3D printer

Creality K2 Plus



institut
FRANÇAIS
de la
MODE

Table of Contents

FabLab safety rules and good practices 3

Safety in 3D printing 3

Description of the 3D printer 5

1. Printer diagram 5
2. Features 5

Preparation of the 3D model 6

Parameterization of 3D printing or "slicing". 7

1. Presentation and environment of CrealityPrint 7
2. Using CrealityPrint 8

Setting up the printer 23

1. Turn the machine on 23
2. Loading of materials 23

Printing on the machine 26

1. Start printing 26
2. During printing 28
3. After printing 28

Cleaning and storage after printing 31

1. Machine cleaning 31
2. Storage 31
3. Pay your filament consumption 31

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.**

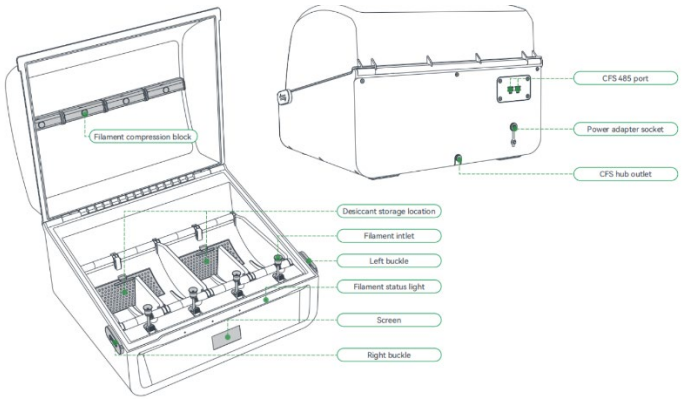
The 3D printer includes belts to move the print head and platen. The force used to move its components can injure the user.rice in case of misuse.

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

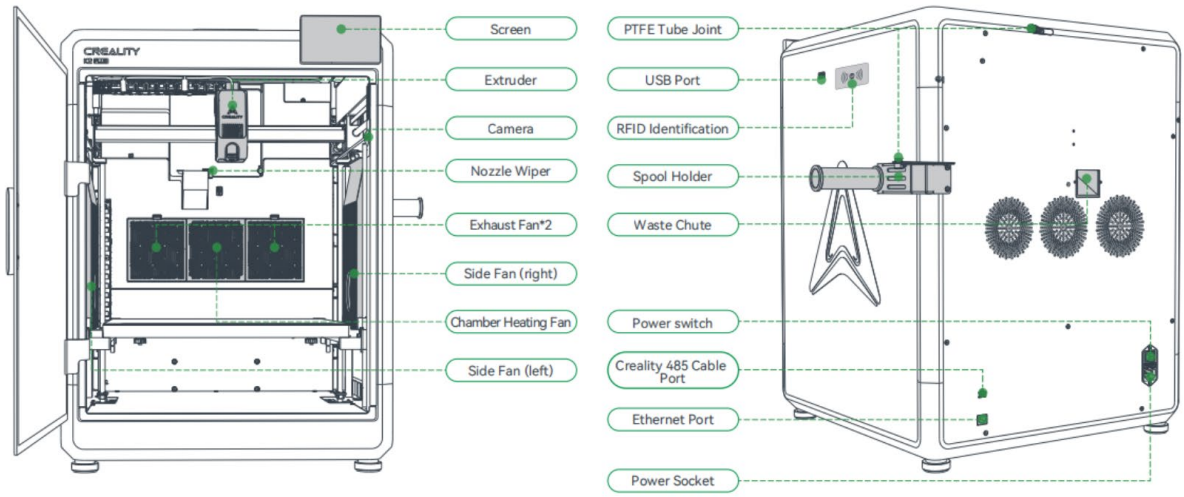
Description of the 3D printer

1. Creality K2 Plus diagram

CFS
(filament system) >



Printer



2. Features

	Parameters
Model	K2 Plus
Print size	350*350*350mm
Pinter size	495*515*640mm
Single set net weight	35kg
Supported filaments	PLA/ABS/PETG/PA-CF/PLA-CF/PET/ASA/PPA-CF
Extruder type	Proximal dual gear extruder
Printing speed	≤600mm/s
Acceleration	≤30000mm/s²
Nozzle diameter	0.4mm
Nozzle temperature	≤350℃
Hotbed temperature	≤120℃
Chamber temperature	≤60℃

Preparation of the 3D model

To obtain a 3D model, you should use a **3D modelling software** of your choice (it could be Rhinoceros, 3DSMax, Fusion 360, Blender, etc.). The software does not matter, however you have to **export the model as an STL file**.

! Always save the original model in the software editable format. You can't modify an STL file.

To avoid any problem, make sure:

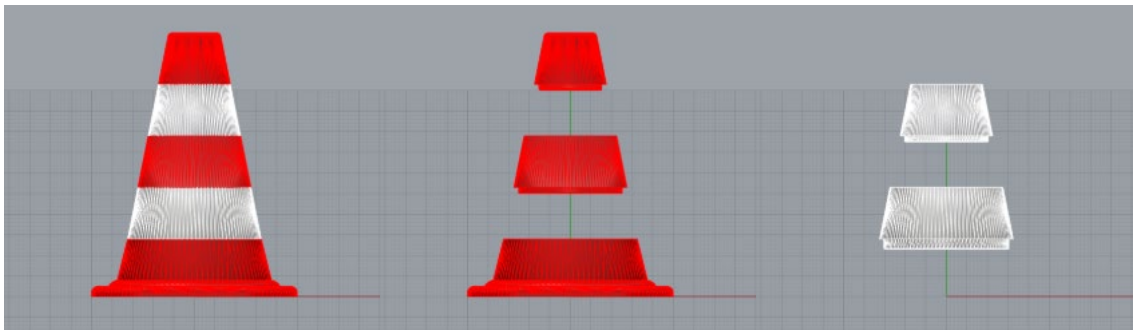
- Your model is completely closed (no hole, no naked edge)
- Your model as a minimum thickness (it can't be just a surface, like a sheet of paper)

You can download the free software MeshMixer to analyze and repair your file.

If you want to print a multi-color object, you should:

- Create one original model cut into different pieces that you save in the editable format
- Export one STL file for each color (never change the original position of each piece in the software)

Example:



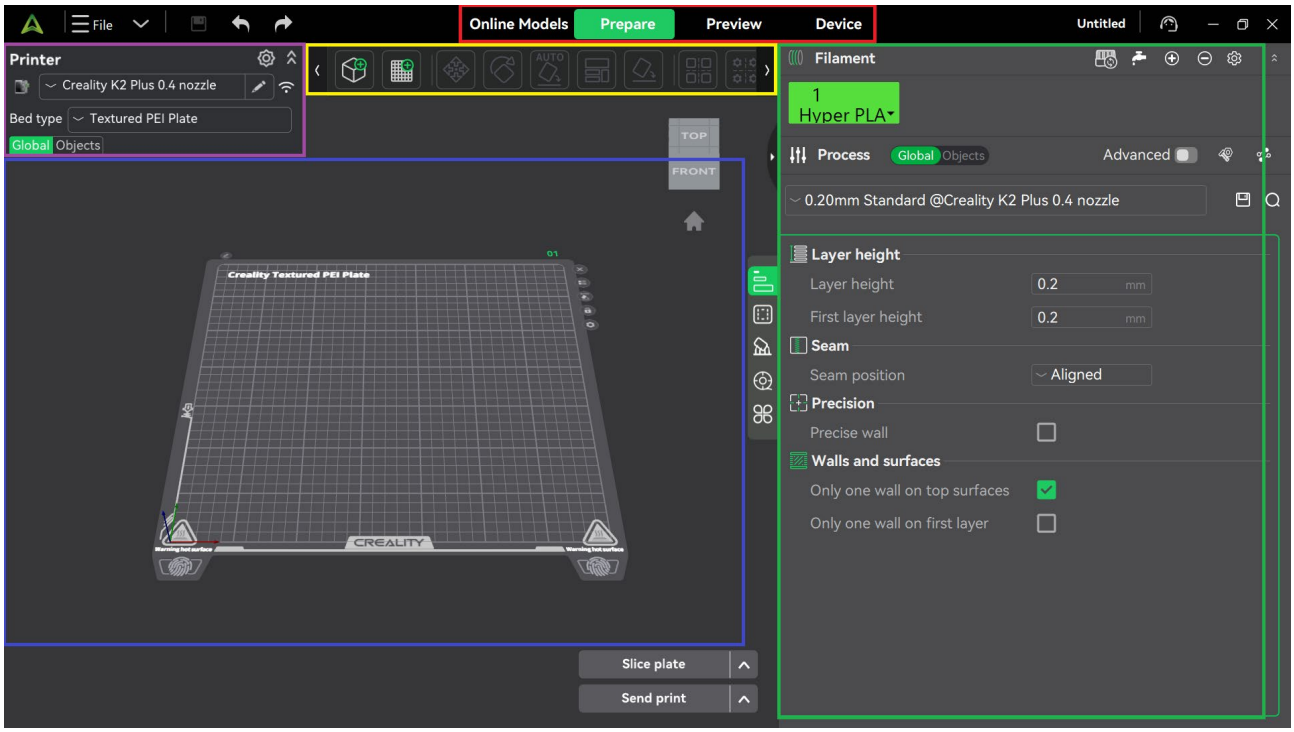
Parameterization of 3D printing or "slicing".

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 CrealityPrint

CrealityPrint is a free software you can download on your computer. To work with the K2 Plus machines, you should use CrealityPrint 5.0 or later version.



Menu / access to different tabs

Machine configuration

Visualization area

Tools section

Parameters section

2. Using Creality Print

PREPARE tab

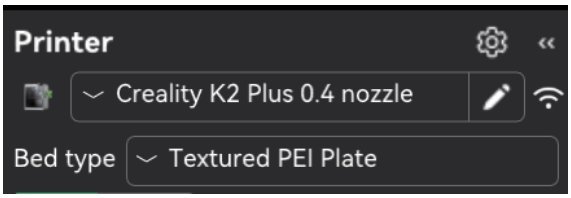


This tab gives access to all the fields that will allow the user to prepare a printout. From the choice of the 3D model to the precise parameterisation of the print, including the types of materials, it is essential to go through this step to make a successful print.

