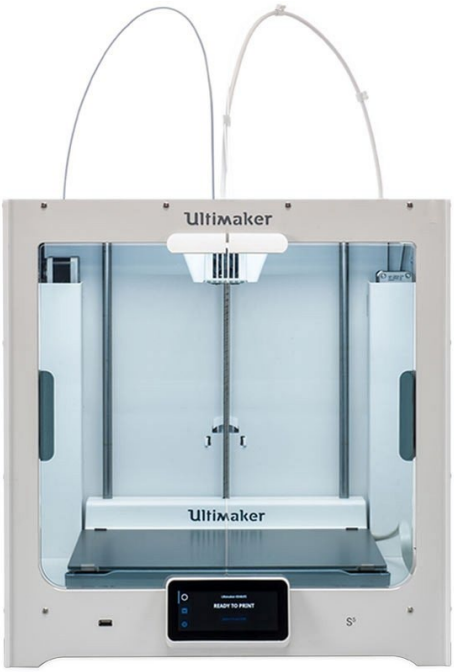
3D printer

UltiMaker 3 & S5





institut
FRANÇAIS
de la
MODE

Table of Contents

FabLab safety rules and good practices **3**

Safety in 3D printing **3**

Principle of FDM 3D printing **4**

Description of the 3D printer **5**

1. Printer schematic 5
2. Features 7

Preparation of the 3D model **8**

Setting up the printer **8**

1. Choice of print cores (print media) 8
2. Loading of materials 9
3. Preparing the tray 11

Parameterization of 3D printing or "slicing". **12**

1. Presentation and environment of Cura 12
2. Using Cura 15
   a. Material selection 16
   b. Principals parameters (quality, strength, supports, adhesion) 18
   c. Others important parameters (temperature, speed, travel, cooling, …) 21
   d. Slicing 24

3D printing **27**

1. Start printing 27
2. During printing 28
3. After printing 28

Cleaning and storage after printing **31**

1. Machine cleaning 31
2. Storage 31

Annexe **32**

1. Knowing the filament length remaining on your spool 33
2. Rhino instructions for a good 3D print file 35

FabLab safety rules and good practices

- Closed shoes are recommended.
- A Wear appropriate clothing for handling machines and tools (no loose clothing, no long jewellery, tie your hair if necessary).
- Never use a machine without first being trained on it.
- Be fully aware of your actions.
    - Do not use machines if you are tired or ill.
    - Do not drink alcohol or take drugs before using the machines and tools.
- Do not distract or surprise other users while using the fablab's machines and tools.
- Never leave a machine running unattended, use only one machine at a time.
- Warn the fabmanageuses in case of.danger.
- Never open a machine, warn the fabmanager.s present in the area for any technical problem related to the machines (malfunction, breakage, etc.).
- Keep the work area clean and tidy after use.
- Store materials and tools in their place.
- Have prepared your files before taking up the machine position.
- Do not force the machines : ask for help, fabmanageuses are there for !
- Switch off the machines after use.
- Do not eat in the Fablab.

Safety in 3D printing

The 3D printer is a machine whose platen and print head are heated to relatively high temperatures in order to shape the material.

The main danger of this machine is burn injury.

- **Never touch the print head (especially the nozzle) while it is heating, printing or cooling. The temperature can reach 280°C.**
- **Do not put your hands on the platen while it is heating, printing or cooling. The temperature can reach 100°C.**

The 3D printer includes belts to move the print head and platen. The force used to move its components can injure the user.rice in case of misuse.

- **Never put your hands inside the machine when it is in operation..**

Principle of FDM 3D printing

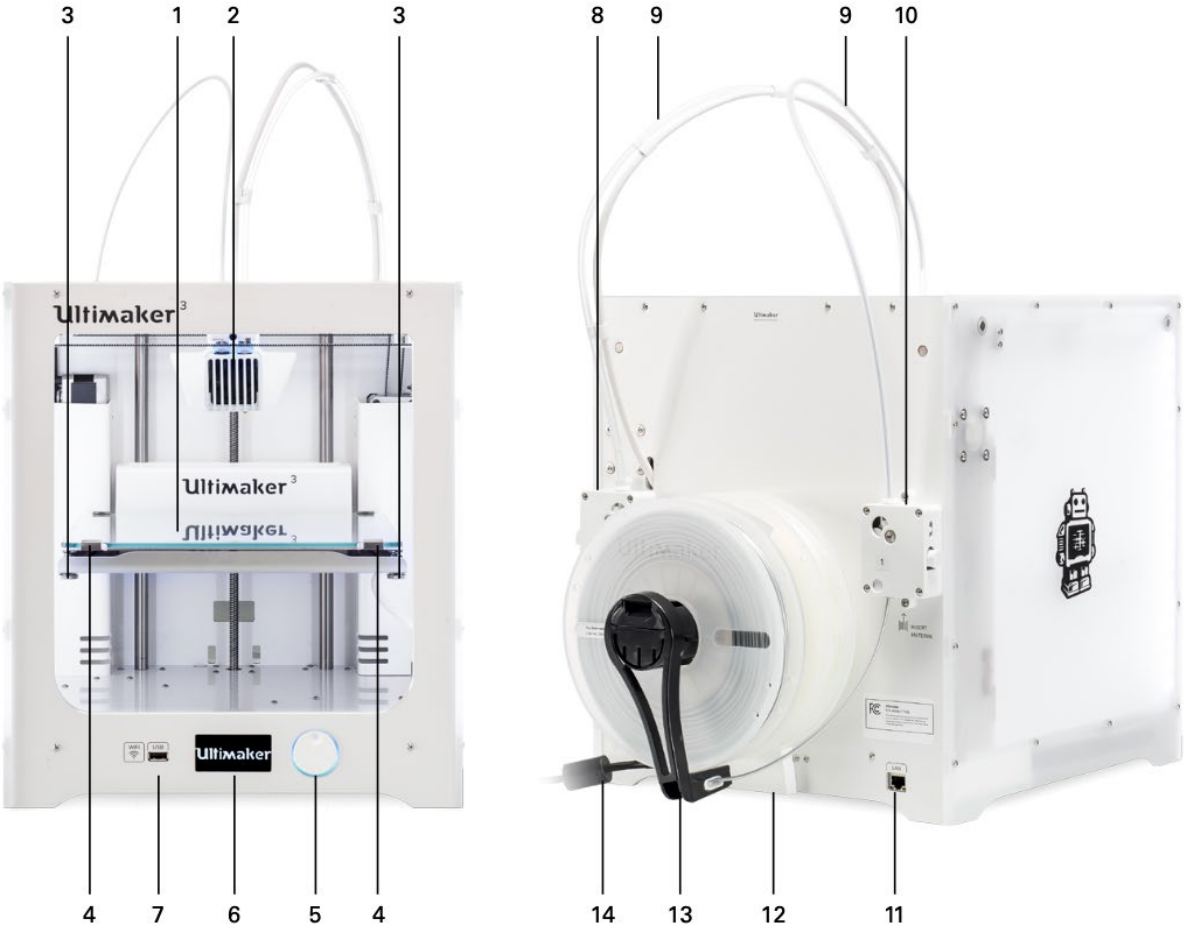
UltiMaker 3 and **S5** 3D printers use the so-called "wire deposition" or FDM (Fuse Deposition Modeling) technique. This is one of the most common additive manufacturing processes.

It is called additive manufacturing because it involves depositing an extruded thermoplastic material by layering. The extrusion of the material is carried out by means of the heating body and nozzle contained in the printhead, as well as the toothed wheels contained in the feeder. The material, in filament form, is driven by the toothed wheels and guided to the printhead by a Bowden tube. Once it reaches the print head, it is heated to its melting temperature (liquefaction of the solid material) and passes through the nozzle to be deposited on the platen.

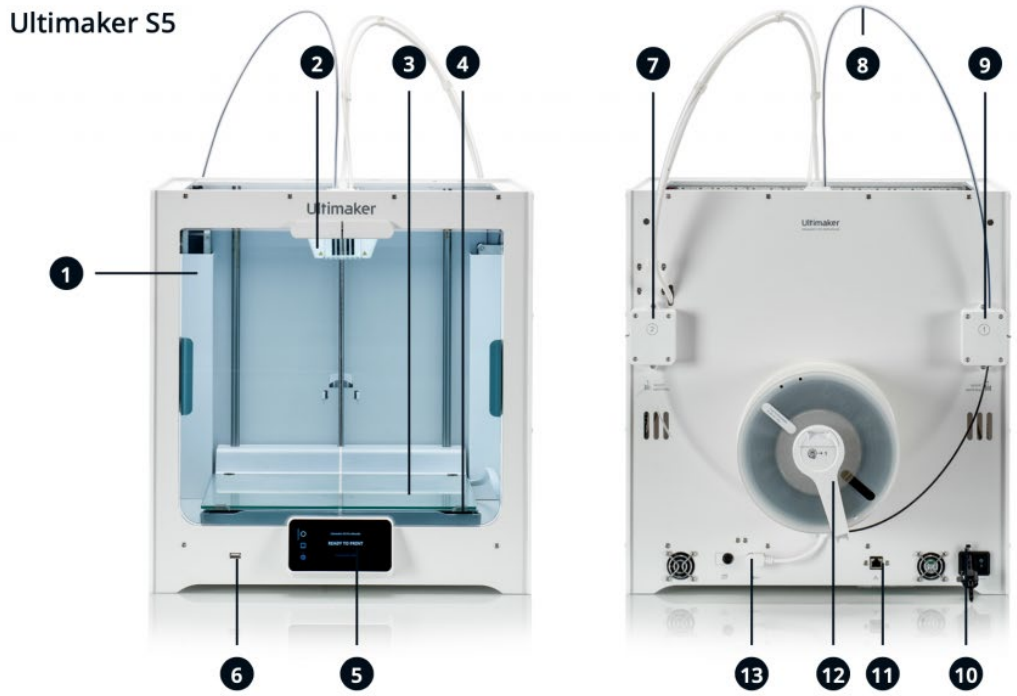
During printing, the machine deposits the material by moving the print head along the X and Y axes of the platen forming a first layer. It is the addition of the layers one on top of the other along the Z axis that allows the formation of a volume object (3D). Once depose is completed, the material fuses with the previous layer and solidifies as it cools.

