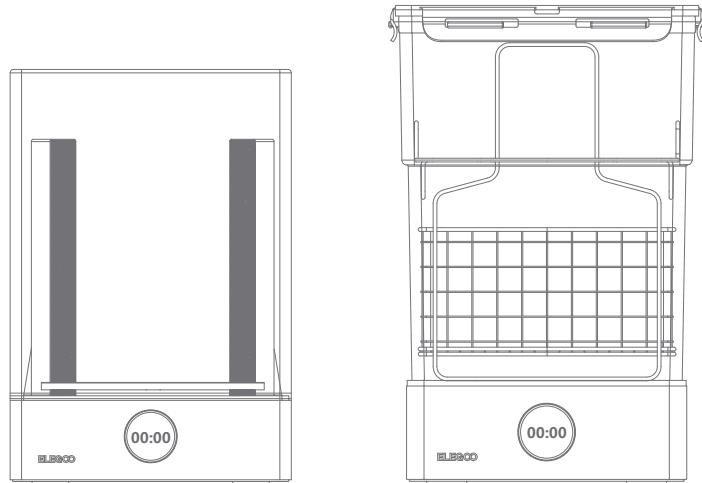
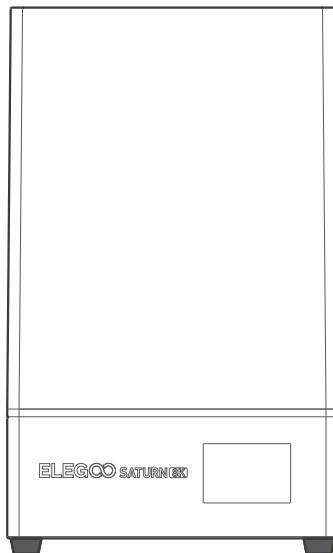


Resin 3D printer

Elegoo Saturn 2 8K



institut
FRANÇAIS
de la
MODE

Table of contents

Fablab safety rules and good practices	3
Safety in 3D printing resin	3
Principle of SLA 3D printing	4
Description of the Elegoo 3D printer	5
1. Elegoo Saturn and Mercury Bundle Diagram	5
2. Technical information	6
Preparation of the 3D model	7
Setting up the printer	7
1. Leveling	7
2. Moving the build plate	9
3. Pouring of materials	9
Preparation of the 3D model	10
1. Selecting the printer and resin in Lychee	10
2. Importing the 3D model and the «Layout» tab	13
3. «Prepare» tab and file settings	14
4. «Export» tab	20
Printing and post-printing	22
1. During printing	22
2. After printing	22
Cleaning and storage after printing	26
1. Machine cleaning	26
2. Storage	26
Troubleshooting	28
1. Missing parts	29
1.1. Raft silhouetting	29

Support de formation	Technique	Modèle machine
1.2. Non-adherence		30
2. Incomplete or damaged parts		32
2.1. Delamination		32
2.2. Cuping blowout		33
2.3. Undeveloped features		34
2.4. Pinholes or cuts		35
3. Poor surface quality		36
3.1. Ragging		36
3.2. Rashing		37
4. Dimensional accuracy		38
4.1. Overcompression		38

FabLab safety rules and good practices

- Closed shoes are recommended.
- 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.
 - o Do not use machines if you are tired or ill.
 - o 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 fabmanager.s 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, fabmanagereuses are there for !
- Do not eat in the Fablab.
- Always know the composition of the material you are using.

Safety in 3D printing resin

The 3D resin printer uses potentially toxic substances, so wearing a mask and gloves is mandatory.

- **Never remove the cover while the machine is working, the UV light could damage your eyes.**

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

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

Principle of SLA 3D printing

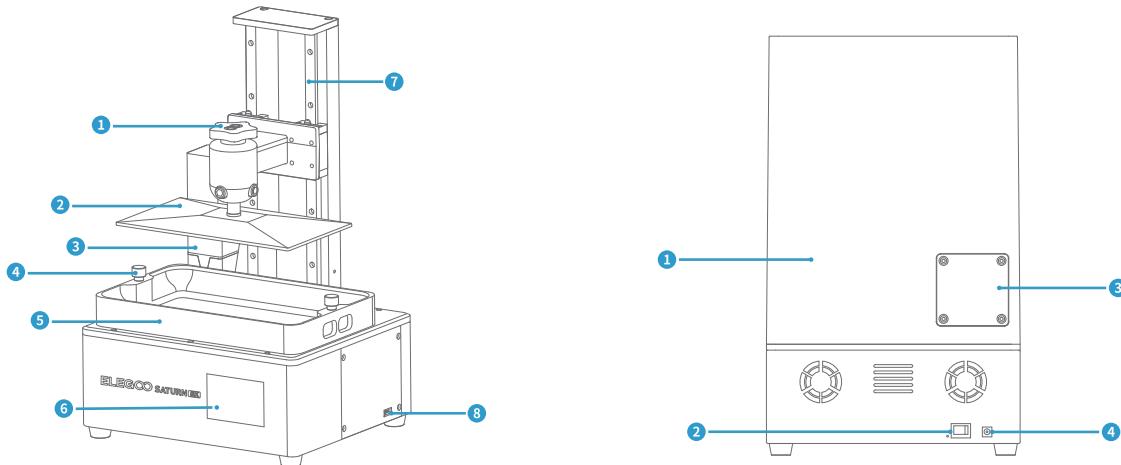
The Elegoo Saturn 3D printer uses the «resin» or SLA (stereolithography) technique. This is another additive manufacturing process.