A. SET THE MACHINE CONFIGURATION

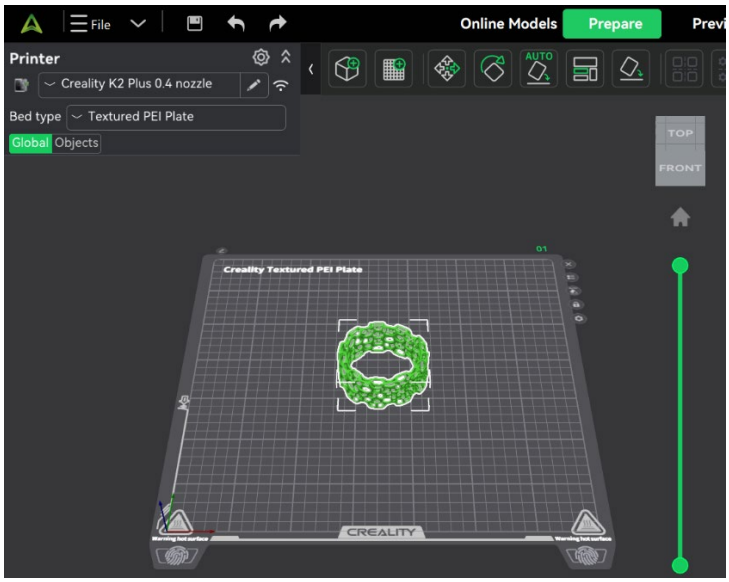
- **Select the right printer model** and **bed type** by choosing in the printer menu:

You should have >>>



B. SET YOUR OBJECT

- In the tool section, **click on**  **to upload your .STL file**
  You can also drag and drop it on the bed in the visualization area.

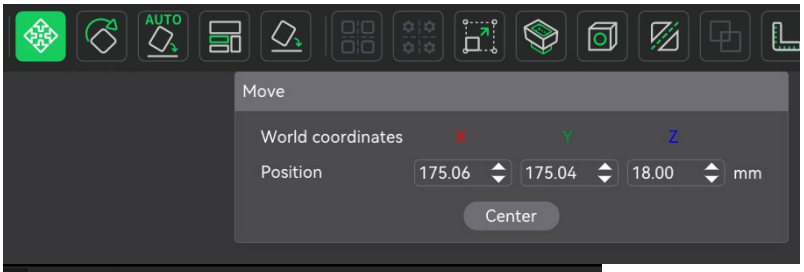


! For a bi-material or bi-colour print, you must have 2 .STL files: one for each material or colour. Import the 2 files directly by dragging them into the Creality Print window.

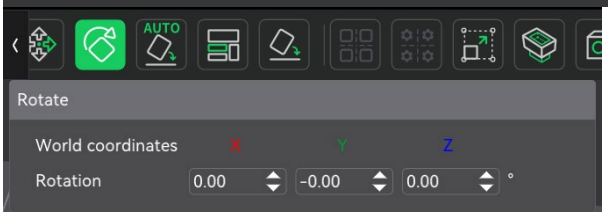
- In the top menu, **you can select different tools** to adjust your object to be printed:



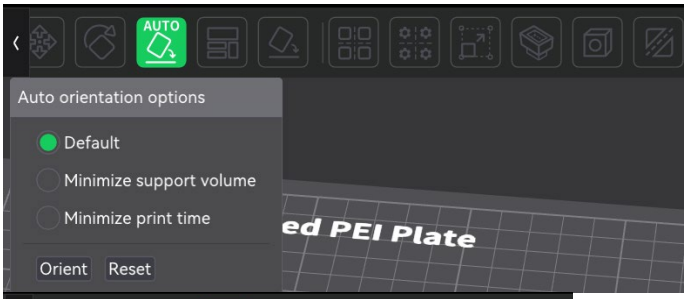
Tools overview:



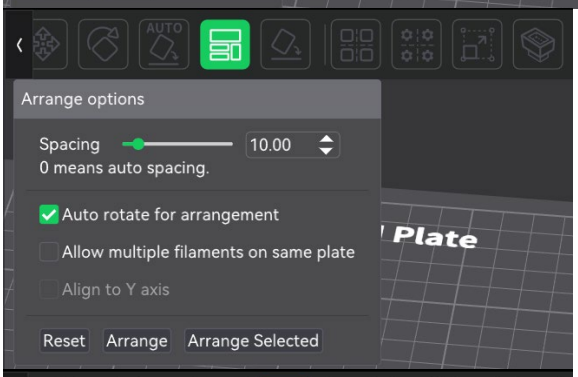
Move is to choose the placement of your object onto the bed



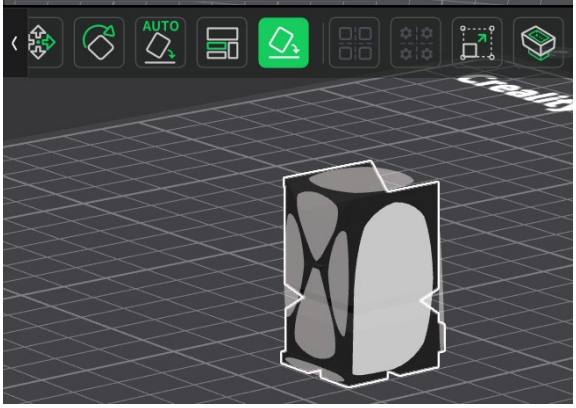
Rotate is to set the orientation of your object, related to its shapes



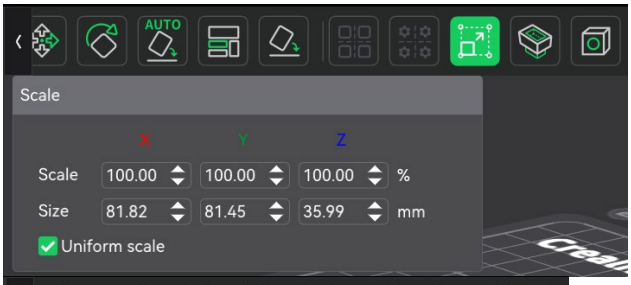
Auto permits you to choose an automatic orientation to reduce support volume or printing time



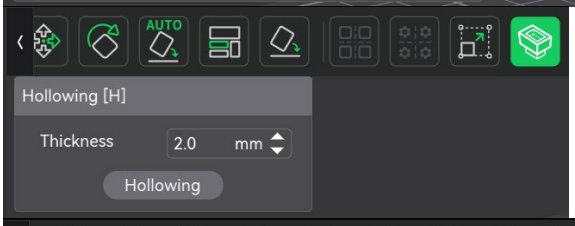
Arrange all objects is usefull when you print multiple objects on the same plate



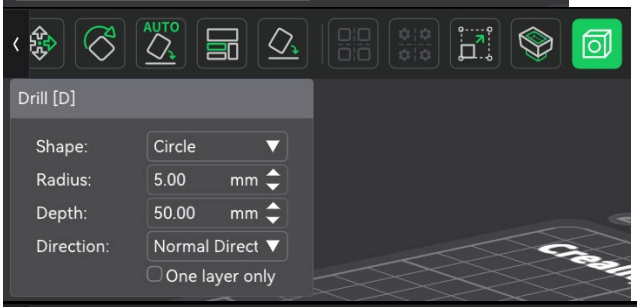
Lay on face permits you to select the face you want the object lay on



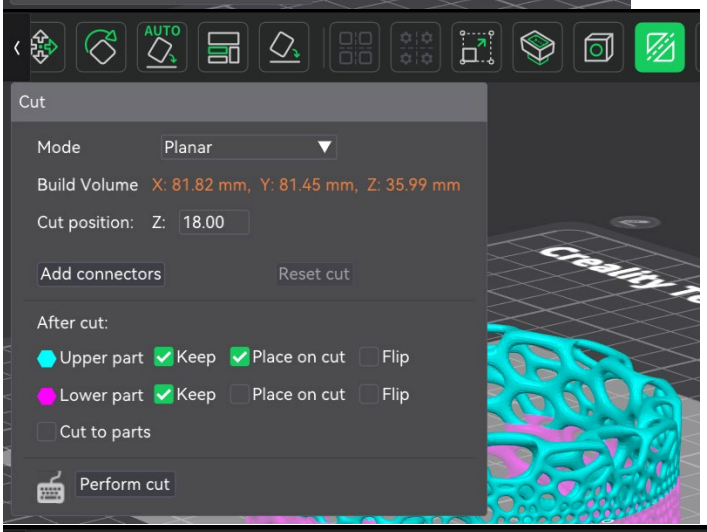
Scale is used to change the size of your model



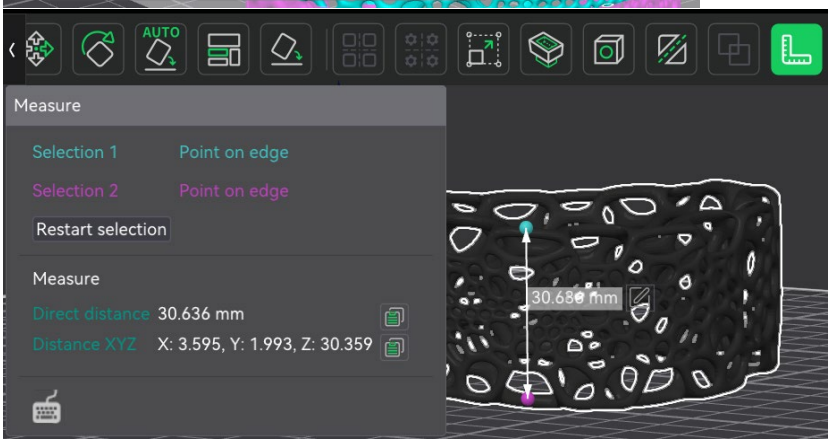
Hollowing is usefull is you want your object to be a shell



