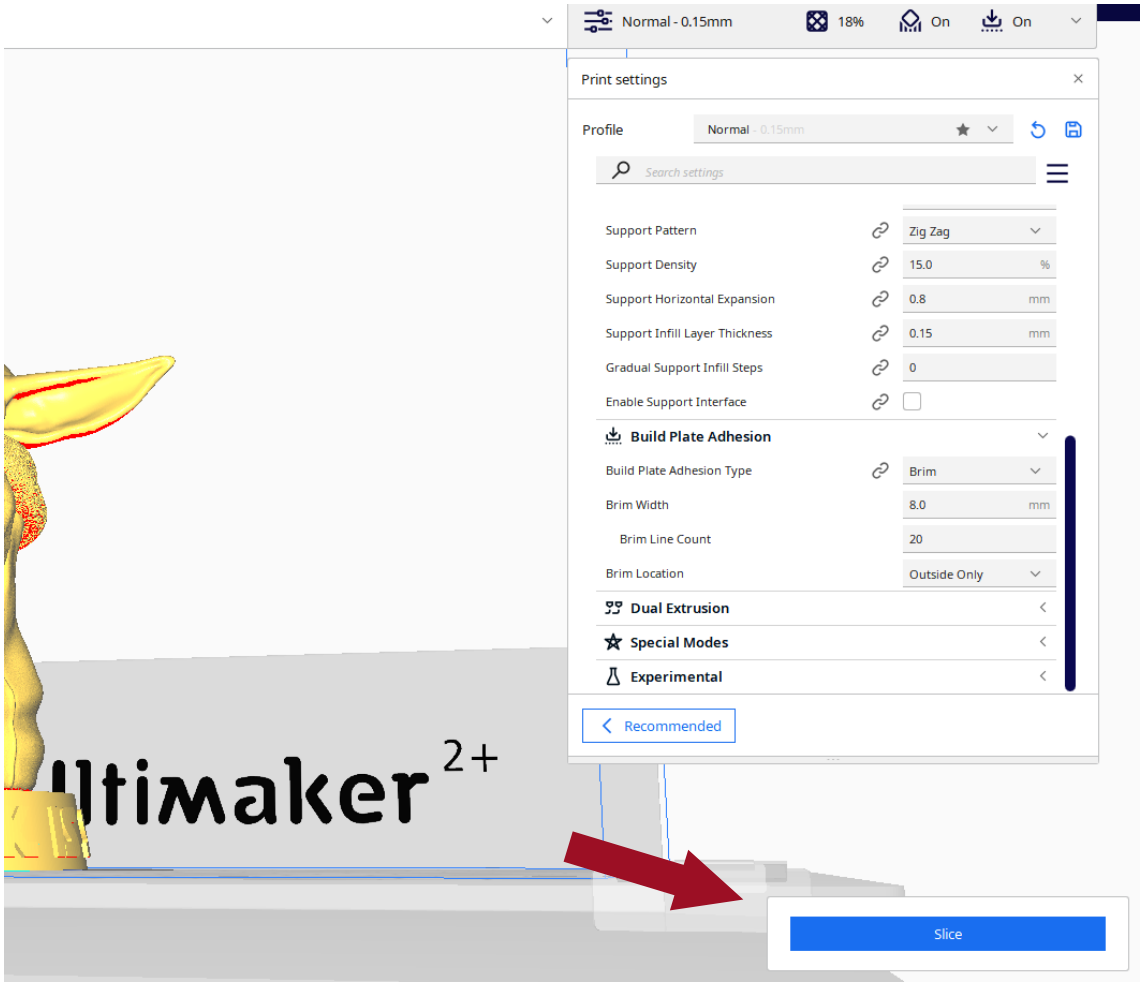


From left to right: skirt, brim and raft.