This technique enables resin parts to be printed using a photochemical process. Using a tank filled with liquid photopolymers solidified by UV light, the object is created by superimposing layers.

After printing, the object must be post-processed: cleaned and polymerized. The part is removed from the platform and excess resin is cleaned with isopropyl alcohol. The resin part then undergoes a UV treatment to achieve its final level of strength and stability.

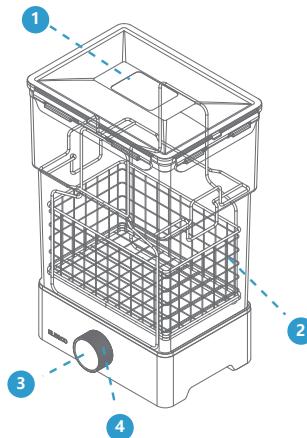
Description of the Elegoo 3D printer

1. Elegoo Saturn and Mercury Bundle Diagram

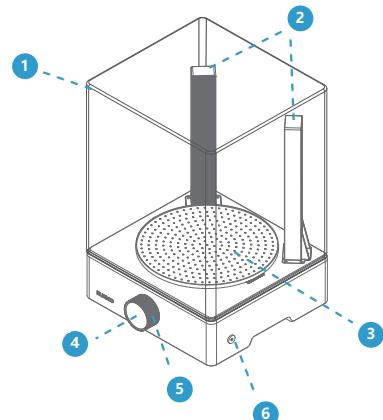


- 1. Rotary knob
- 2. Build Plate
- 3. Air purifier
- 4. Rotary knob
- 5. Resin tank
- 6. Touch screen
- 7. Z-Axis
- 8. USB

- 1. Cover
- 2. Switch
- 3. Vent cover
- 4. Power socket



- 1. Cleaning Bucket
- 2. Cleaning Basket
- 3. Display screen On/Off Button
- 4. Rotary Button for Time Setting



- 1. Anti-UV lid
- 2. Curing light
- 3. Curing Turntable
- 4. Display screen
- 5. Rotary Button for Time Setting
- 6. Handheld UV Lamp

2. Technical information

- 3D printer printing dimensions: W 218 x D 123 x H 250 mm
- Maximum cleaning volume with platform: 180 x 121 x 153 mm
 - without platform: 201 x 124 x 255 mm
- Maximum curing volume in machine: Φ 200 x 260 mm
- Printing materials: standard UV resins, water washable, lost wax, vegetable-based, thermoactive/ thermochromic, flexible or rigid, with or without pigments.

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.).

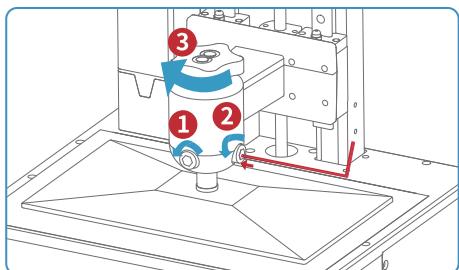
The modeling software used does not matter, however it is essential that the format of the 3D model is in. STL.

! Be careful to take into account the machine's maximum printing dimensions.

Setting up the printer

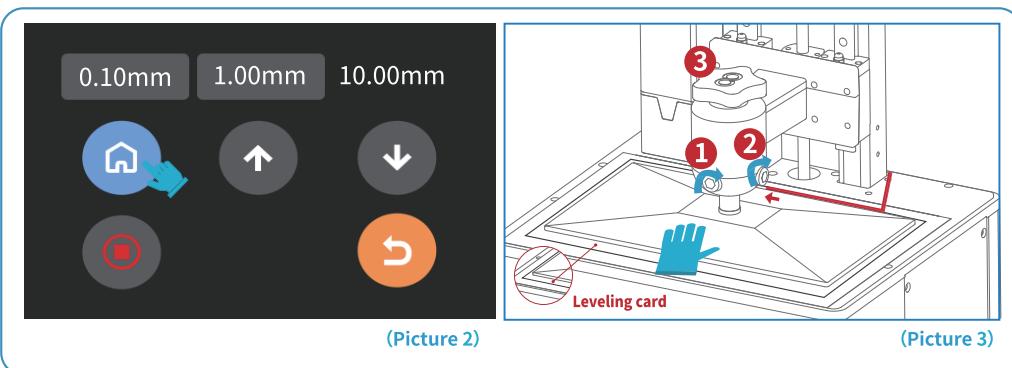
1. Leveling

! Check with the fabmanager first if this step has been done already.