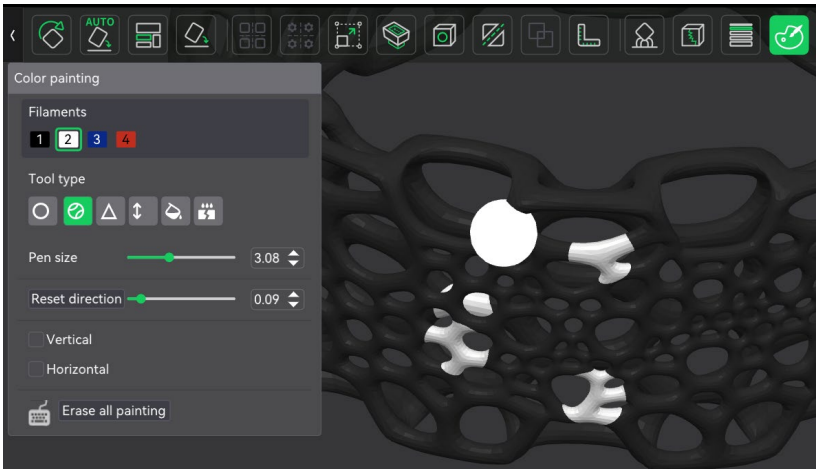
Drill is used to create a perforation in your model



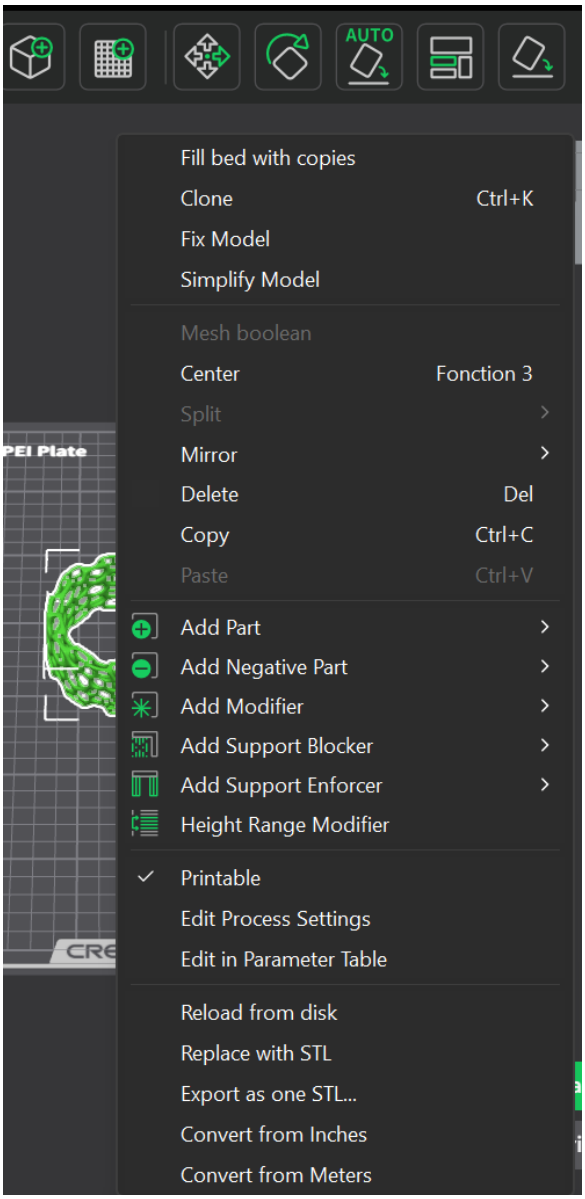
Cut permits you to cut your object



Mesure is used to know the distance between two positions



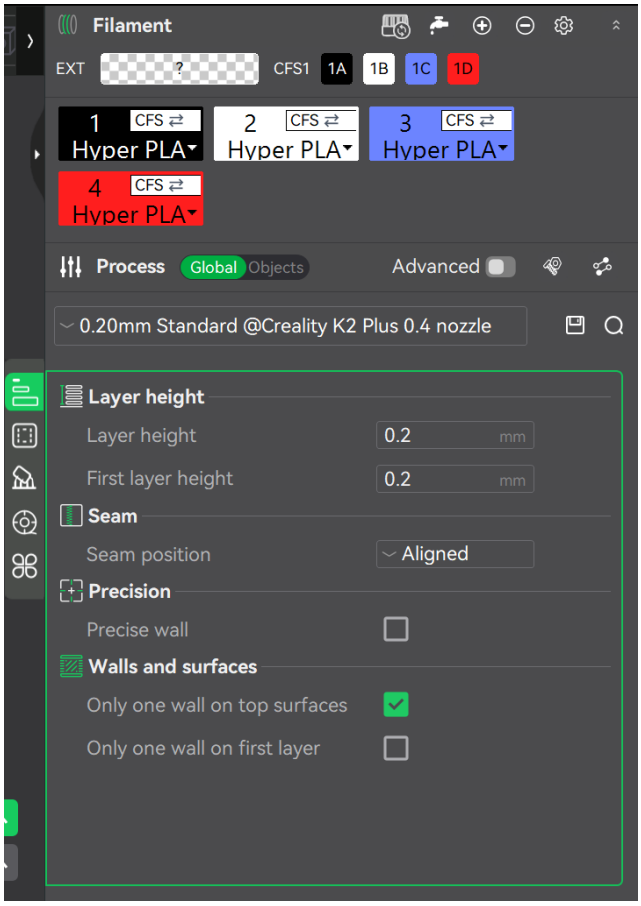
Color painting is made to let you draw on the model with another filament color



By **mouse right-click**, you can access other functions.

C. SET THE PARAMETERS

You have to set theses tabs to obtain the printout you expect:



Filament to determine the filament(s) you want to print with, related to what filament is installed in the CFS

Process to decide if you want to prepare settings for the global print or for each object (when multiple object on same bed)

Settings to prepare the exact parameter you want for your printout.

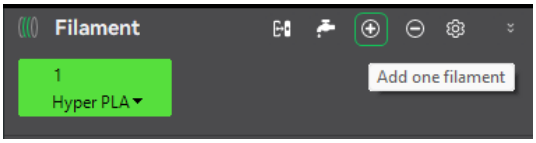
- **Set Creality Print software with the same materials set in the CFS**

  ! You're not allowed to connect your computer to the machines. Please, use the following instructions!

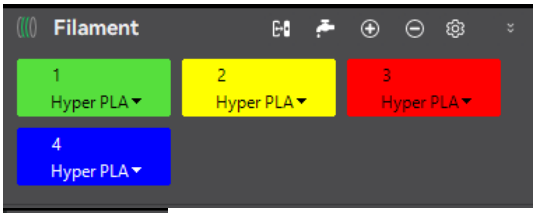
  - **Check the CFS and the order of materials**



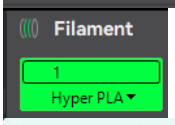
- In the filament section on Creality Print, **click on the button *Add one filament***
  ! Make sure you add Hyper PLA!



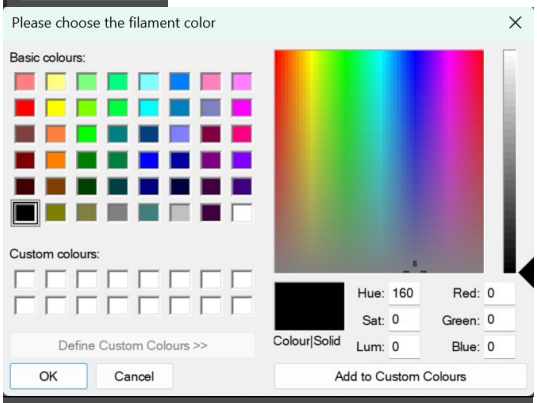
- **Add new ones till you have 4** materials in the window like >>>



- **Click on the number**

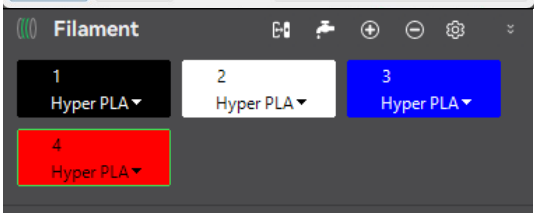


- And **select the right color** related to the filament number in the software and in the CFS

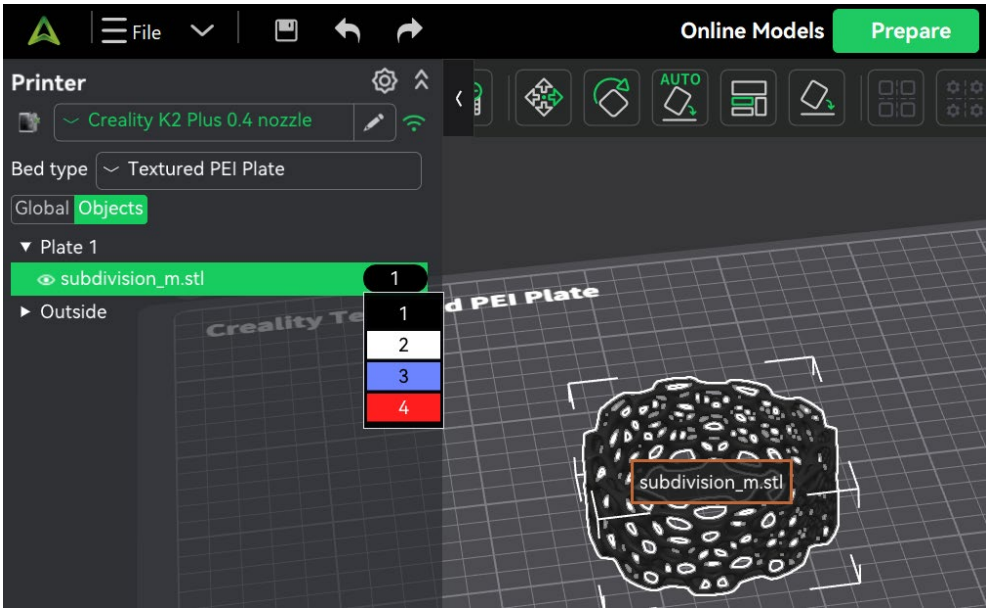


- **Do it for the 4 filaments** to obtain:

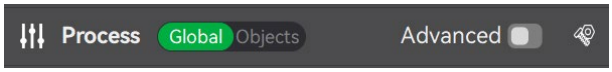
  ! If you need another material configuration, ask the fabmanager!