Description of 3D printers

1. UltiMaker 3 and S5 Printers Diagram



1- Heated tray
2- Double print head
3- Screws of the plate
4- Spindle of the tray
5- Push button/rotary button
6- Screen
7- USB port
8- Wire charger n°2
9- Bowden tubes
10- Wire charger n°1
11- Ethernet port
12- Cable cover
13- Double reel holder with NFC cable
14- Power supply cable



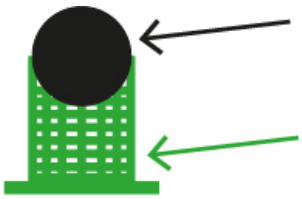
1 - Glass door
2 - Print head
3 - Build plate
4 - Build plate clamps
5 - Touchscreen
6 - USB port
7 - Feeder 2
8 - Bowden tubes
9 - Feeder 1
10 - Power socket and switch
11 - Ethernet port
12 - Double spooler holder with NFC cable
13 - NFC socket

2. Features

UltiMaker 3 and **S5** are a dual extrusion printers, meaning that they can use both nozzles on the extrusion head simultaneously to :

- **Print 1 piece with one material + one carrier material**

Example :

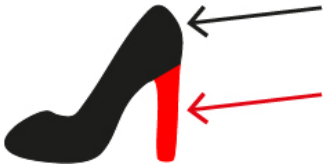


One material to print the final object, like PLA, TPU, …

One carrier material to print supports for your final object, like PLA or PVA.

- **Print 1 piece with 2 different materials (two-material effect)**

Example :

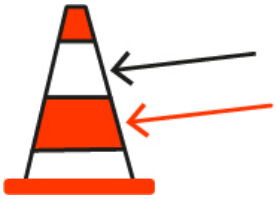


One material to print the soft part of your object, like TPU.

One material to print the rigid part of your object, like PLA.

- **Print 1 piece with 2-colour material (2-colour effect)**

Example :



One material to print the white areas, like white PLA.

One material to print the orange areas, like orange PLA.

Technical information :

- **UltiMaker 3** Printing dimension : W 197 x D 215 x H 200 mm
- **UltiMaker S5** Printing dimension : W 330 x D 240 x H 300 mm
- Printing materials: PLA, ABS, CPE or PETG, Nylon, TPU or TPE (flexible), PP, PVA (soluble carrier material).
- Filament diameter : 2.85mm
- How it works : **Blue LED** when the print head is **cold**, **Red LED** when the print head is **warm**.

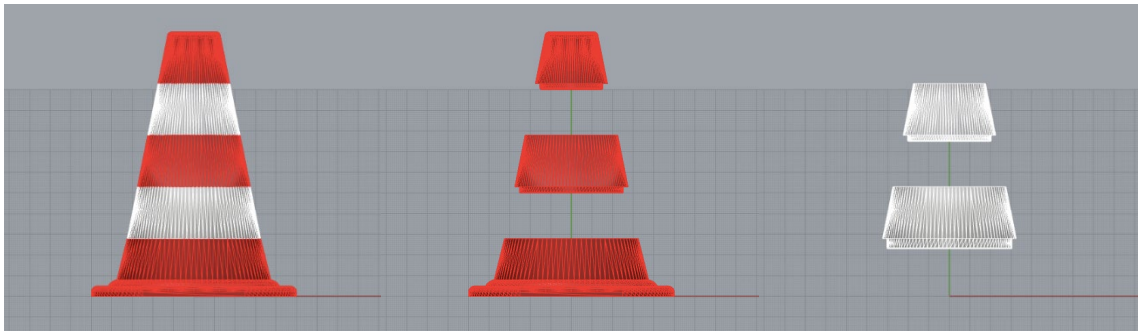
Preparation of the 3D model

A 3D print can only be made from a 3-dimensional model prepared in advance with a modelling software (Rhinoceros, 3DSMax, Fusion 360, Blender, etc.).

The modeling software used does not matter, however it is essential that the format of the 3D model is in**. STL**.

! The printing of a single bi-material or bi-colour part must be made from 2 3D models, registered separately in. STL.

Tip : In the 3D modelling software, save the complete model first (part 1 + part 2), then save each part in 1 independent .STL file, deleting one part and then the other (do not change the position of each part).



Setting up the printer

1. Choice of print cores (print media)

The **UltiMaker 3** and **S5** allows you to print **2 materials at the same time**. It is possible to print 1 carrier material and 1 building material or to print 2 building materials in different colours.

Before starting a print job, the user.rice must know which materials are going to be printed and therefore which print cores (print media) will be required.

How to choose ?

- Type **AA print cores** can be used to print **PLA, ABS, CPE and Nylon**.
- **AA print cores with an "F"** on top are the only ones to use with flexible filaments like **TPU**.
- **Type BB print cores** can be used for **PVA**,
- **The black print core or CC type** is for **mixed material** like PLA with ceramic, metal, egg shell, … .

! Once the print cores have been determined, the user should ask the fabmanageuse to install them in the machine if they have not already been installed.

The combinations recommended by Ultimaker for double printing are : PLA - PVA, Nylon - PVA, PLA - PLA, PLA - TPU (TPE), ABS - ABS, CPE - CPE.

In the case of single printing, the user.rice must ensure that the **print core type AA** is already installed in the machine.

2. Loading of materials

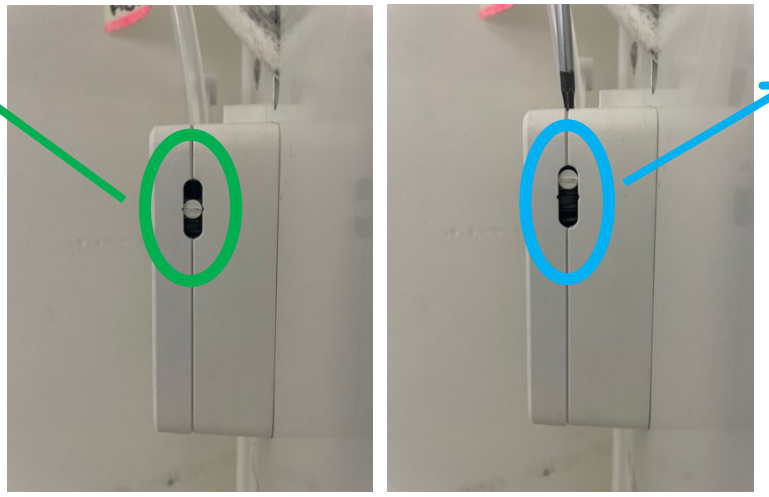
Before starting a print job, the user must load the material(s) selected for printing into the machine.

! Attention, in the case of dual material printing, ***material 2 should always be installed before material 1.***

- **CHECKING THE FEEDER TENSION.**

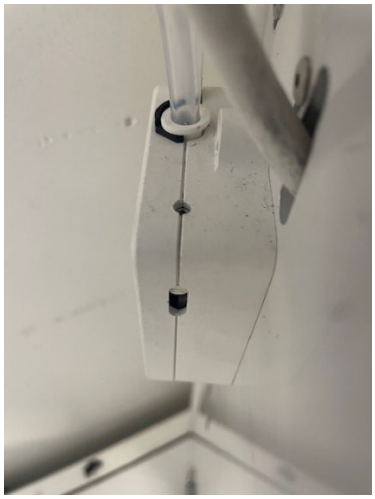
Before loading any filament, it is important to check and, if necessary, **adjust the tension on the feeders** you are using. This allows you to reduce or increase the pressure on the filament to feed it into the print head.

Push the marker's tension in the middle of the gauge when using **rigid materials.**



Push the marker's tension on top of the gauge when using **flexible materials.**

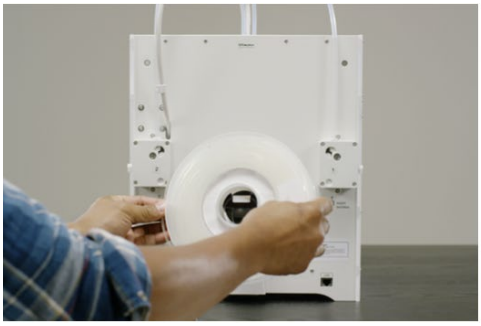
To change the position of the maker feeder tension :


- Use the appropriate small screwdriver,
- Check for the small hole on top of the feeder,
- Fasten or unfasten the screw inside the small hole, depends if you're using flexible or rigid filament.

Loading material 2 :

- **Place the roll** with the material **on the roll holder**. Be sure to position it with the material placed **clockwise**, so that the material can enter Feeder 2 from below. Slightly straighten the material so that it can easily enter the feeder.
- **Wait** until the **UltiMaker 3** or **S5** has detected the material (if using Ultimaker material).
- **Insert** the end of the **material into the feeder 2** and gently push it until it is caught by the feeder and is visible in the Bowden tube.
- **Wai**t for the **UltiMaker 3** or **S5 to heat up the print core 2** and **load the material** into the print head.
- **Confirm** when the new **material extrudes** from the print head.
- **Wait** for the **print core 2 to cool down** for a few moments.