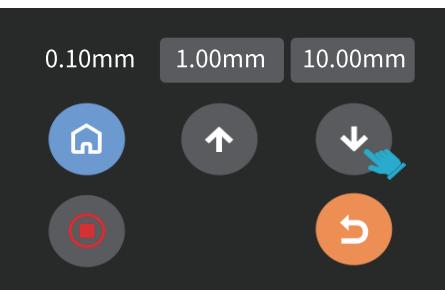
Take out the resin tank carefully, first fasten the rotary knob of the machine build platform, then loosen and level the two screws in turn with an M8 Allen wrench.

Put the leveling card between the build platform and the LCD screen, touch the screen to click «Move Z axis to zero». (Picture 2) When the build platform stops moving, use an M8 Allen key to tighten the screws in the order of the serial number. (Picture 3)

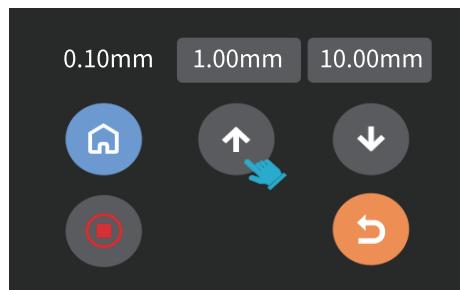


Since the distance between the build platform and the screen will be changed during the process of tightening the screws of the build platform, if you find that the leveling card can be pulled out with no resistance, please click the «down» button (step value is 0.1mm) until there is slight resistance to pull out the leveling card. (Picture 4)

If you find the resistance of pulling out the card is too high, please click the «up» button (step value is 0.1mm) until the card can be pulled out with slight resistance. (Picture 5)



(Picture 4)

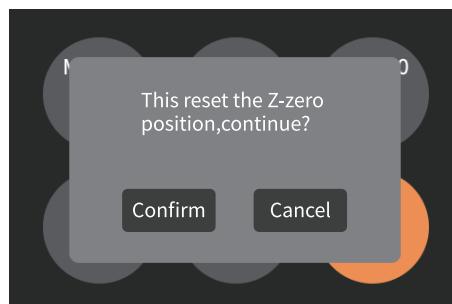
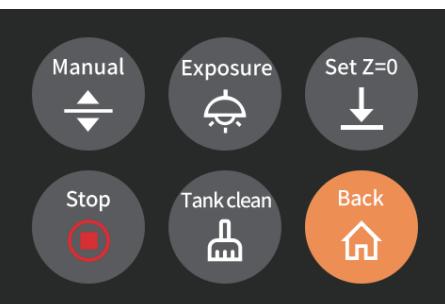


(Picture 5)

After finishing the above leveling, set the current position of Z-axis as the initial height of the first printing layer. The operation is as follows: Return to the previous interface and click «Set Z=0».

At this time, a message will pop up on the screen as shown in the picture.

Then click «confirm» to finish. (Note: «Set Z=0» only takes effect in print file mode.)

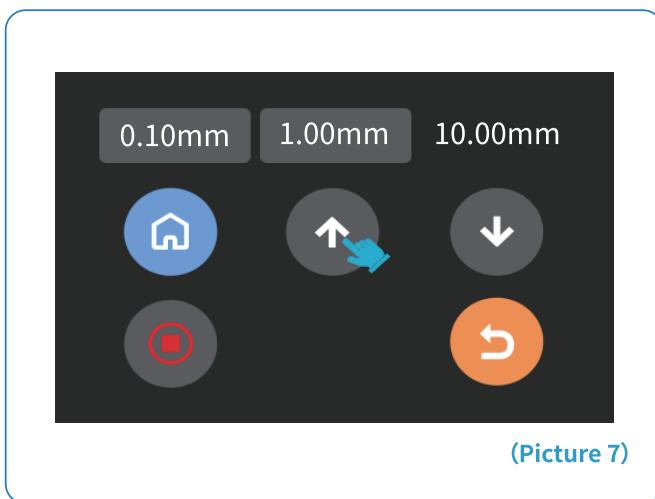


(Picture 6)

2. Moving the build plate

Press the «up» button on the touch screen 10 times until the build platform rises to 100mm. Every time you click the «up» button on the touch screen, the Z-axis will rise 10mm. (Picture 7)

If you did a leveling before, put the resin tank back and fasten the screws to ready for printing.



3. Pouring of materials

Be carefull that the resin tank is placed and fastened tightly. Wear a mask and gloves (avoid direct contact with the skin), and then add resin slowly, making sure that the printer is level and will not wobble. Finally, cover the machine with the anti-UV cover and plug the USB key.

Note: If the resin is not enough to compete the model in the printing process, you can press «pause», and add more resin to the tank, the press «print» to continue printing.