- You can now **apply the material to your object(s)** by selecting it/them **and click on the material number** you want in the configuration menu:



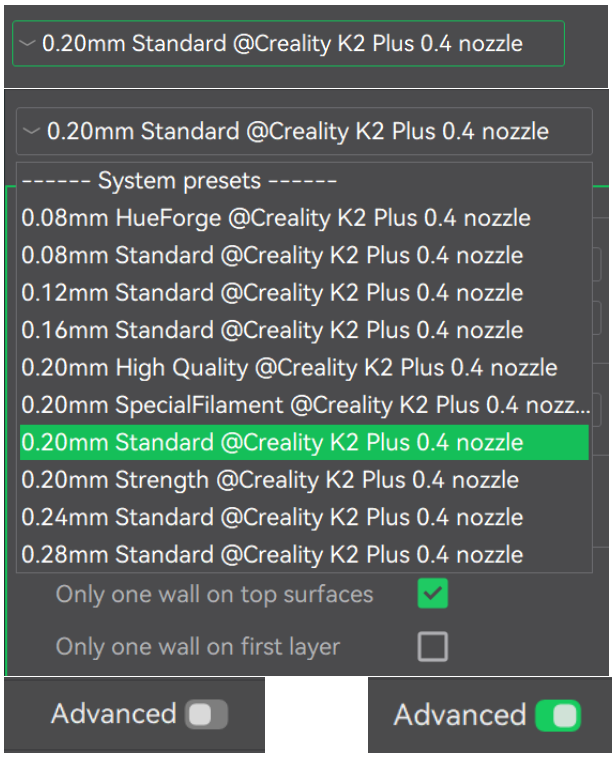
- In the ***Process tab***, you can choose different options:



With ***Global*** you will choose the parameters for all the objects deposed on the bed.



With ***Objects*** you can select each object on the bed and apply different settings to them.



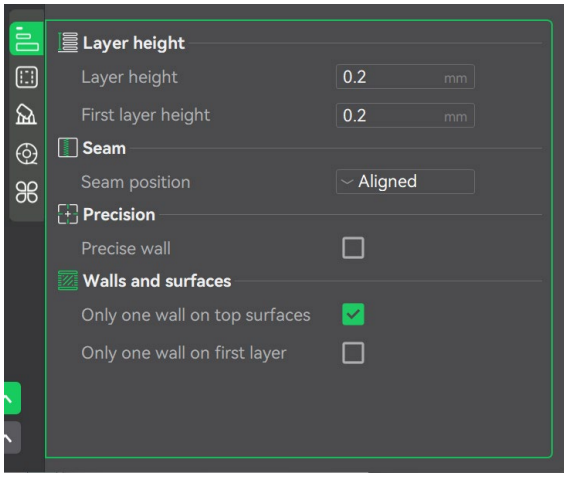
With this menu, you can select different **presets**.
Click on it to see the ones you could choose from.

! 0.4mm nozzles are set on the machines.

*! For a first test print, we recommend **0.2mm Standard @Creality K2 Plus 0.4 nozzle***

The ***Advanced*** button reveals some hidden settings.

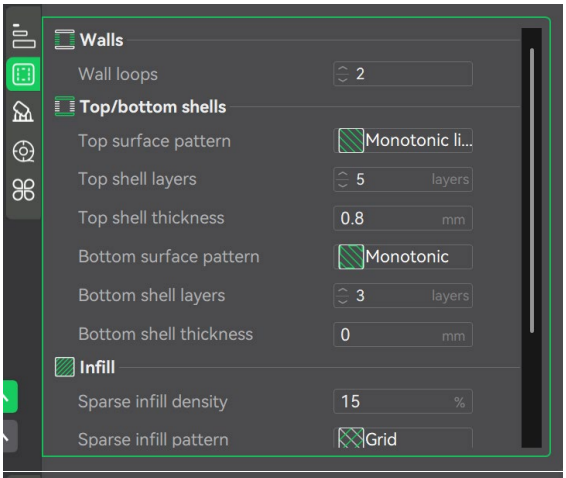
- **Adjust the settings** if needed:
  ! The following image captures contain basic settings, not necessarily what you should set for your specific printouts!



Layer tab is about the surface aspect of your printout.

Set ***Layer height*** / ***Seam*** / ***Precision*** / ***Walls and surfaces*** related to your needs.

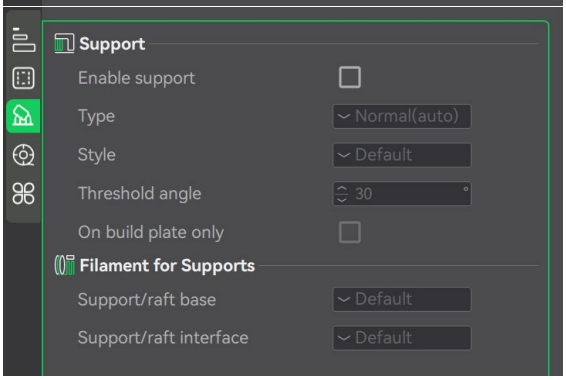
! Choose 0.2 mm layer height for first test print and try thiner if needed.



Strength tab permits you to define the structure of your object.

Set ***Walls*** / ***Top/bottom shells*** / ***Infill*** related to your needs.

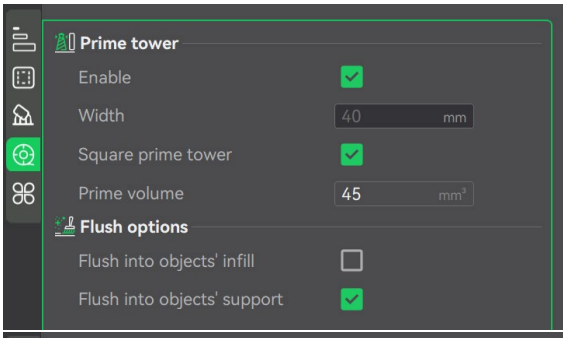
! Do not set less than 1 wall loop and 10% of infill except for experimental printout!



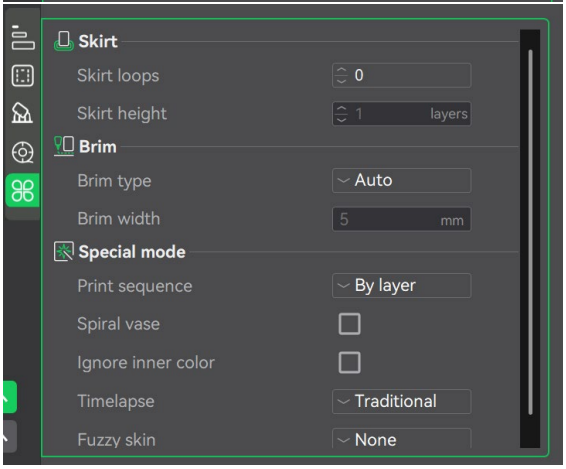
Support tab is used when the support option is needed for the print.
Tick ***Enable support*** to access the settings.

*! You should use the **Support** option for thin or angled and horizontal shapes.*

Visit the link below for more informations:
https://wiki.creality.com/en/software/update-released/Support/support-settings



Multifilament tab is used when you print with minimum 2 filaments (to clean the nozzle during each layer when filament color changes).

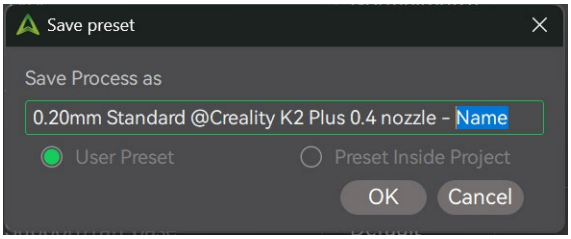


Others tab gives access to other settings you may need.

Set ***Skirt*** / ***Brim*** / ***Special mode*** related to your needs.

*! If tiny surface of your object in contact with the bed, apply the **Brim** option.*

- If you want, **you can save your own preset** by clicking on 



Choose a name and click **OK** to confirm the saving.

D. Create the G-code

You need the G-code file to print an object on the machine.