Loading material 1 :

- **Take the material guide** and hold it with the outside facing you.
- **Place the roll of material on the material guide** with the material **counterclockwise**.
- **Guide the end of the material** into the hole in the material guide. When you are finished, select **Continue**.
- Place the material guide, with **material 1 on top**, on the roll holder **behind material 2**, and wait for it to be detected by the printer.
- Insert the end of the material **into Feeder 1** and gently push it until it is caught by the feeder and is visible in the Bowden tube.
- **Wait** for the **UltiMaker 3** or **S5 to heat up** the print core 1 and load the material into the print head.
- **Confirm** when the new material extrudes from the print head.
- **Wait for the print core 1 to cool down** for a few moments.



3. Preparing the tray

The platen of **UltiMaker 3** and **S5** are heated platens which improves the adhesion of the thermoplastic material to the printing surface. The glass plate is placed on a movable part (up/down movement) made of metal and held in place by metal pins.

! It is recommended to wash with soap the glass plate to improve the adhesion performance and cancel the "warping" (lifting of the material).

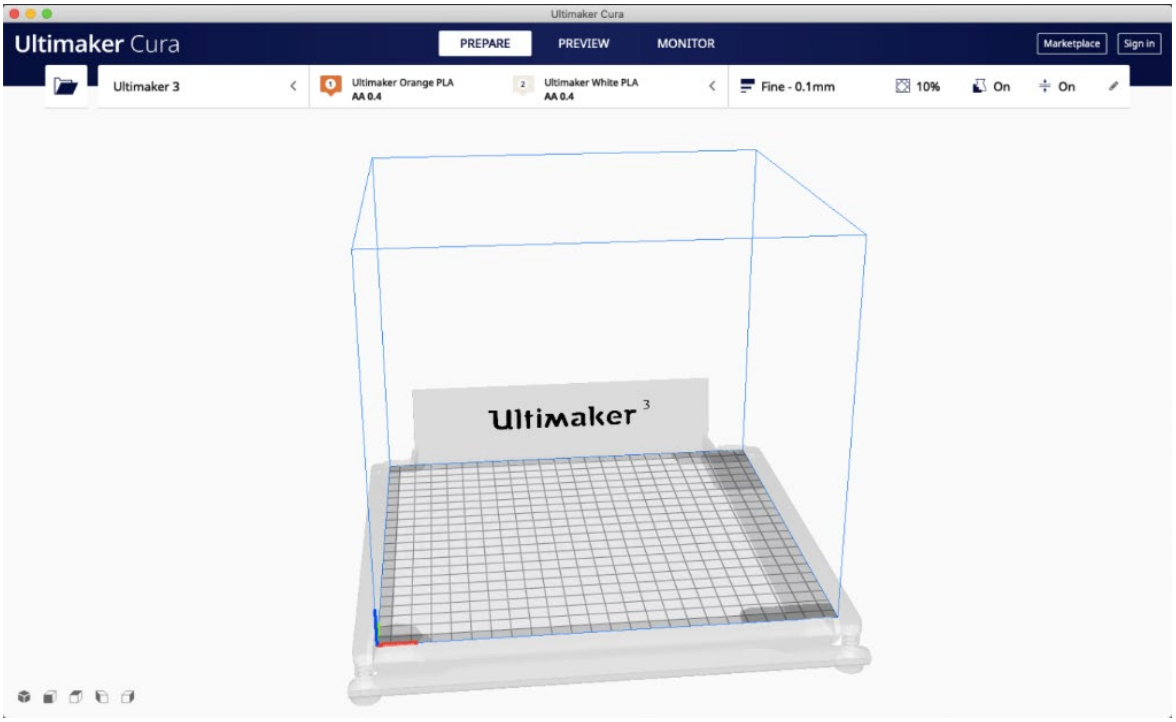
Parameterization of 3D printing or "slicing".

The print setting corresponds to the passage of the 3D model file in. STL to the file that contains data that can be printed by the machine, **the G-code file**.

“Slicing" is the operation that consists **of translating a volume model into a succession of textual information (code).** The information contained in the **G-code** provides the machine with precise instructions such as movement coordinates, layer thickness, movement speed, heating temperatures, etc. The G-code file contains the information required for slicing.

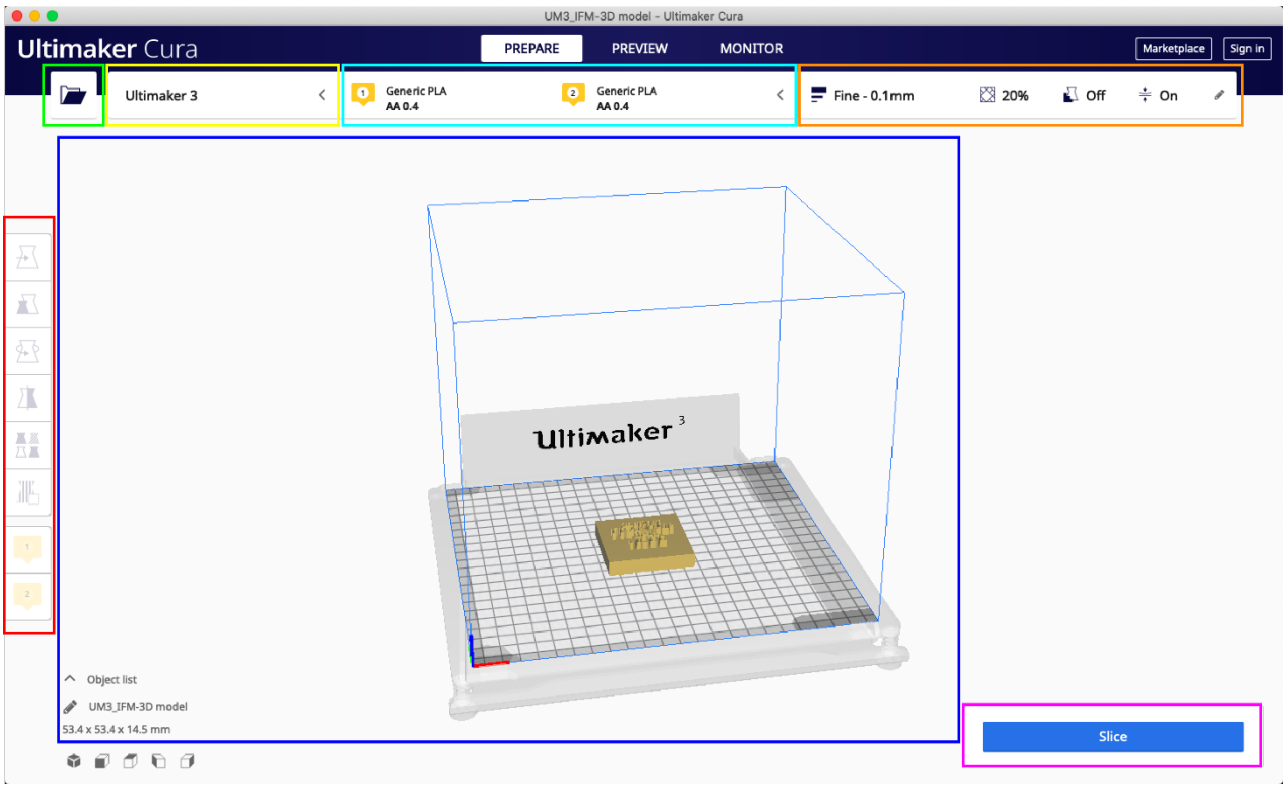
1. Presentation and environment of Cura

For **UltiMaker 3** and **S5** printers, the printing or slicing settings are made with the **Cura software**. **Cura** is a software developed by UltiMaker, but free of rights, which can be installed free of charge on PC, Mac and Linux.



PREPARE tab

This tab gives access to all the fields that will allow the user to prepare a printout. From the choice of the 3D model to the precise parameterisation of the print, including the types of materials, it is essential to go through this step to make a successful print.