WARNING: If there is already resin in the tank, check that there are no residues that could damage the machine and the print.

Preparation of the 3D model

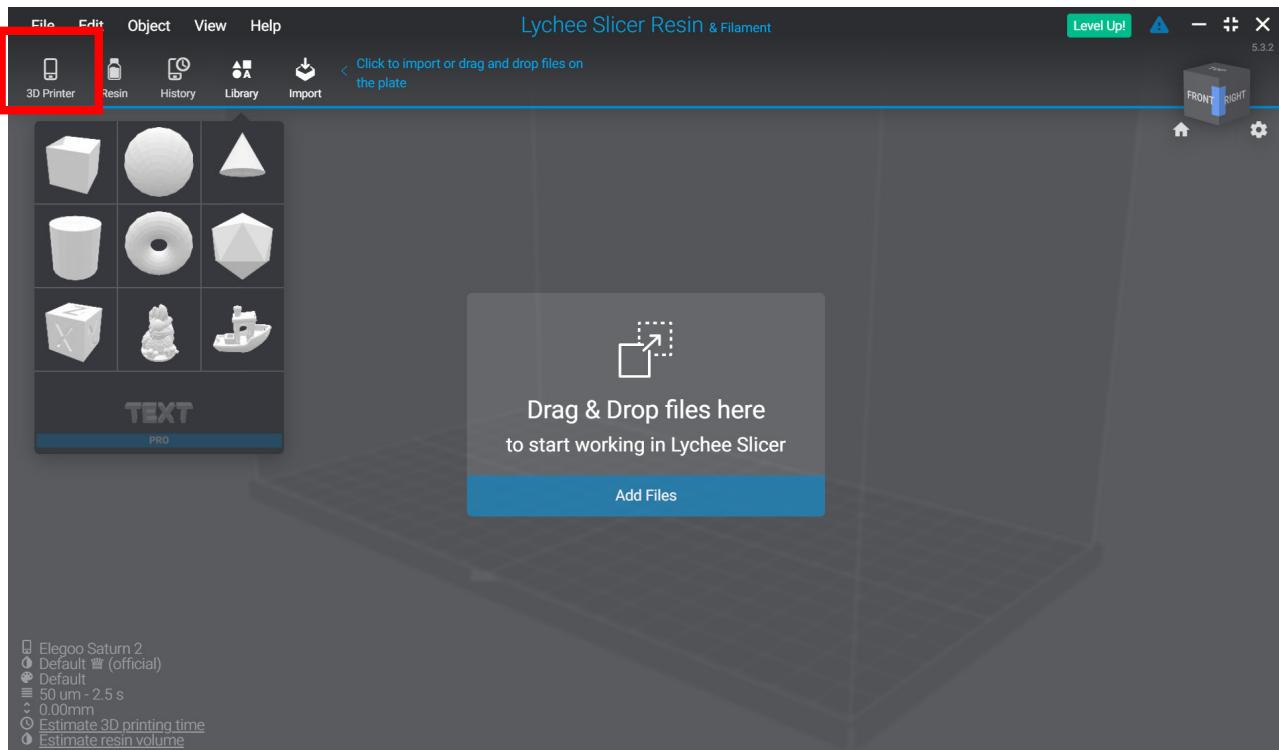
A 3D print can only be made from a 3-dimensional model prepared in advance with a modelling software (Rhinoceros, 3DSMax, Fusion 360, Blender, etc.). It is essential that the format of the 3D model is in STL.

The slicing software allows you to go from the 3D model file in .STL format to the file containing data that can be printed by the machine, the G-code file.