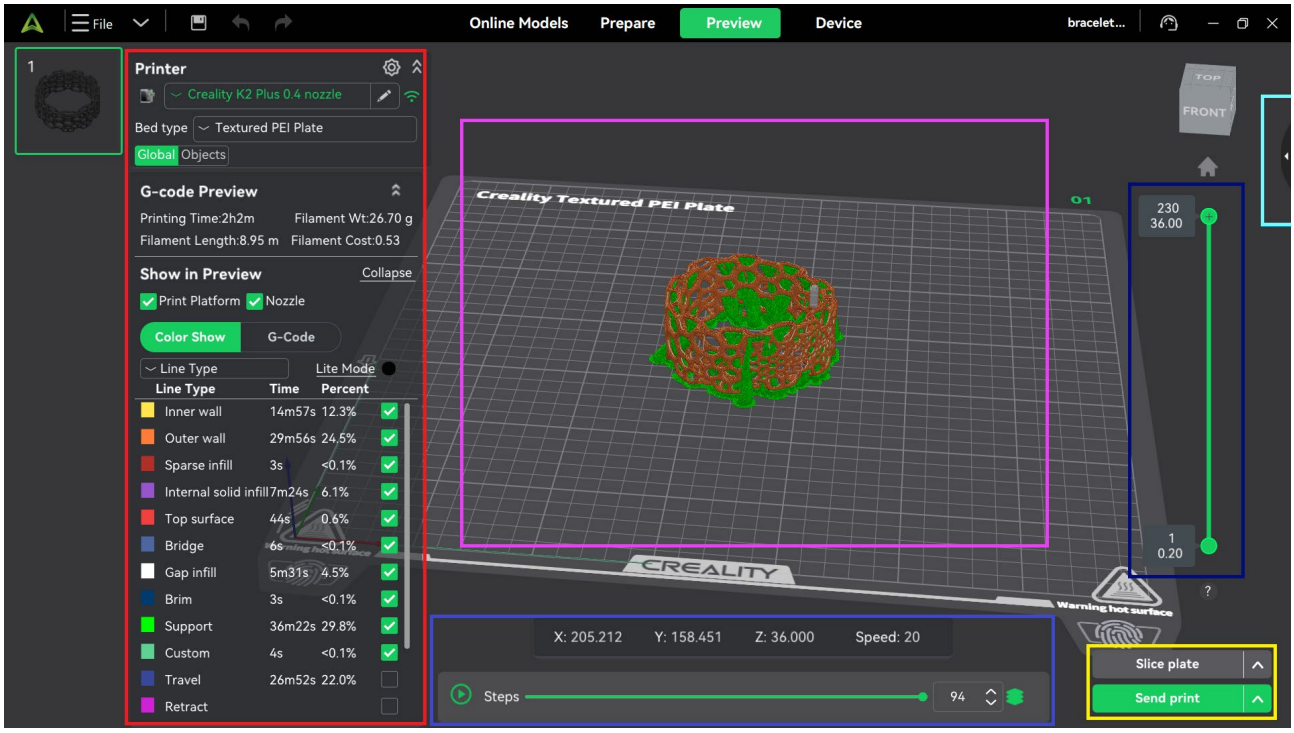
- **Click on** 

to create the printing file and have a look at the preview of the printout.
! *As soon as you change the configuration or print settings, you must click on* ***Slice plate*** *to update the G-code.*

PREVIEW tab



In this tab, the user can simulate their 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 testing phase.



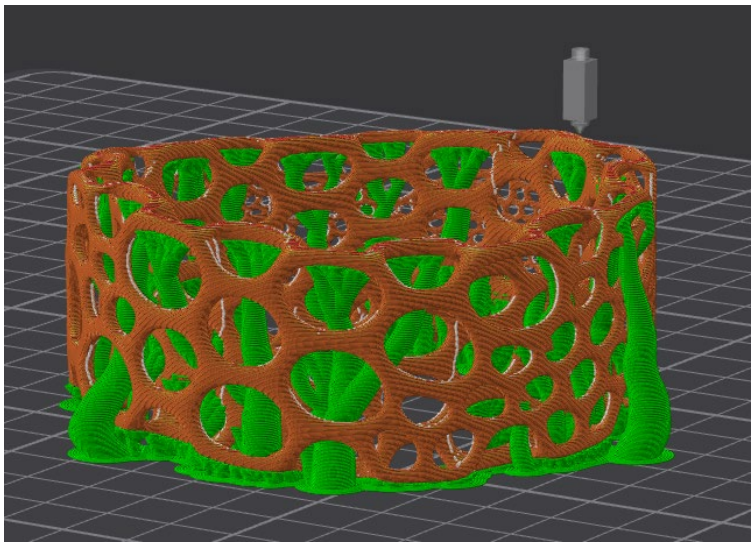
Preview of the object
Preview of details
Step reader

Layer-by-layer cursor
Setting tab access button
Slice and send print buttons

A. Analyse the preview

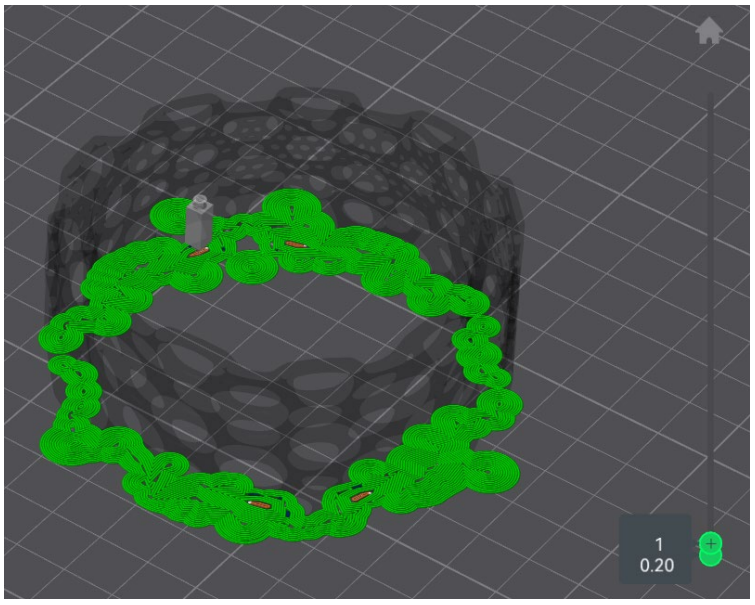
The preview of your printout can be decisive for a great result!

- **Check the general aspect** of the printout:



- **Brim**
- **Support**
- **Object**

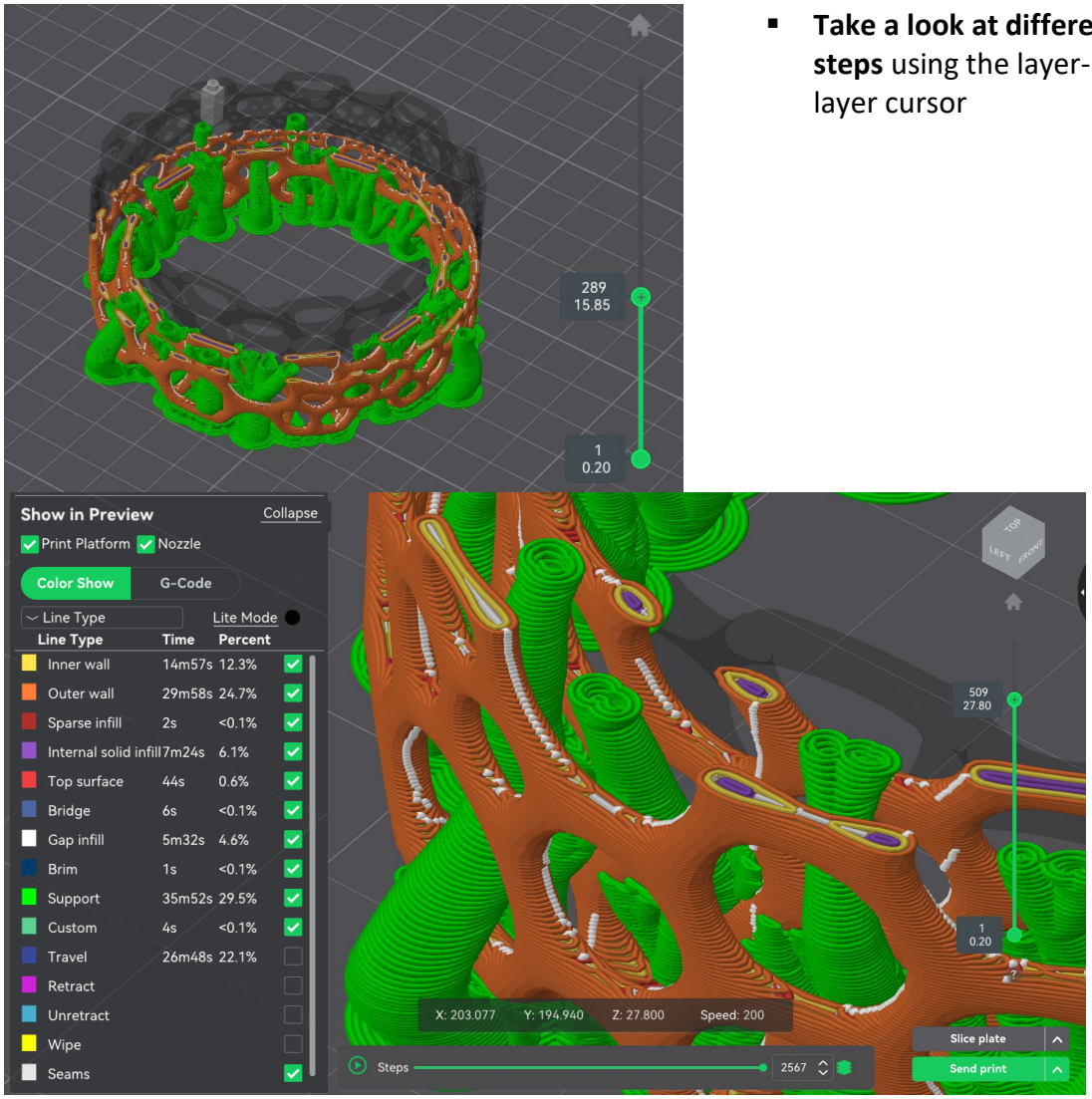
- **Check the aspect of the first layer using the layer-by-layer cursor**



- **Go to the first layer and check the volume of the first layer in contact with the bed.**

! Surface in contact with the bed is decisive for a great result. If no surface or tiny surface of the object touching the bed, any other layer will fail!