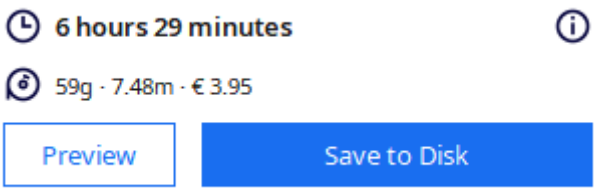
PART 7: SLICE AND PREVIEW

Once you optimized all the settings correctly, it is time to send the model to your printing. To do so, first we need to slice.

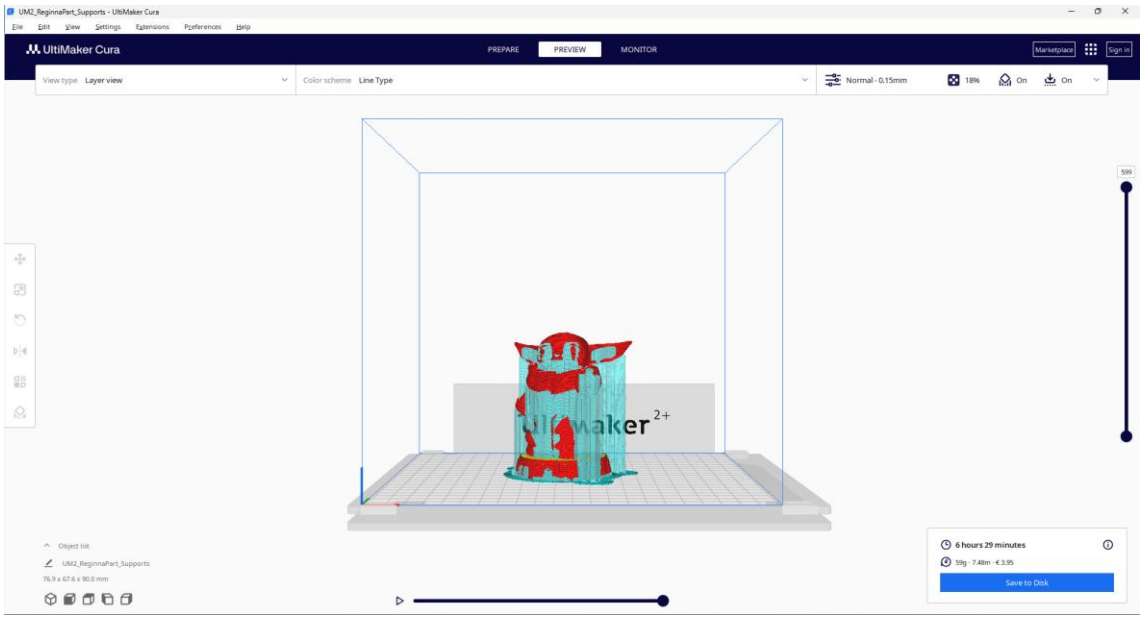
Step 1: Click on the slice buttom.



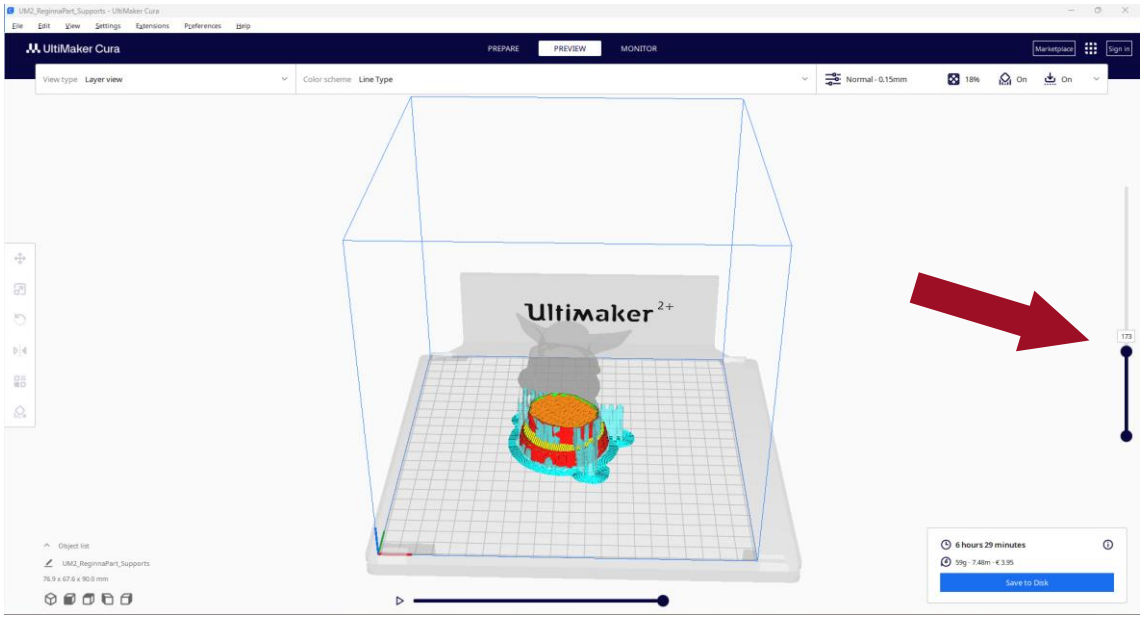
It will slice the model and show you a preview of the print, the amount of material it will use, and the printing time.