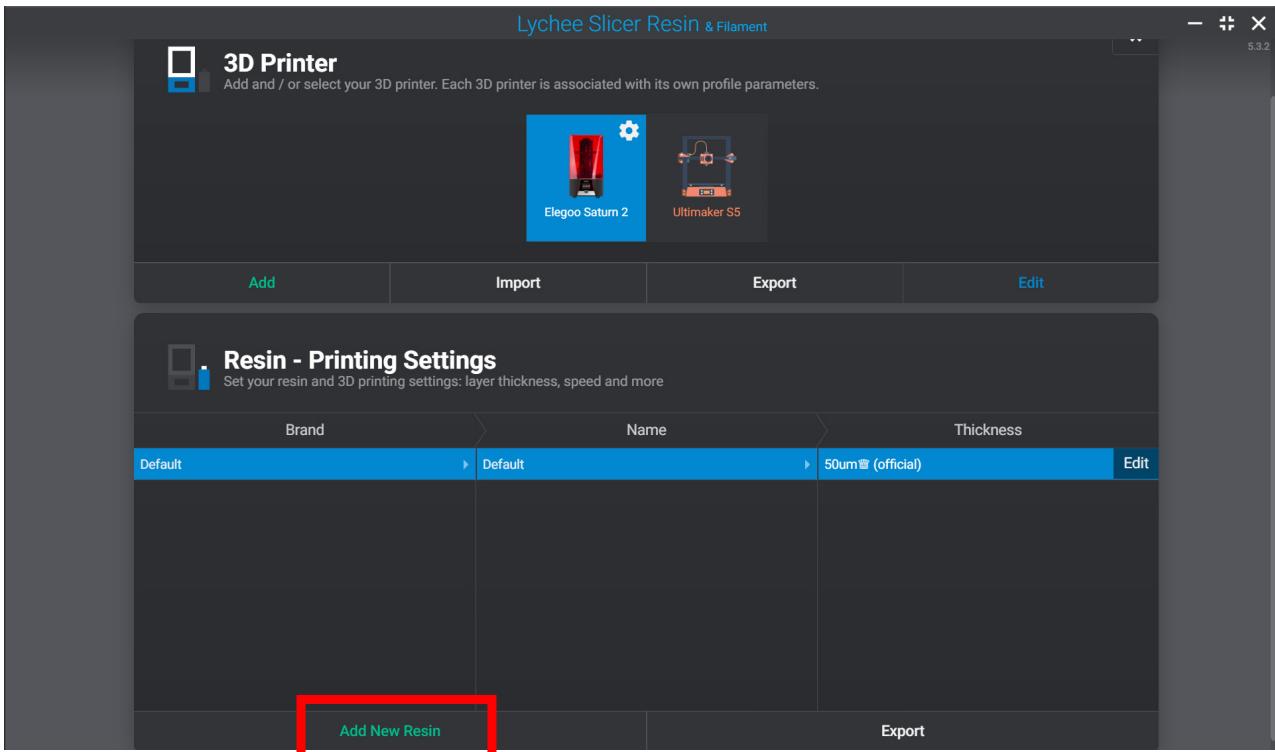
Slicing is the operation that consists of translating a volumetric model into a succession of textual information (code). The information contained in the G-code gives the machine precise instructions such as polymerisation coordinates, layer thickness, build plate speed, exposure time, etc...

1. Selecting the printer and resin in Lychee

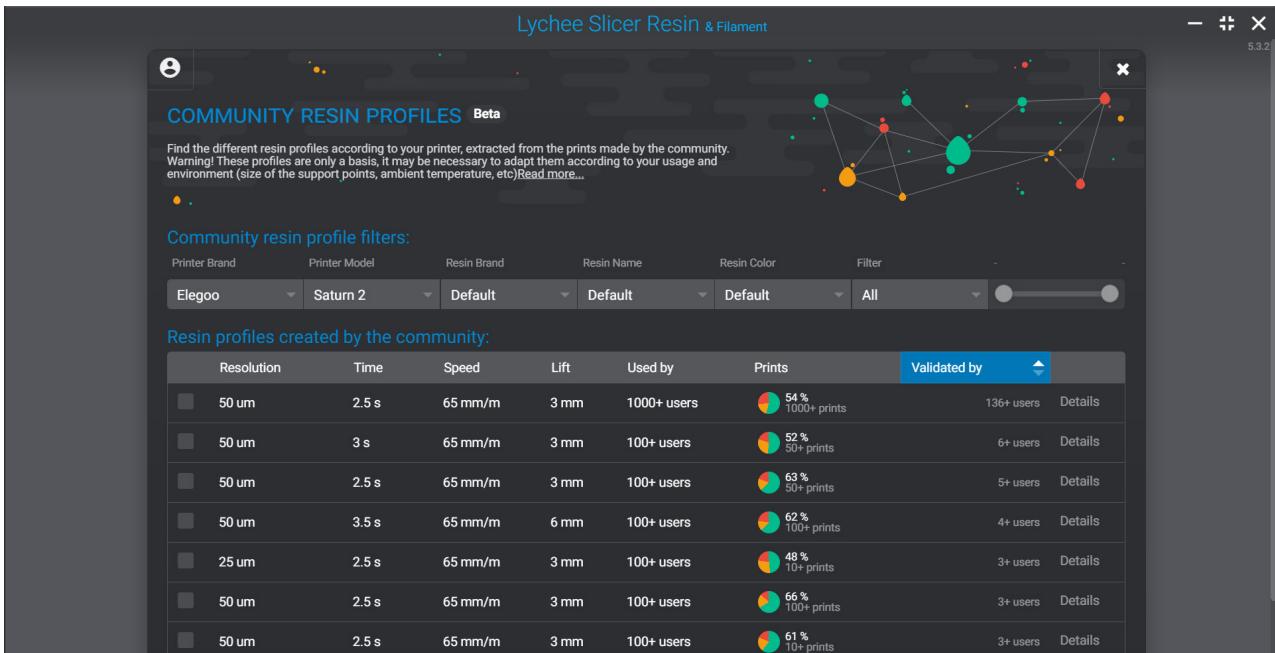
For Elegoo printers, printing and slicing settings are made using Lychee software, which can be installed free of charge on PC, Mac and Linux.