- **Check out some other layers**



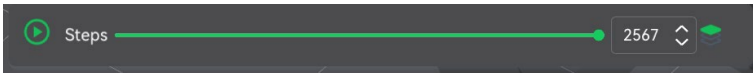
- **Take a look at different steps** using the layer-by-layer cursor

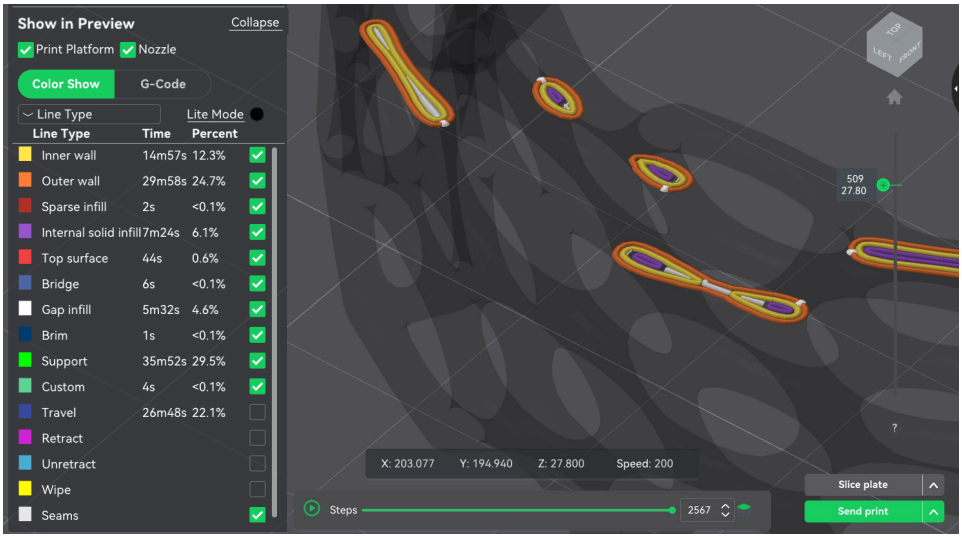
- **Analyze how the layers are made** thanks to the ***Show in preview*** **caption** (walls, infill, seams, top surface, etc.).

- **Controle** the thickness of the shell, the quantity of material inside the object, the shapes of the support, the density of the support.

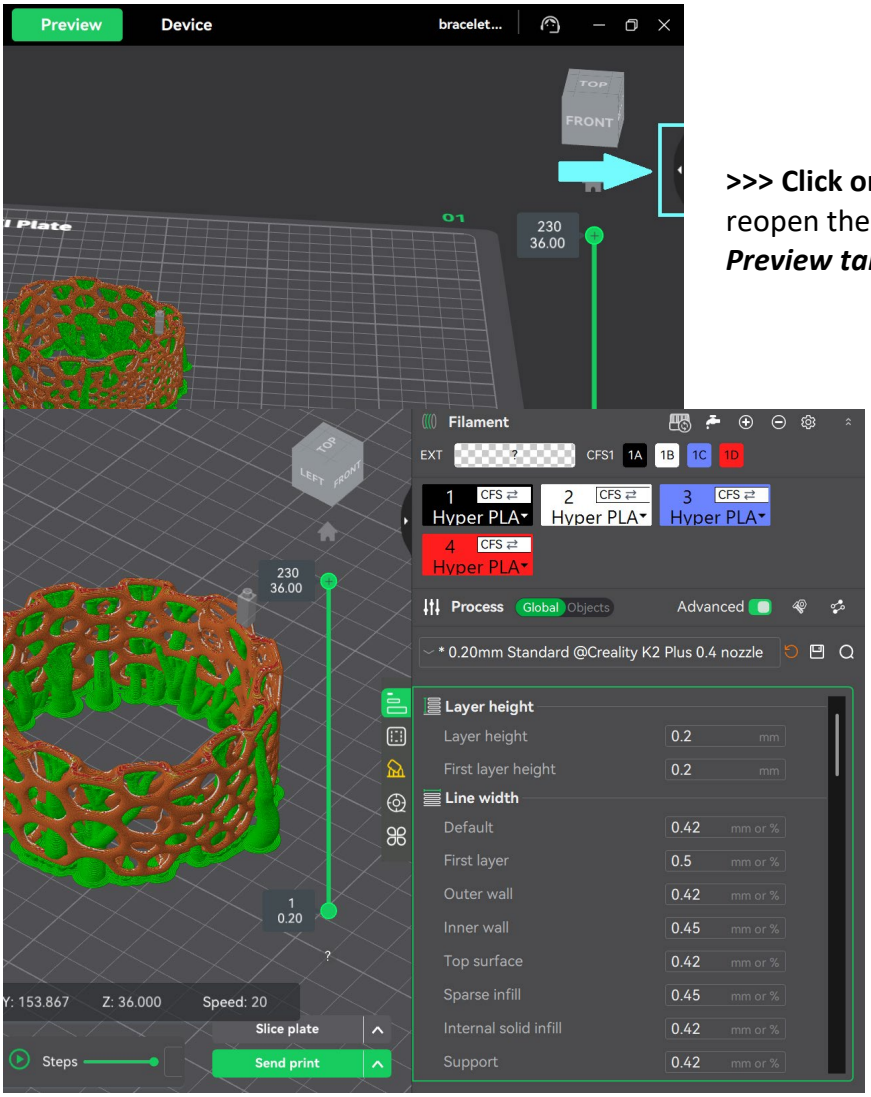
  ! There is no main rule we can tell. What would work depends on the shape of your object, thickness and volume. Do not hesitate to ask the fabmanager!

- **Click on**  **in the *Step player*** to have a different view of each layer:





- **Adjust the settings** if needed

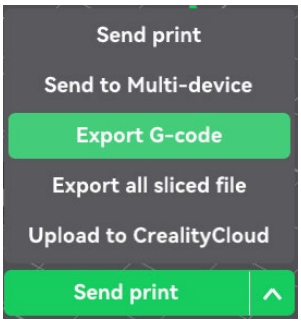


>>> Click on this button to reopen the ***Setting tab*** from the ***Preview tab***

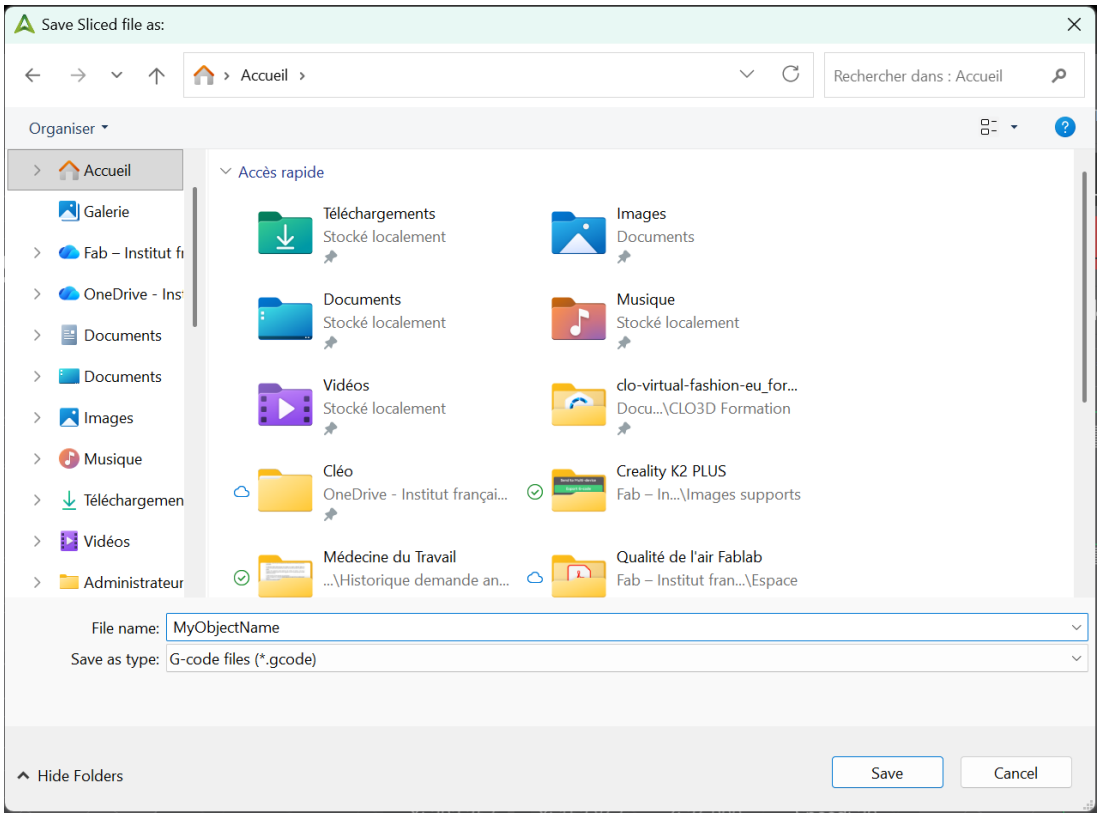
B. Save the Creality Print files

- **Click on** 

to open the menu:

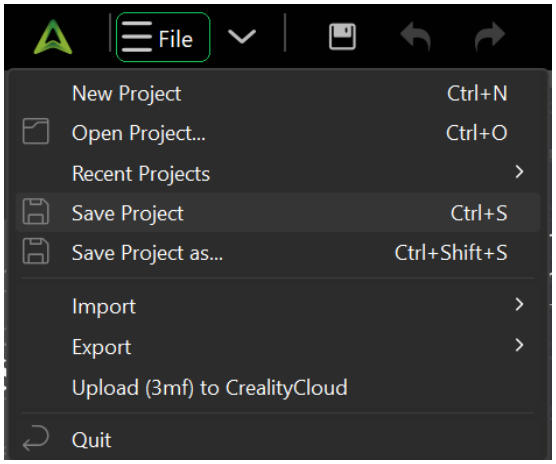


>>> **Select *Export G-code***

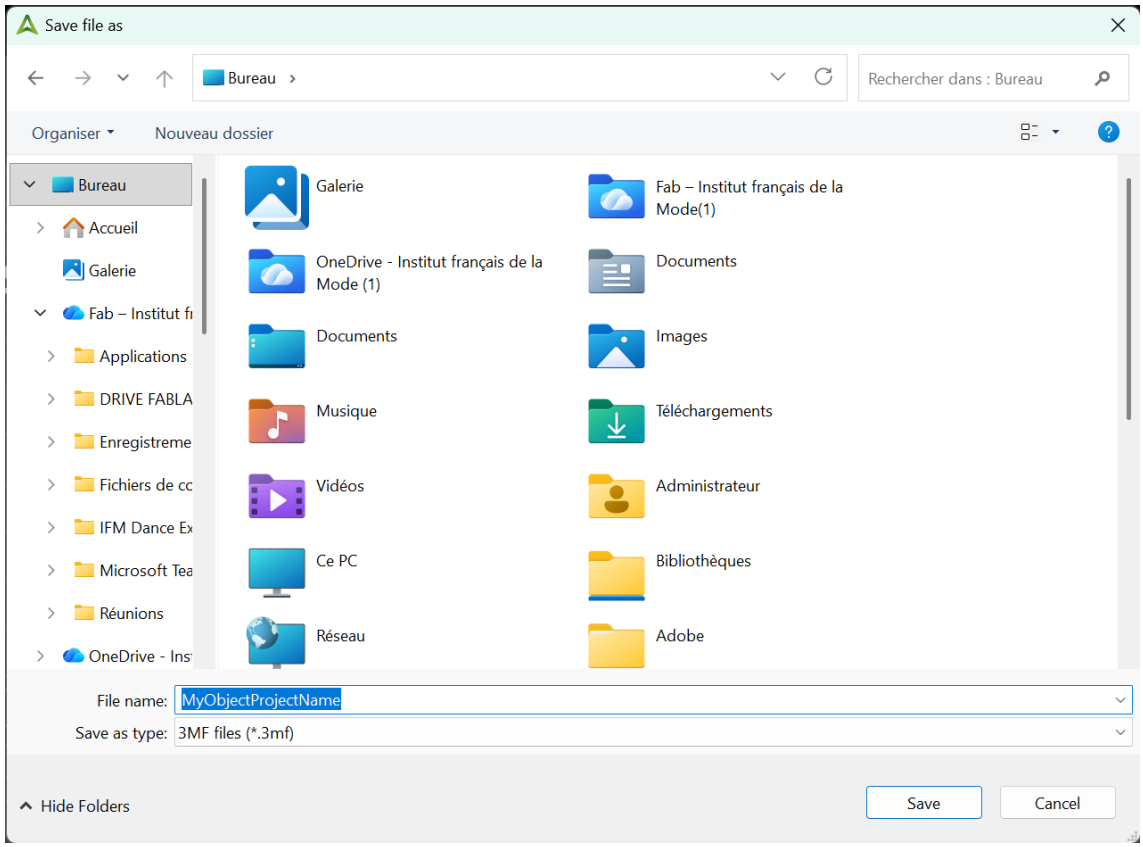


! The G-code can't be modified. ! You also have to save the project file to be able to change settings if any problem!

- **Save the project by clicking on** 



>>> Select *Save Project* or *Save Project as...*



- **Copy your G-code file on a USB drive**

Setting up the printer

1. Turn the machine on



From the left side, reach around to the back of the machine to locate the ON/OFF button (above the power cable).

The machine and the CFS will turn on at the same time.

2. Loading of materials...

CAUTION :

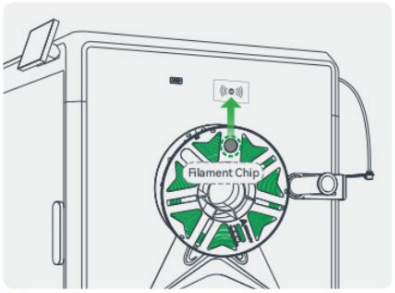
The machine can run the following materials:

PLA / **PETG** / **PA-CF** (Carbon Fiber nylon) / **PLA-CF** (Carbon Fiber PLA) / **PET** / **ASA** (UV resistant) / **PPA-CF** (high strengh Carbon Fiber nylon) / **TPU** (flexible).

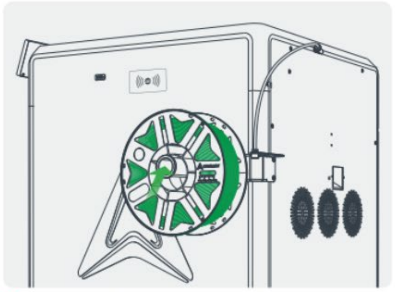
BUT :

- **TPU can only be set on the spool holder (NEVER in the CFS)**
- **Ask the fabmanager if you need to use anything but PLA in the CFS**

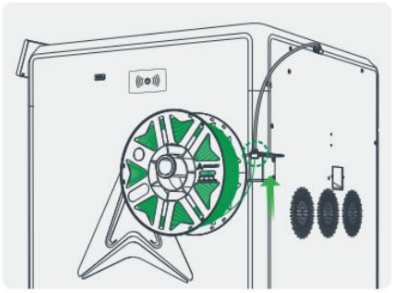
A. ...From Spool Holder



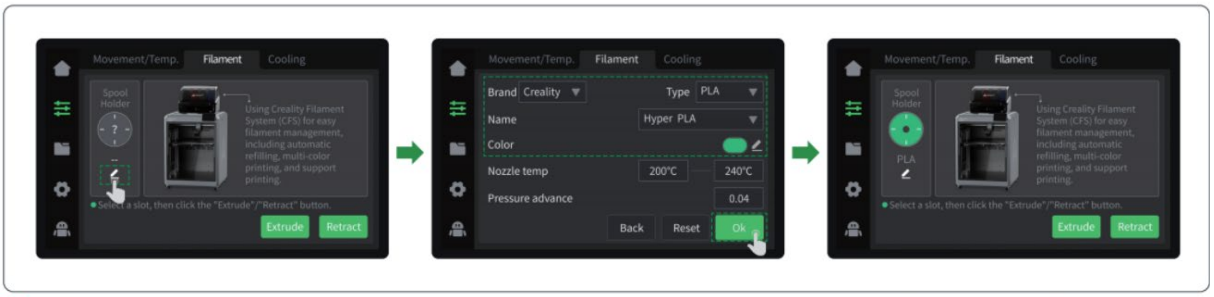
1 If the filaments support RFID recognition, align the chip on the filaments with the RFID recognition position of the machine body to scan the filaments, and the filament information can be automatically set;



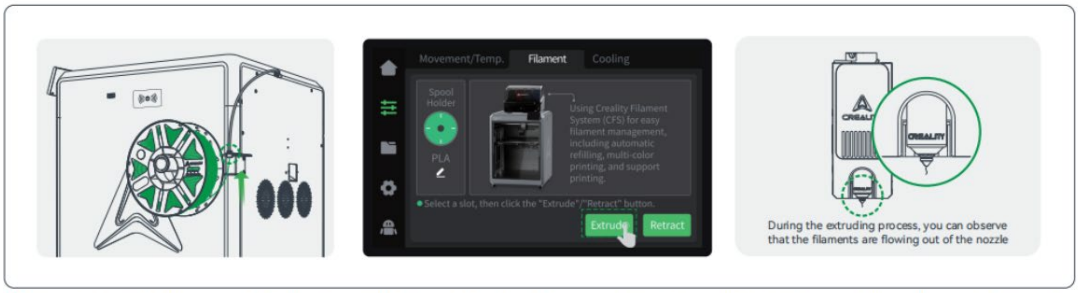
2 Hang the filament on the spool holder;



3 Thread the filaments into the Teflon tube and push gently until they cannot be pushed;



4 If the filaments do not support RFID recognition, you need to manually click on the screen to set the filament information: Adjustment page → Filaments → Edit (as shown below the material rack), set the filament brand-type-name-color respectively, and finally click OK to save the settings;



5 Extrude: Push the filament gently by hand, and click "Extrude" on the filament interface. The machine will automatically set the current filament temperature, and will automatically extrude the filament after the heating is completed;

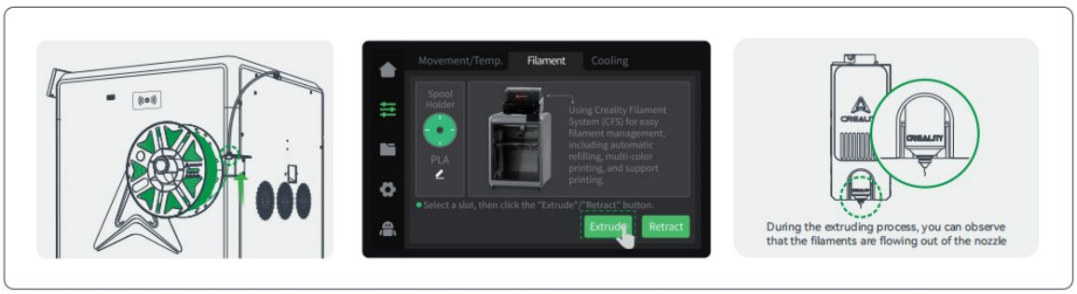


6 Retract: A. Click Retract on the filament management page, and the extruder will automatically move to the left front to cut off the filament for retraction;
B. Wait for the retraction to end, and extract the filament from the Teflon tube behind the machine.

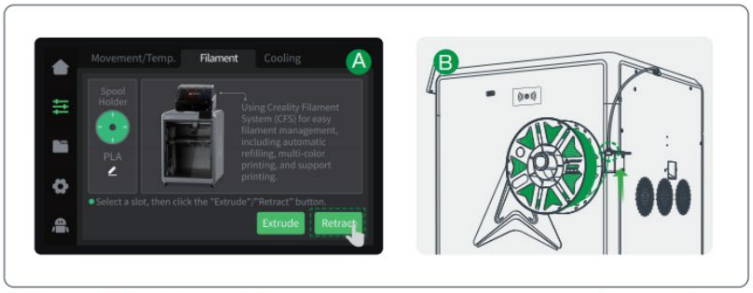
During extruding, you can observe whether there is filament flowing out of the nozzle. If no flow-out is observed, you can gently push the filament toward the extruder at the Teflon tube behind the machine, and then click "Extrude" again.

The current interface is for reference only. Please refer to the latest software/firmware UI on the official website for updates.