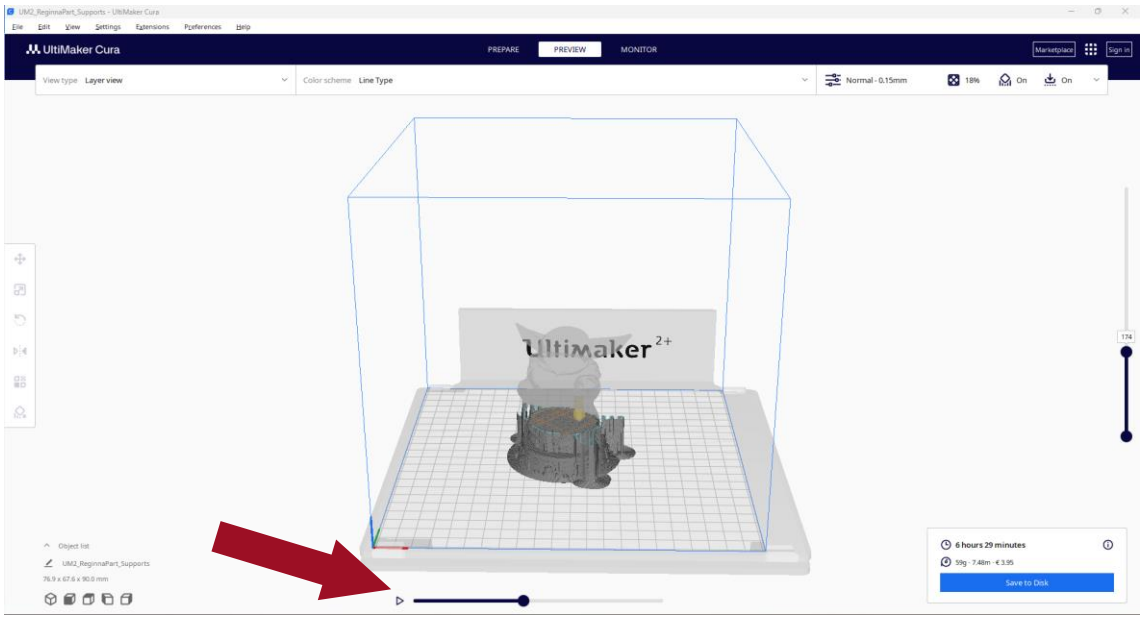
Step 2: click on preview to see a simulation how is going to be printed your model.



Also, we can slide the right bar, and watch layer by layer how it is going to be manufactured.



If we slide the bottom bar we can see which trajectory the nozzle will follow in the selected layer.


UltiMaker Cura
PREPARE
PREVIEW
MONITOR
Ultimaker 2+
6 hours 29 minutes
Save to Disk