- To choose the printer you want to work with: click on the tab «3D printer»
- Here, the printer is in the category Resin and is called Elegoo Saturn 2.

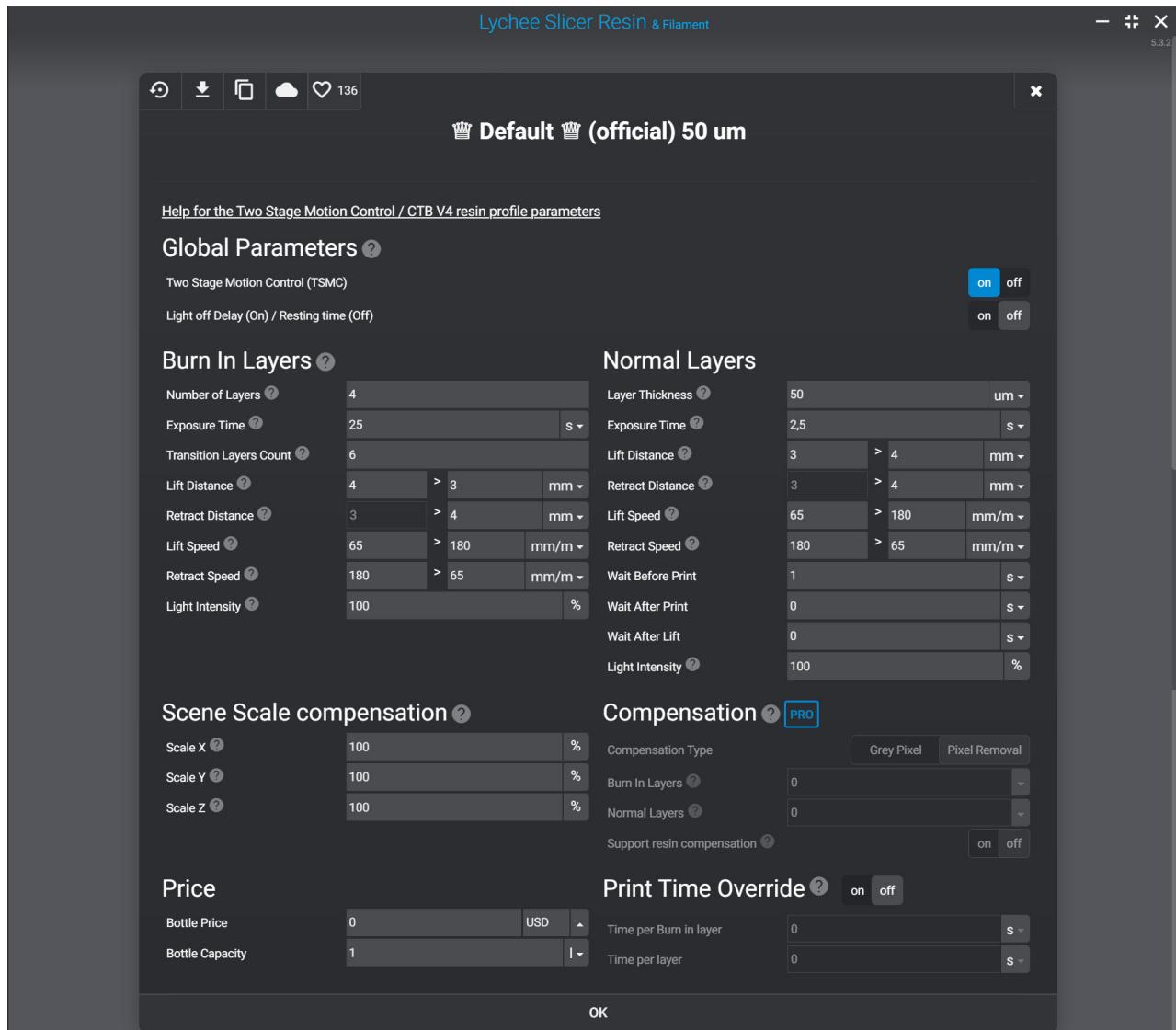


- Then select the type of resin you are going to use in the «Add new resin» tab.
- What's special about Lychee is that you can choose your resin profile from a database compiled by the software's community of users.



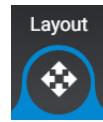
Note that you will still need to check the resin settings depending on the profile and resin you are using.

To do this, simply click on the icon  and various modifiable settings will appear.