Open a file

Printer selection

File information and viewing of the part

Tools for modifying the part

Selection of printing materials

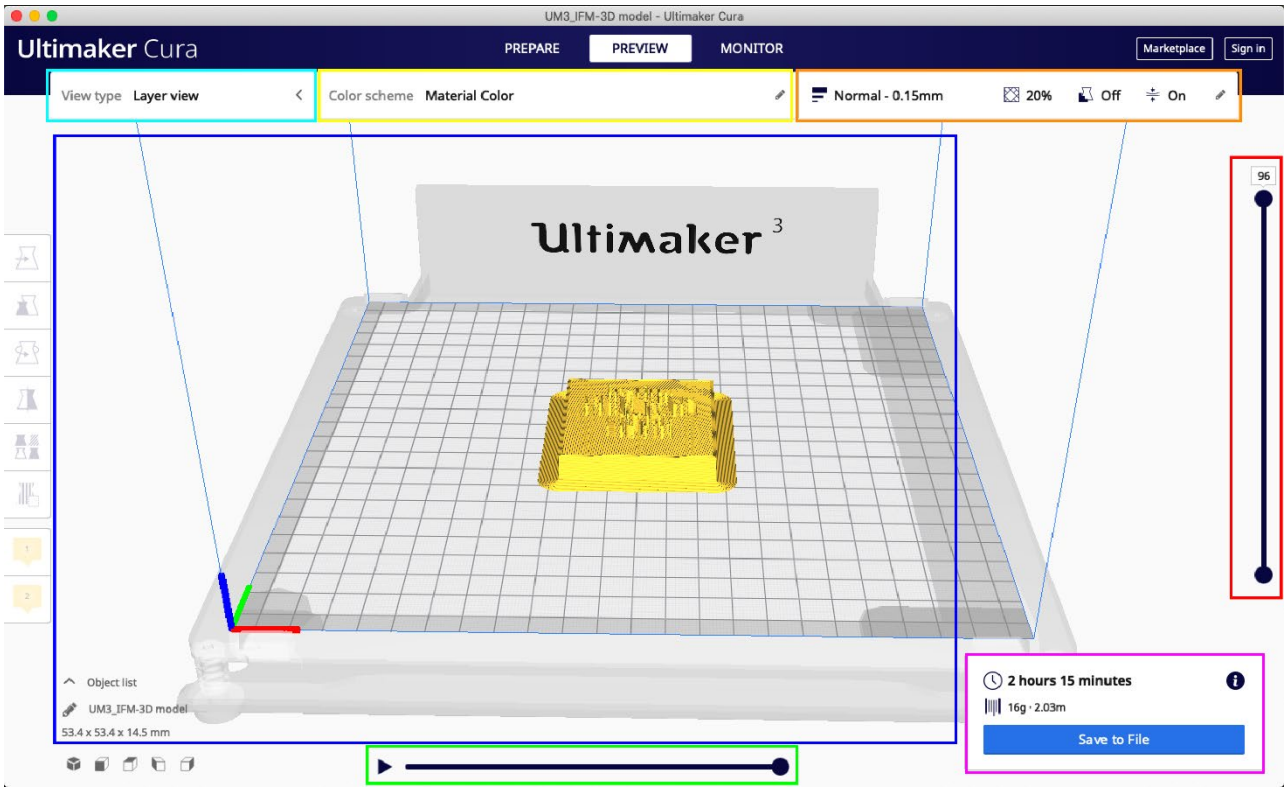
Print settings

Slicing / slicing operation

PREVIEW tab

In this tab, the user can **simulate his/her printout** and **obtain several important information** about it.

By observing the print simulation**, it is possible to predict the time needed to manufacture the part, anticipate errors, avoid omissions or even improve the results** of a printout in a search phase.



Type of view of the simulation

Type of room structure diagram

Print settings

File information and viewing of the part

Layer-by-layer print status frieze

Print simulation reader

Printing and registration information

2. Using Cura

*! If you are printing in **two materials** (manufacturing material/soluble carrier material or manufacturing material1- manufacturing material2) or **two colours**, special features apply to the following steps.*

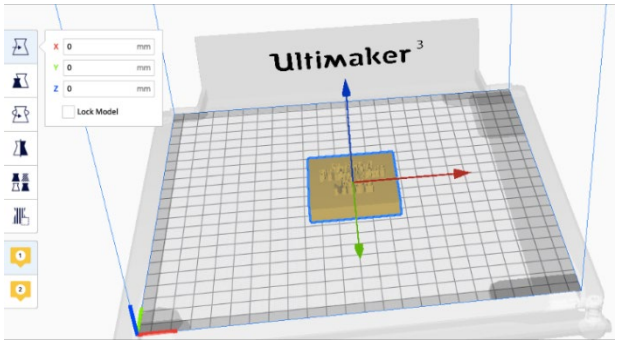
- **IMPORT THE .STL FILE** to be printed by clicking on the icon or by dragging the file directly into the Cura window. The piece will appear in the viewing area.

  *! For a **bi-material** or **bi-colour print**, you must have **2 .STL files**: one for each material or colour. Import the 2 files directly by dragging them into the Cura window.*

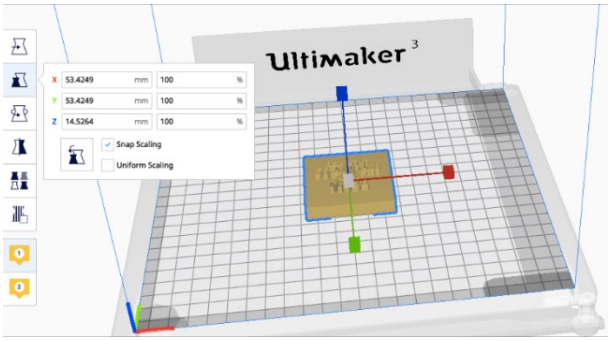
- **SELECT THE PRINTER** you have prepared for printing.

- **If necessary, MODIFY** some characteristics of the part using the tools in the left panel :

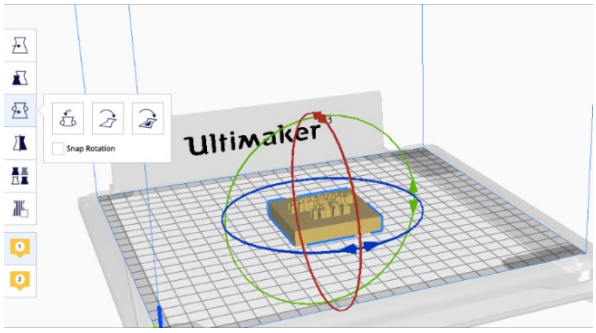
Move over the printing area



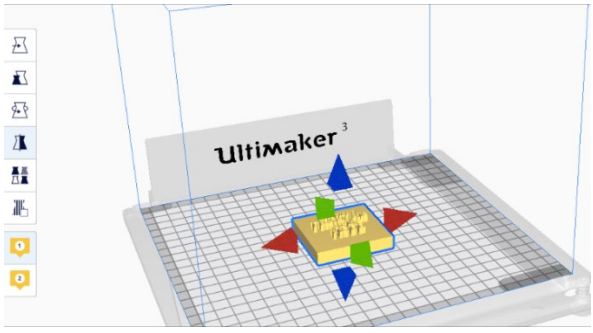
Change of scale or proportion



Rotation

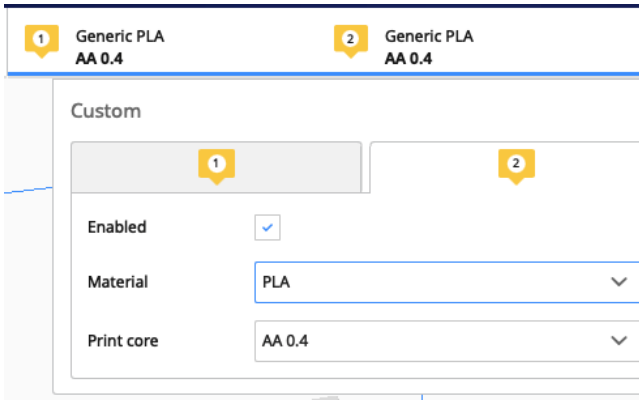


Symétrie (miroir)



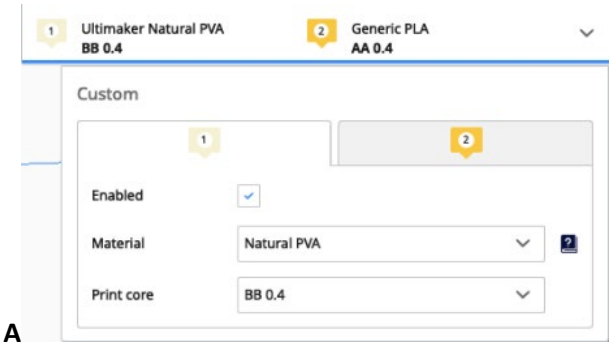
- **SELECT THE MATERIAL(S)** to be printed in If you are printing with **only one material, select location 2** (check that the filament is inserted in feeder 2).

Caractéristiques des matériaux 1 et 2 :



- **Enabled** must be ticked to use the material. If you only want to print with Material 2, you must clear the Enabled check box for Material 1.
- **Select the correct material** from the Material drop-down menu.
- **Select Print core** AA 0.4 or BB 0.4 depending on the material(s) selected.

A

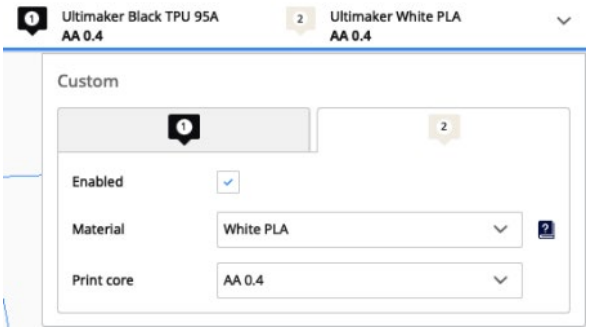


! If you print in bi-material or bi-colour.