B. ...From CFS (if not already set)



5 Extrude: Push the filament gently by hand, and click "Extrude" on the filament interface. The machine will automatically set the current filament temperature, and will automatically extrude the filament after the heating is completed;



During extruding, you can observe whether there is filament flowing out of the nozzle. If no flow-out is observed, you can gently push the filament toward the extruder at the Teflon tube behind the machine, and then click "Extrude" again.

The current interface is for reference only. Please refer to the latest software/firmware UI on the official website for updates.

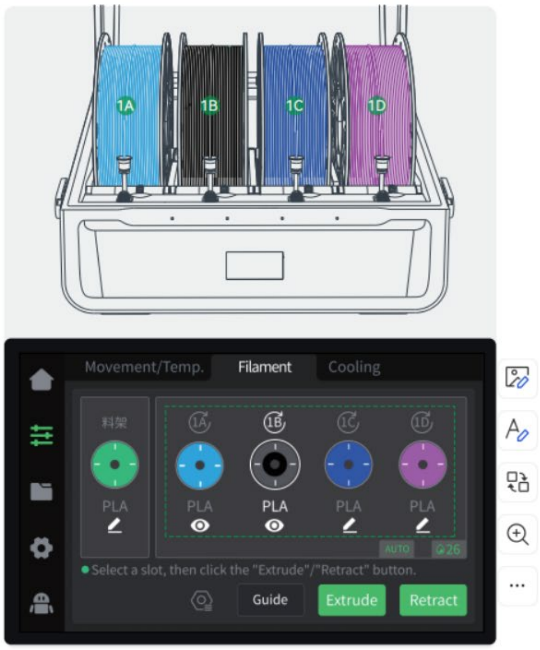
6 Retract: A. Click Retract on the filament management page, and the extruder will automatically move to the left front to cut off the filament for retraction;
B. Wait for the retraction to end, and extract the filament from the Teflon tube behind the machine.

C. Check the corresponding filaments on the screen of the machine



1 Put in filament and wait for tightening (RFID filament does not need to be edited, in case of non-RFID filament, "?" will be displayed after reading, and filament needs to be edited manually);

The current interface is for reference only. Please refer to the latest software/firmware UI on the official website for updates.



2 Check whether the filament information displayed on the screen corresponds to the filament in CFS.

Printing on the machine

1. Start printing

- **Turn the machine on!**



You will find the **turn ON/OFF button** at the **lower left corner** of the machine.

- **Insert the usb drive containing your file** into the Creality K2 Plus (top right side of the machine)



- **Follow the process to start the print:**



- This is the **home menu**: **check that** you see the **right color materials** in the **CFS**

- **Click on the folder pictogram** and **select the USB drive tab**

- **Check the print time** and **quantity of material.**

  ! You have to pay for your consumption of material !

- If needed, you can still change the color of material (only if it is the same product reference)

- **Click on *Print***

- The machine will start to print.

  ! It is important to stay in front of the machine to watch the first layer to be printed!

2. During printing

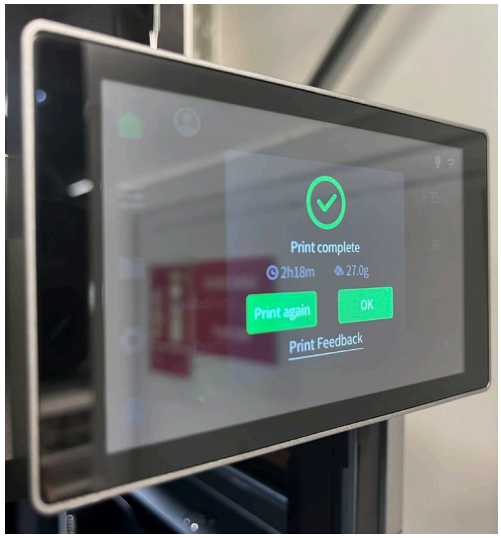
Although the machine is self-contained and rather safe, it is mandatory to stay close to the machine during operation. When you use the machine, you are solely responsible for it.
Printing may go wrong for several reasons with different consequences:

- 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 and paid. 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.

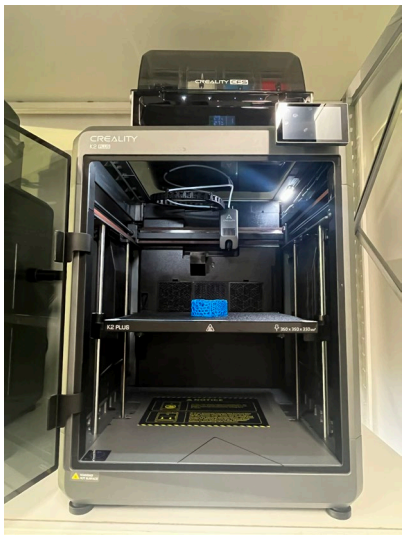
3. After printing

- **Check that the print is really done**
  ! Wait until the tray is automatically lowered in the machine (mid-height).

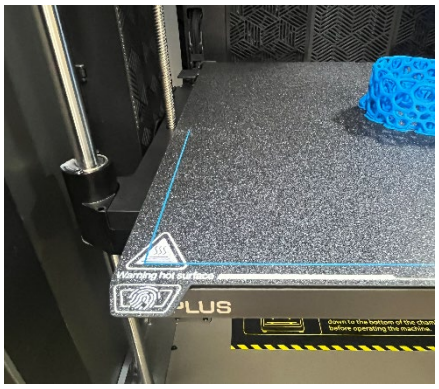
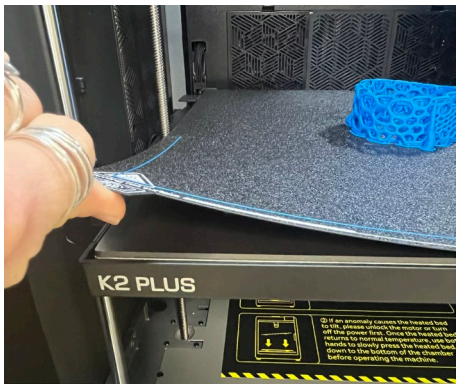




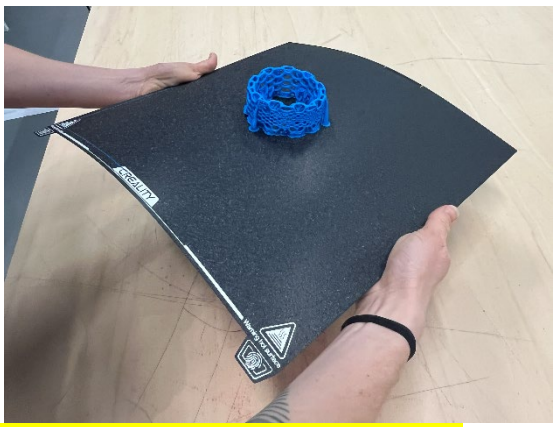
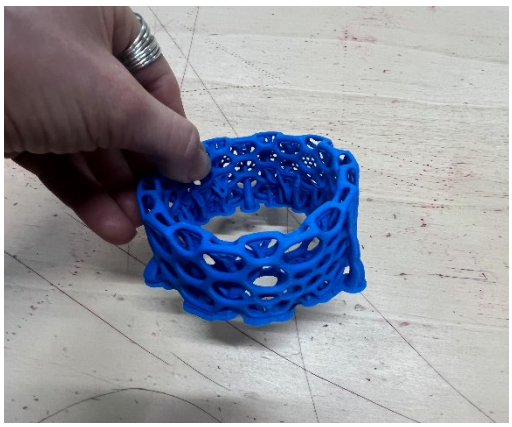
- **Open the door**

- **Push the corners up** and **take the magnetic bed out** of the machine

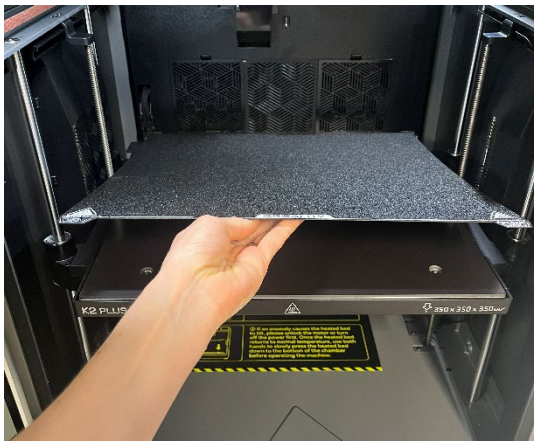



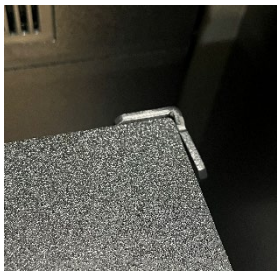
- **Bend the bed carefully** until the object starts to pop

! Do not use any spatula on this bed!

- **Put the bed back in place**





! Check that the corners are properly matching with the structure before placing it down.

- **Make sure there is no residu or object** in the chamber of the machine



! If some waste stay in the machine it could make a crash when starting to print again!

- **Close the door, turn the machine off and take back your USB drive**

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 bin.
- **TURN THE MACHINE OFF** if not already done.

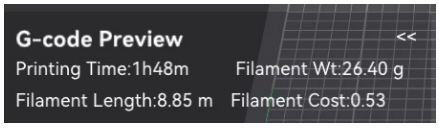
2. Storage

- **STORE TOOLS** in their respective places.
- Make sure you **LEAVE THE WORKING AREA** as you found it when you arrived.

  ! *Inform the fabmanagers if the workspace was not in order, clean and tidy when you arrived at the machine.*

3. Pay your filament consumption!

- **CHECK THE CONSUMPTION OF YOUR PRINT** in CrealityPrint software



! *The cost you see in the software is not the right one.*

- **TELL THE AMOUNT OF MATERIAL** you used to the fabmanager **before you leave the fablab**
- **PAY BY CREDIT CARD** or **DIGITAL WALLET**