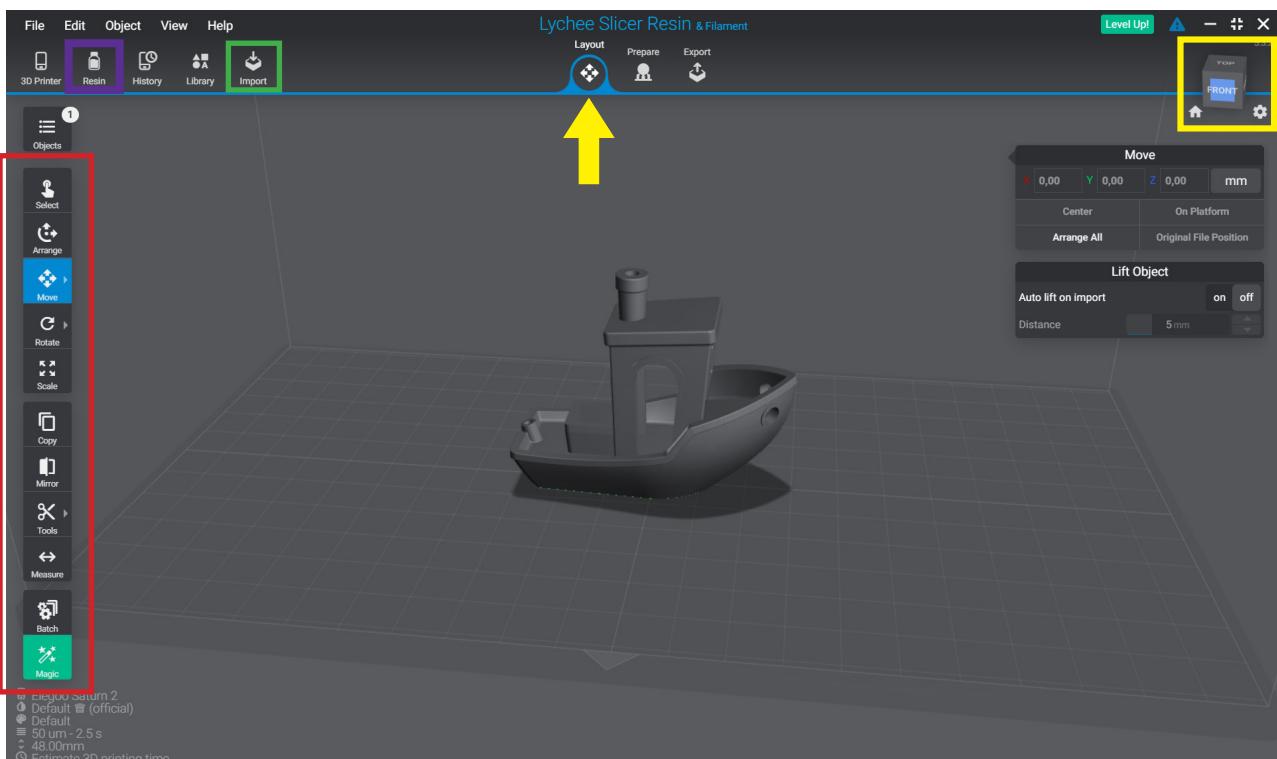


- Layer thickness:** The thickness of each printed layer; Recommended height is 0.05mm but you can set it from 0.01-0.2mm. The higher you set, the longer time it will take for exposure time of each layer
- Exposure time:** The exposure time of the normal printing layer; The default exposure time is 2.5 seconds.
- Lift Speed:** In the normal printing process, the movement speed of the build platform away from the printing surface each time; the default setting is 65&180mm/min.

2. Importing the 3D model and the «Layout» tab



- IMPORT THE .STL file** to be printed by clicking on the icon  or by dragging the file directly in the Lychee window. The model then appears in the viewing area of the «Layout» tab. This gives access to all the settings that will enable the user to modify the 3D model for optimal printing.
- In particular, it is strongly recommended that you **TURN** the 3D model a little, as it is preferable to have some material on the tray gradually, so that the model can be removed easily and does not fall into the tray during printing.



Open a new file

Viewcube

Model editing tools

Resin settings

Notes:

- Scroll the mouse wheel to zoom in or zoom out the model
- Long press the right click to see different perspective of the model
- Long press the right click +Shift to drag the model to the position you want