*<u>Example **A** :</u>*

*- Material 1 = **PVA** (carrier material), print core type **BB 0,4***
*- Material 2 = **PLA** (manufacturing material), print core type **AA 0,4***

B

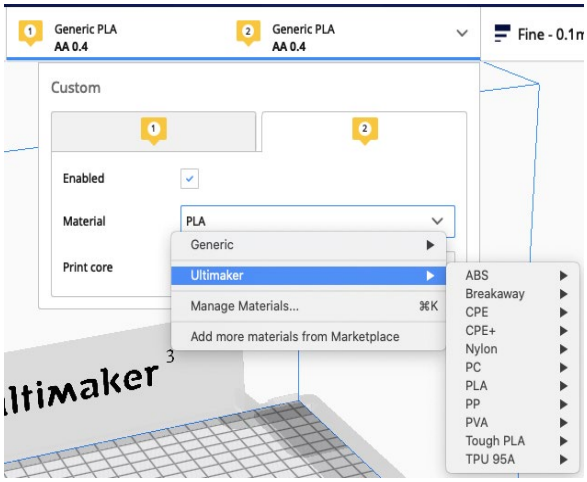


*<u>Example **B** :</u>*

*- Material 1 = **TPU** (flexible manufacturing material), print core type **AA 0,4***
*- Material 2 = **PLA** (manufacturing material), print core type **AA 0,4***

Material selection :

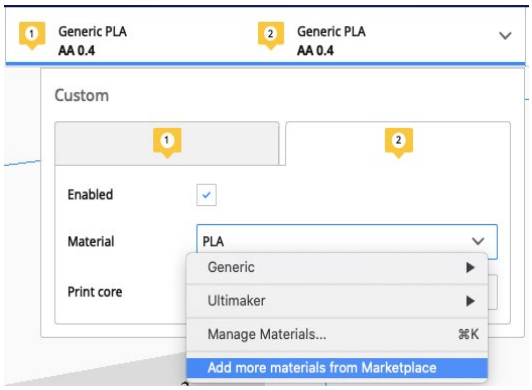
OPTION A : Your filament is from UltiMaker



- **If the filament you are going to print with is UltiMaker, you must select UltiMaker** from the drop down menu and then choose the exact material.

  ! *An UltiMaker PLA and a PLA of another brand may react differently to identical print settings. UltiMaker has established optimised profiles containing the best possible settings for printing the UltiMaker PLA (same for other materials).*

OPTION B : Your filament is from another brand

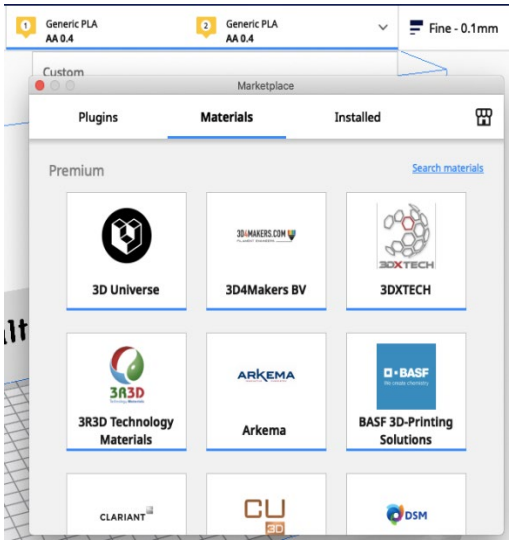


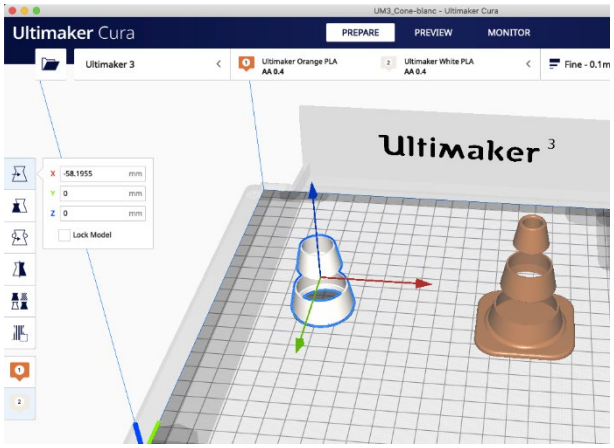
- **If the filament you are going to print with is not UltiMaker**, you can **add the material from the Marketplace** (free of charge) by clicking on Add more materials from Marketplace.

  ! *Other filament manufacturers have also established profiles for their material types and contribute to the database integrated in Cura.*

  You can search for the manufacturer of your filament.

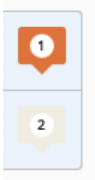
  In the manufacturer's file, search for the material number and click on Install so that it appears in the initial drop-down menu.

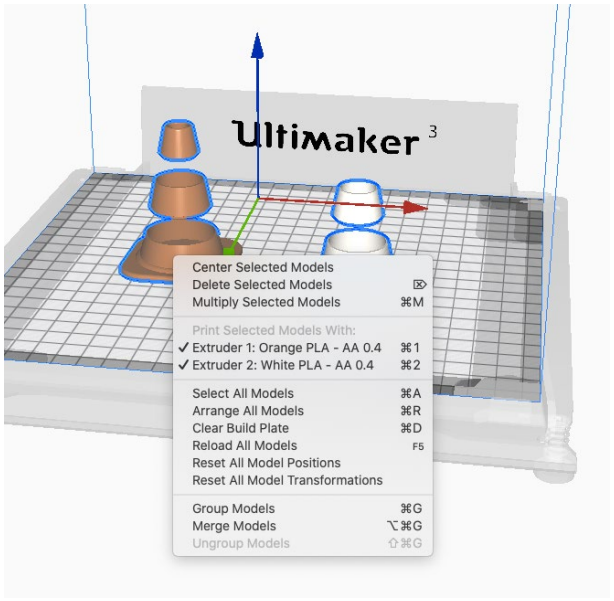




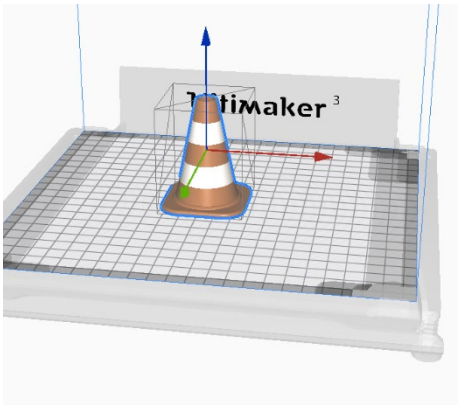
! If you print in bi-material or bi-colour.

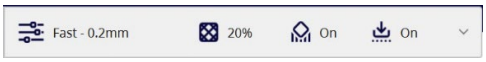
It is necessary to assign a material to each part by clicking on buttons 1 and 2 at the bottom of the left toolbar.





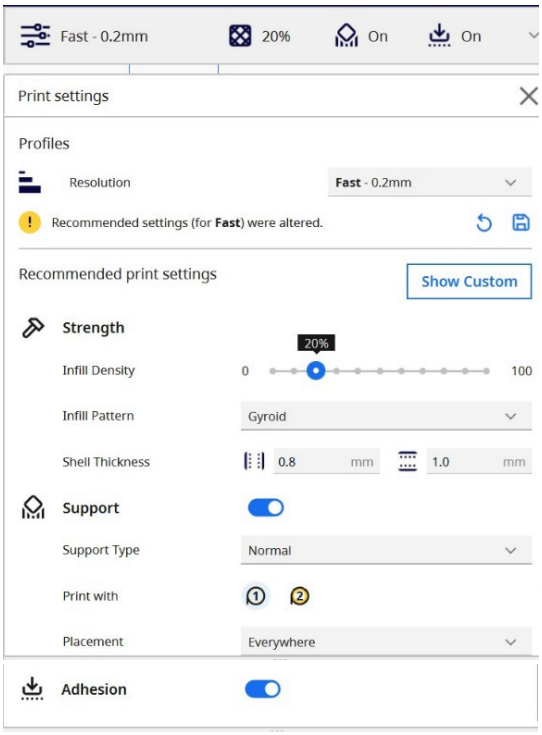
*Then select both parts. Right click and choose **Merge Models**.*
The two parts of your object are assembled.



- **PARAMETER THE PRINTING** by clicking on the tab 

access **the principals parameters** of a 3D printing:

1. **Layer thickness,**
2. **Part filling rate / Strength**
3. **Substrates / Supports,**
4. **Adhesion.**



- ***Default Profiles / Layer thickness / Print Quality :***

Defines the height of each layer of the print. The layer height determines the print **resolution.**

The thinner the layer is, the more precise the part will be, but the printing time will be longer.

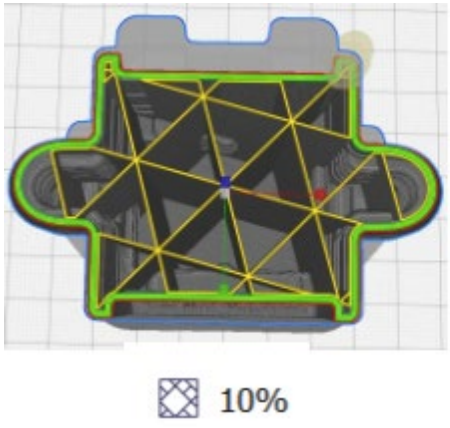
The thicker the layer is, the less accurate the part will be, but the printing time will be shorter.

- ***Strength* :**

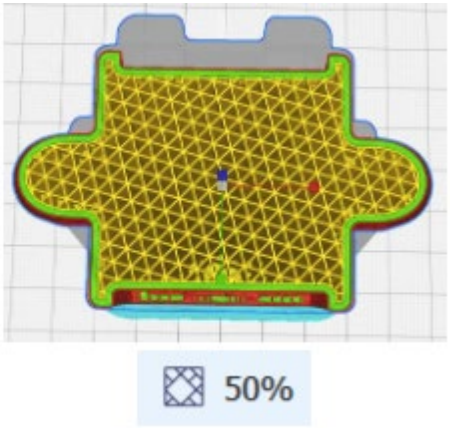
Defines the fill rate of the part.
The part fill corresponds to the holding structure of the part on itself.

➔ **Choose the right Infill Density**
The more the part is filled, the heavier the part will be and the longer the print time will be.

! A more filled part will not necessarily be stronger and more resistant than a less filled part.



+ fast
+ light
- solid



+ slow
+ heavy
+ solid

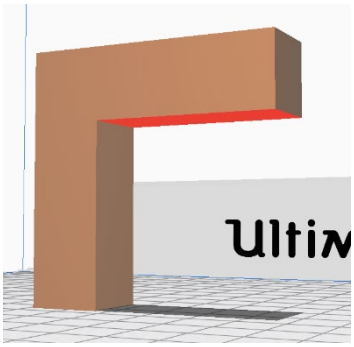
Gradual infill allows the room to be filled gradually: at the bottom it will be less filled, at the top it will be more filled (like arches in architecture).
This option allows you to save material and reduce printing time while maintaining a good fit.

➔ **Choose the right Infill Pattern**
Some patterns are stronger than others, as Grid, Triangles and Cross which permite you to get more solid objects.

➔ **Shell Thickness**
Don't forget to check also the thickness of your object walls, top and bottom. You can increase them to get a more rigid object.

- ***Support :***

An option that allows Cura **to generate media when printing where the room contains overhangs** with an angle of incidence greater than 60 degrees. The media is only **used during printing** (such as scaffolding during construction) **and will be removed after printing.**



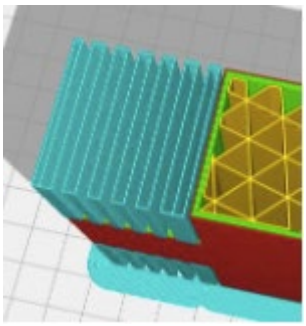
Cura indicates by **red areas when media is required** for printing.
In this example, the overhang is 90 degrees, so media is required.

➔ **Choose the right Support Type**
For the Ulitmaker 3 and S5, please choose **Norma**l supports. Tree supports do not print well on these machines and may cause the print to fail.

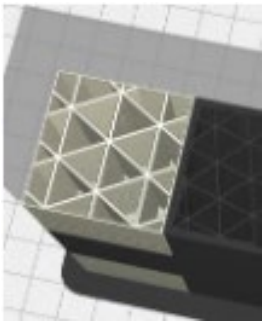
➔ **Print with …**
Select the right Feeder and Printcore you want to print your supports with.

➔ **Placement**
Select the number of supports you need to ensure high print quality.
"Touching build plate" is suitable for simple objects that require supports at the base or in areas touching the build plate. In this case, make sure there are no complex areas higher up in the object's print, otherwise the supports will not be sufficient to build it. If you are printing more complex objects with holes and many angles, choose **"Everywhere"** instead.

! *The media can be made of different materials. Depending on the expected result and the constraints, it is possible to create medias with the same material as your final part (PLA, ABS, CPE, PP, etc.) or medias with the specific soluble material (PVA).*



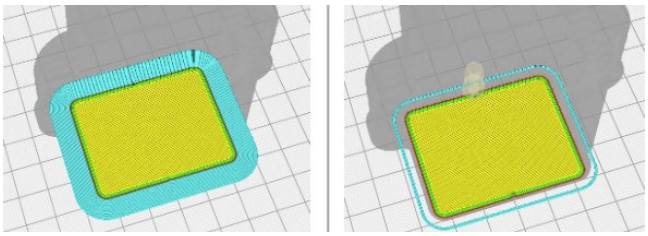
Media with the same rigid material



Media with soluble material

- ***Adhesion*** **:**

  Option that allows Cura to **generate a base on the piece so that it adheres better to the tray** and is easier to peel off after printing.
  This base acts as a raft or brim base for printing.



- ***Custom*** **:**

  Access button for advanced settings.
  ! *For the discovery of 3D printing and the first tests, it is not recommended to modify the advanced settings.*

- **OTHER IMPORTANT PRINTING PARAMETERS :**

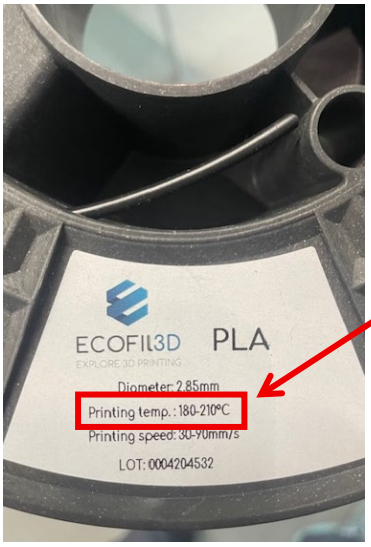
1. **Material / Temperature,**
2. **Speed,**
3. **Travel / Retraction,**
4. **Cooling / Fan Speed,**
5. **Interface,**
6. **Dual Extrusion.**

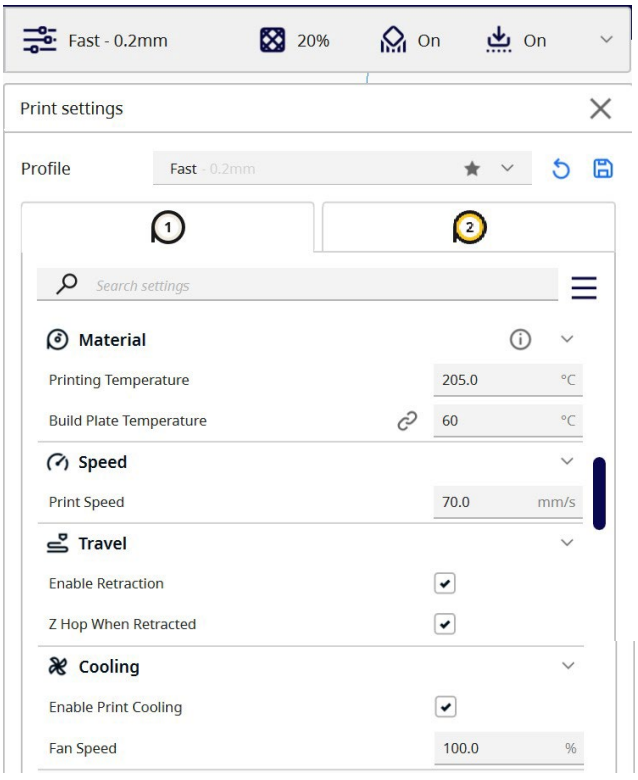
- ***Material :***

Defines the nozzle and build plate temperature**. These temperatures depends of the recommendations and materials you're printing with.** After checking the recommendations for your material you can increase or reduce the build plate and nozzle temperature to get more adhesion**, within the specified range.**

For the material recommendations you can check on the filament brand's website or directly on the spools :







- ***Print Speed :***

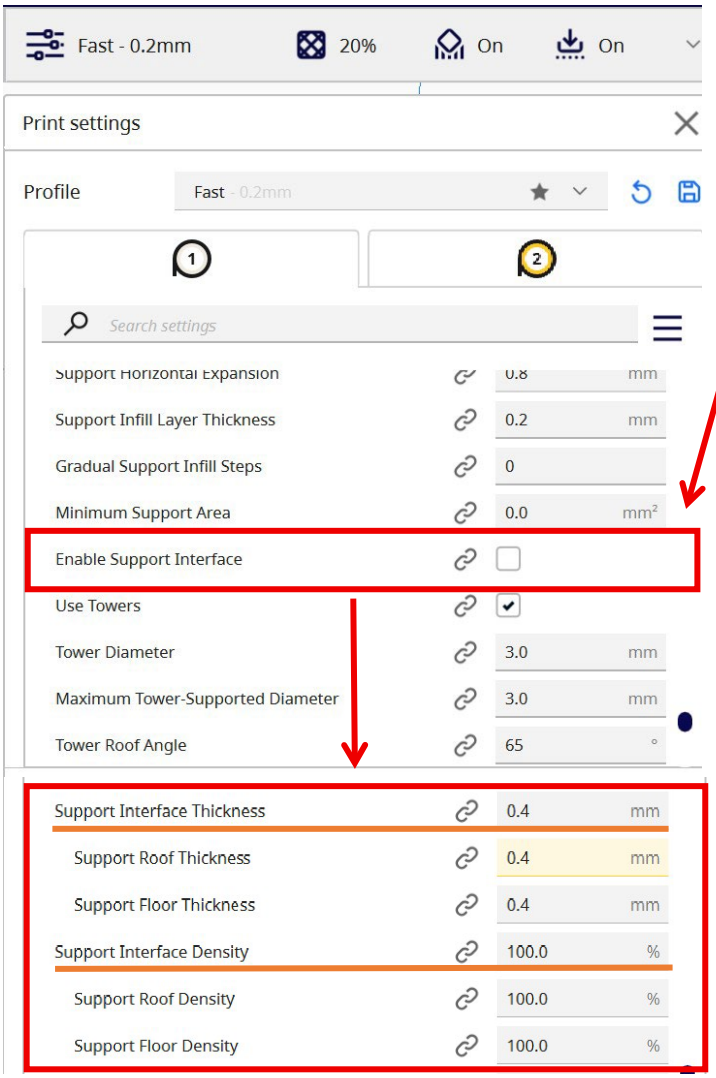
Sets the speed at which **the nozzle will move to deposit the molten filament onto the bed**. If your print failed you can also play with the speed and reduce it to ensure the good quality of your print.

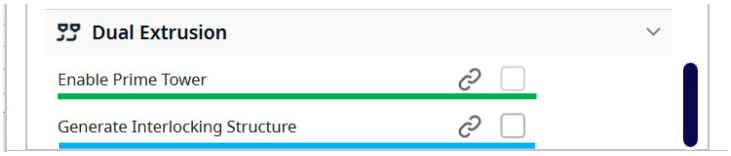
- ***Travel / Enable Retraction :***

Permite the retraction of the filament when the nozzle is moving over a non-printed area to avoid residues.

- ***Cooling :***

Enabled the print cooling fans while printing. The fans improve print quality on layers with short layer times and bridging / overhangs.





- ***Interface :***

Inside **Supports** options, in the **Expert mode**, you can play with Interface settings. Check **Enable Supports Interface** to access to the parameters.

! Those settings are important to look at when you need to print with supports or a carrier material, to remove them easily.

➔ ***Support Interface Thickness :***

Sets the thickness of the interface of the support where it touches with the model on the bottom or the top.

➔ ***Support Interface Density :***

Adjusts the density of the roofs and floors of the support structure. A higher value results in better overhangs, but the supports are harder to remove.

- ***Dual Extrusion :***

Gives you tools to get a good result with printing with two different materials

➔ ***Enable Prime Tower:***

Print a tower next to the print which serves to prime the material avec each nozzle switch.

➔ ***Generate Interlocking Structures :***

Improves the adhesion between models, especially models in different materials.

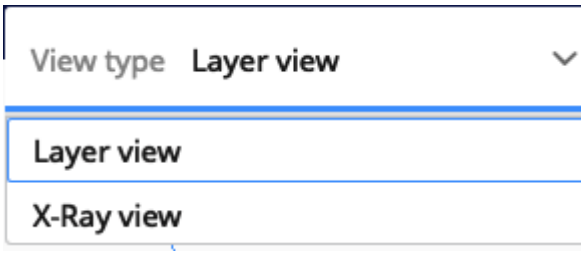
- **SLICE THE SUBJECT** by clicking on the button 

  Only after clicking on Slice will you know the print time and filament consumption for the parameters you have just determined.

  ! *As soon as you change the materials used or one or more print parameters, you must click on Slice to apply the changes.*

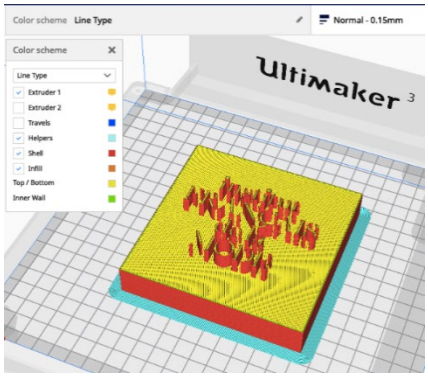
- **ANTICIPATE** your printing thanks to the tab 

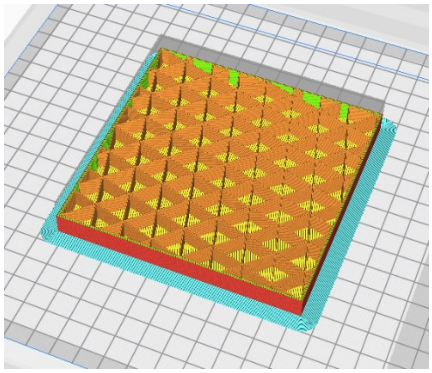
  You can simulate printing, observe what is going to be printed, see how it will be printed and **estimate the printing time**. This step is not mandatory but **highly recommended** as it will allow you to anticipate errors, save material and time.



- ***View type*** allows you to choose the type of view.

  ! *It is advisable to choose Layer view since this is what will allow you to see the structure and all the layers provided for printing.*





- ***Color scheme*** is the most useful tool to anticipate the construction of your room. This tool simulates the result of your printing by colouring the part according to several criteria.

You have the possibility to choose among the modes :

- ***Material color***, useful when printing in bi-material or bi-colour.

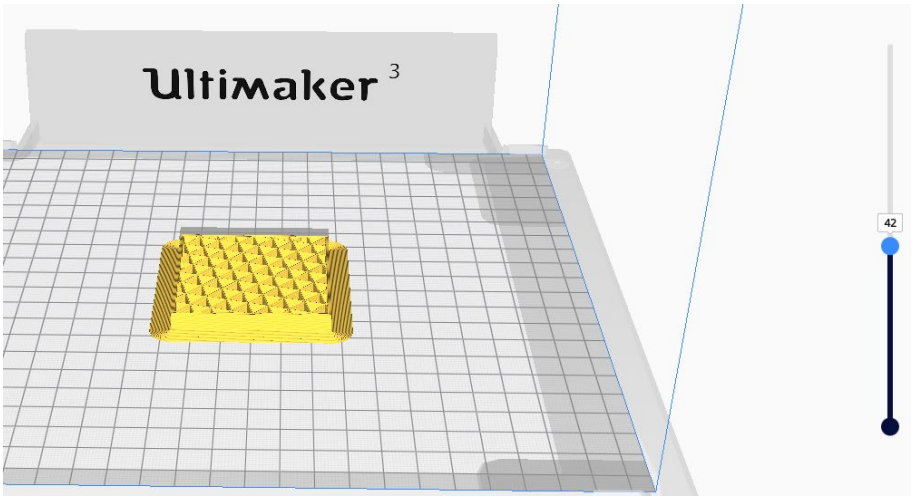
- ***Line type***, highly recommended for all prints because this mode shows the whole structure of the future piece.

. *Extrude 1* to show what will be made with material 1
. *Extrude 2* to show what will be made with material 2.
. *Travels* to see the path of the print head
. *Helpers* to see the brim, raft or supports
. *Shell* to see the outside walls of the room
. *Infill* to see the filling structure of the room
. *Top/Bottom* to see the bottom and roof of the room
. *Inner Wall* to see the inner walls of the room

- ***Speed***, useful to follow the evolution of the printing speed.

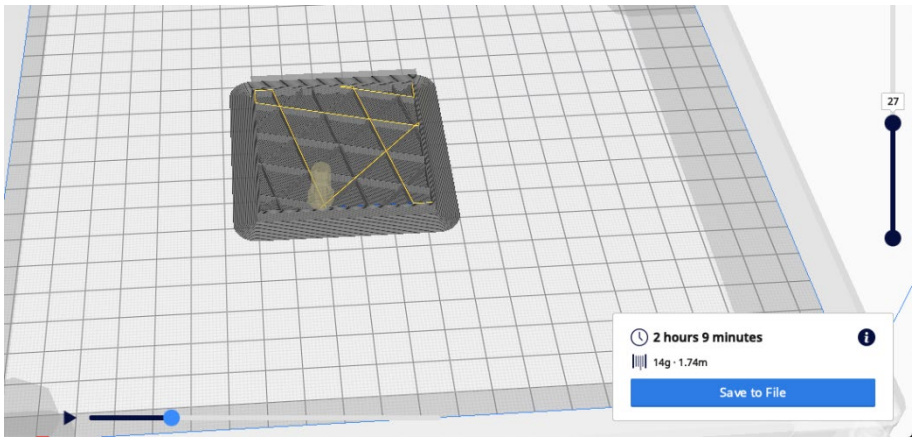
- ***Layer Thickness***, useful for checking layer thicknesses.

- ***The Layer Thickness Progress slider*** allows you to count the total number of layers that will make up the part and to see the state of the part at any stage of the printing process.

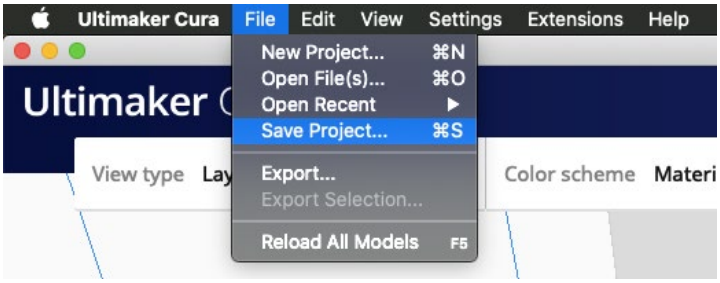


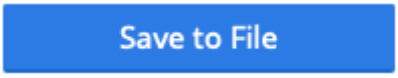
- ***The animation*** shows the movements planned by the machine for the production of each layer.

Choose at which stage of the printing process (layer number) you wish to see the path of the print head and then click Play.

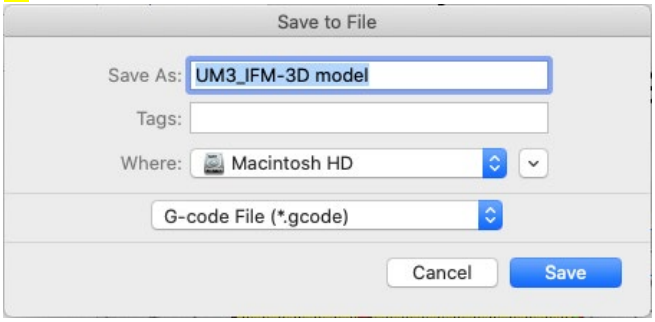


- ***!*** *You can also **save your project** if you want to finish the settings at another time or if you want to change settings without going through all the steps again.*



- **REGISTER THE G-CODE** using the button  on the **PREPARE or PREVIEW tabs**.

This step is essential ! Without the G-Code file, no printing is possible.
! *Check that the format is .gcode.*



3D printing

1. Start printing

- **SAVE** the print file **(.gcode)** on a USB key.

- **INSERT THE USB WRENCH** into the **UltiMaker 3** or **S5** (front side, next to the screen) that you have already prepared (materials and corresponding print cores).



- **SELECT THE PRINT ORDER** using the wheel button then click to access the list of files on the key. Select the file to be printed with the wheel-button.

  The printer heats up (nozzles and platen) and printing starts as soon as the components have reached the right temperature.

  ! Warning, touching the inside of the machine after printing has started is dangerous. You can burn yourself severely. !

1. During printing

Although the machine is self-contained and rather safe, it is mandatory to stay close to the machine during operation. When you use the machine, you are solely responsible for it.

Printing may go wrong for several reasons with different consequences:

- **Damage to the machine :** if another person touches the machine while it is in use, if an external element is present inside the machine enclosure, if the print head is out of adjustment and collides with the workpiece, etc., the machine may be damaged and will not be able to operate until it is repaired, penalizing the progress of each person's work.

- **Waste of material :** in the event that printing does not proceed as planned, it is possible that the material will continue to be extruded by the machine, sometimes for several hours. Extruded material is not reusable and must therefore be thrown away. If you monitor the printing process, you can stop the extrusion more quickly and save the material.

- **Wasting your time:** A printing failure can occur several hours after starting but also in the first few moments. The more closely you monitor the progress of your print run, the more you can detect the problem and restart a new print run, thus preserving your precious time.

2. After printing

- **REMOVE THE PART** from the tray after printing.

  Wait until the tray is automatically lowered to the bottom of the machine.

  ! *Caution, the glass plate may still be very hot. Gently tap the tray to control its temperature.*

  - **To remove the tray**, open the 2 small metal pins at the front of the tray and slide it out by pulling it towards you.

  ! *Never attempt to retrieve your print directly from the machine. This could damage the printer and its components.*





- Place the tray on a flat surface and use a metal spatula. Hold the tray with one hand and slide the spatula to the workpiece, trying to insert the tool under the first layer of the print.

- **If necessary, REMOVE THE SUPPORT** from your print.

  - **Construction material support** (PLA, ABS, CPE or PETG, Nylon, etc.) :
    Take a pair of cutting pliers and pinch the supporting parts of the print. Make small movements with the pliers until the media releases.
    ! Be careful not to damage your print.

  - **Soluble material** support (PVA) :

    Prints using PVA require post-processing. To do this, simply dissolve the PVA in water.

    - **Immersion of the print in water :**

      When placing a print with PVA in water, the PVA dissolves slowly.
      However, you can speed up this process in the following ways:

      - By using warmer water, the dissolving time is slightly shorter. When doing so, please take care not to use water with a temperature higher than 35 °C if your building material is PLA, in order to avoid any deformation of the PLA. Do not exceed 50°C for Nylon to avoid burning.

      - Use stirred/running water to considerably reduce the dissolving time. When water is stirred, PVA dissolves much faster, so it can be dissolved in less than three hours.

      - In addition, you make it easier to dissolve PVA by first placing the print in water for about 10 minutes and then removing most of the

substrate with tongs. Then put the print back into the water and the PVA residue will dissolve.

- **Rinsing with water :**

  Once the PVA substrates have completely dissolved, rinse the print with water to remove any excess PVA.

- **Drying of the print :**

  Allow the print to dry completely and if necessary apply further after-treatment to the building material.

- **Disposal of waste water :**

  As PVA is biodegradable, subsequent water removal is relatively easy. Once the water has been removed, run hot tap water for approximately 30 seconds to remove any excess PVA-saturated water in the pipe and to avoid long-term blockage.

! It is possible to use the water for several prints, but this may extend the dissolving time. The water will be saturated by the previously dissolved PVA. Therefore, for the fastest result, it is always recommended to use clean water.

- **UNLOAD THE FILAMENT(S) from the printhead from the printer menu.** Select the Material/PrintCore menu with the wheel knob. Click to enter the menu and choose ...

Cleaning and storage after printing

It is essential that everyone participates in the proper maintenance of the fablab and the machines to ensure quality and comfort for everyone. The machine and its work area must be cleaned and tidied up each time it is used.

1. Machine cleaning

- **REMOVE THE RESIDUES** of filaments inside the machine and dispose of them in the container provided.

- **If necessary, CLEAN THE GLASS TRAY** before putting it back in the machine.

  ! When the tray has been printed a lot and therefore several successive layers of glue, the adhesion of the pieces may be impaired.

2. Storage

- **STORE TOOLS** (pliers, squeegee, glue, etc.) in their respective places.

- *Be sure to **LEAVE THE WORKING AREA** as you found it when you arrived.*

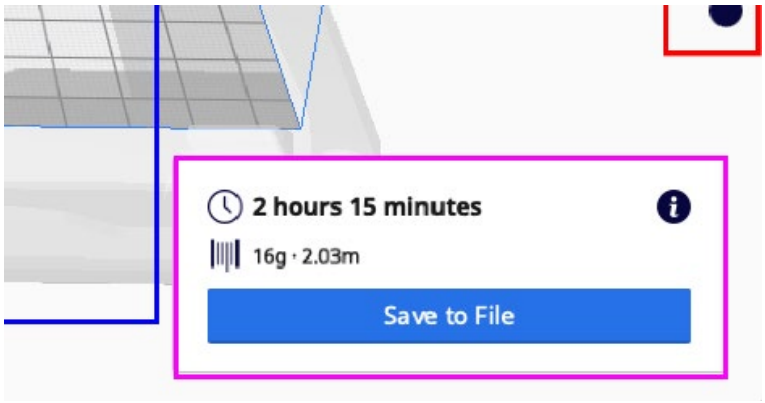
! Inform the fabmanageuse present in the fablab if the workspace was not in order, clean and tidy when you arrived at the machine.

ANNEXES

ANNEXE n°1

Check how much filament is left on your spool :

After slicing, you'll know exactly how many grams or meters of filament you'll need to print your object. Imagine you have a spool that you've already used for one or more previous prints and you're not sure if you'll have enough filament for this one.



In that case, check the following steps :





- Take the exact same spool as the one you're printing with.
- Ask a fabmanger to get you a scale and measure the weight **of the empty spool**.





- Measure the weight **of your spool with filament**.
- **Compare the weights** and subtract the weight of the empty spool from the weight of the spool containing the thread. This will give you the number of grams remaining on your spool.

YOU CAN ALSO REFER TO THIS TABLE FOR THE EMPTY SPOOLS ALREADY SCALED :

Brand	Bobin type	Weight
Ultimaker	750gr	230gr
Creality	Transparent 1kg	230gr
Polymaker	Cardboard 1kg	150gr
Winkle	Grey 1kg	222gr
Neofil3D	Black 750gr	250gr

- **THEN, BEFORE PRINTING, check that you have enough filament for your print :**

! DON'T FORGET TO ADD AT LEAST 50 CM MORE THAN NECESSARY, AS THE PRINTER WILL NOT USE THE LAST 50 CM OF YOUR SPOOL !

ANNEXE n°2

Rhino instructions for a good 3D print file :

- Once you are done with your 3d design, **make a copy!**

- In case you want to print one single piece, **you have to join all the single parts together**. In Rhino your design can have open and closed geometries, as it can be the result of surface or solid commands.

Remember:

➔ to **join open geometries** (surfaces and open polysurfaces) you will use the command **JOIN**

➔ to **join closed geometries** (closed polysurfaces and solids) you will use the command **BOOLEAN UNION**

- Be sure that **your 3d model doesn't have openings !**

➔ Use the command **VOLUME** to check it.

! If the result is a nurb, you can move to step 4 !

➔ Otherwise you can zoom and orbit around your 3d to check it.

➔ If you have flat openings, you can easily close them with the command **CAP**.

*! In the case the openings are not flat, you will probably use the commands **SWEEP1** or **SWEEP2** to create a proper surface that can close the opening (much better than PATCH!!).*

And then join all together (step 2.).

- Now **you have to create a polygon MESH** from your Rhino 3d model which is a **NURBS** geometry. To print we are obliged to convert the nurbs object into a mesh one.

➔ The command you're using is **FROM NURBS OBJECT** (first command under the Mesh menu). Move the slider to the right in order to have more polygons, a more detailed approximation.

If you want to play and explore how the mesh translation is done under the *detailed controls*, you'll find more info at this **link :**

https://docs.mcneel.com/rhino/5/help/en-us/popup_moreinformation/polygon_mesh_detailed_options.htm

- Now it's time to **analyze the mesh**. And to do it, it is better to switch to a **wireframe display.**

- Select the mesh just created and use the command **SHOW EDGES** to check that your model doesn't have naked edges or non-manifold edges.
  This is the **magenta test**. **Basically if you see magenta edges on your model, you have an issue.**

Remember:

➔ **naked edges are open edges**; it means that you have openings in your model (if you pass the volume test, the 3d model should be ok...but after the creation of a mesh naked edges could appear)

➔ non-manifold edges means that you have more than one face joined to a single edge

- **If you don't see magenta in your viewport, your 3d model is valid and printable**.

- You now need to use the command **EXPORT SELECTED** and save with the proper file format which is **STL**.

! If you have naked edges or non-manifold edges you have to go back to step 3, or to step 1. Sometimes it is better to start again if you see that it takes too long to fix the geometries. Sometimes you can easily fix issues. It depends on your 3d model, on the kinds of errors.... !

Remember:

1 object = 1 STL file. **Do not save multiple objects in a single STL** because if one object contains an error, you won't be able to print the other objects of the STL. **You can prepare a plate with multiple STL files in the 3D printer slicer softwares**.

➔ If you can't use Rhino or your model is from an online 3D models bank but you need to repare it, **you can also use MeshMixer and the repare option**.