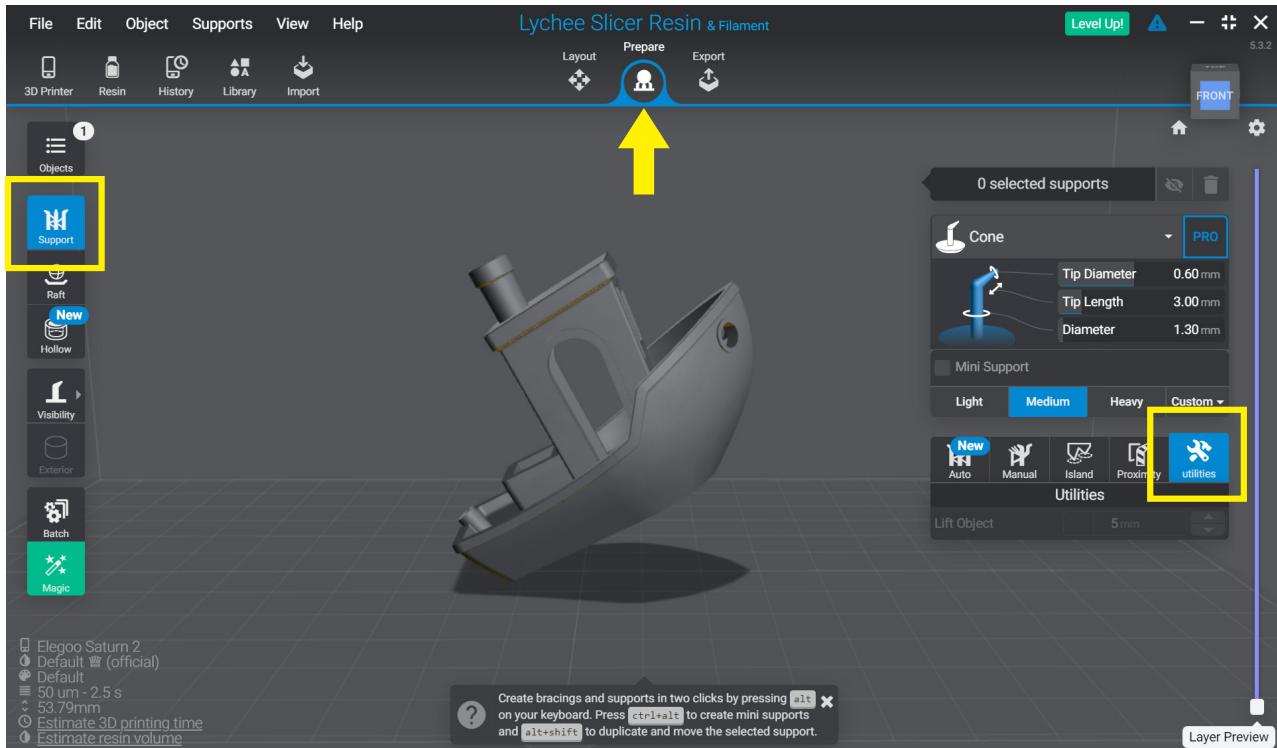
3. «Prepare» tab and file settings



This tab gives access to all the settings that will enable the user to prepare a printout. It is essential to go through this stage if you want to print successfully.

- **Supports** 

A setting that allows Lychee to generate supports during printing when the part contains overhangs with an angle of incidence greater than 60 degrees. The supports are only used during printing (like scaffolding during construction) and will be removed after printing.



First of all, it's best not to print directly on the tray. To lift the model off the tray, go to the Utilities tool  then press the top arrow at «Lift Object».

This avoids the risk of damage, local swelling of the resin and overexposure of the first layers. In fact, the tray is only used for adhesion and not for construction purposes.

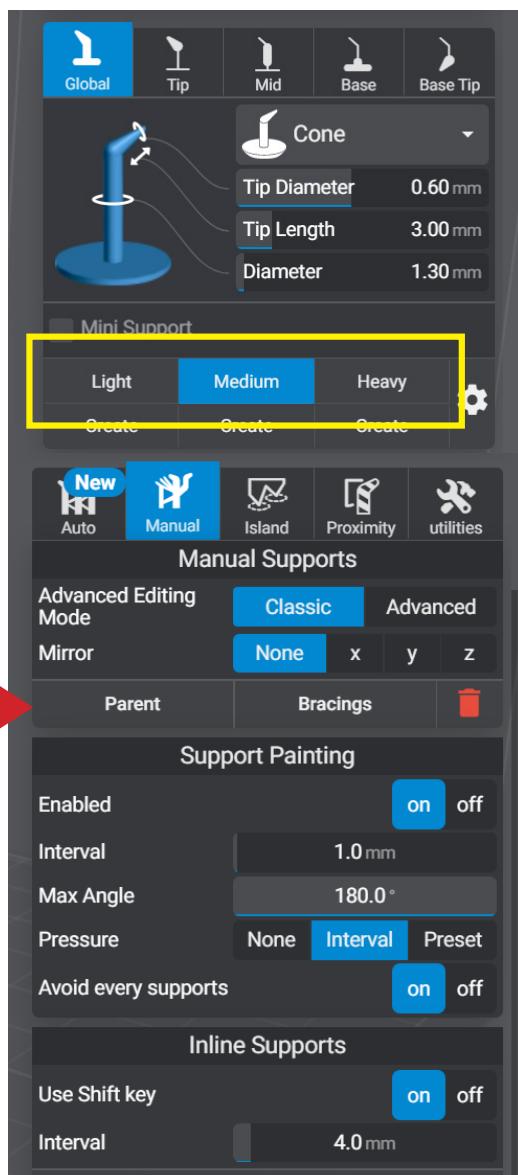
To add support :

- Select the support type (Light, Medium or Heavy)
- Click on the Auto icon  to generate supports automatically
- You can add or subtract supports in manual mode 

Notes: It is better to have too many supports than too few.

The further they are from the centre, the more stable the structure will be.

You can also place a few heavy supports in strategic places to stabilise the structure.

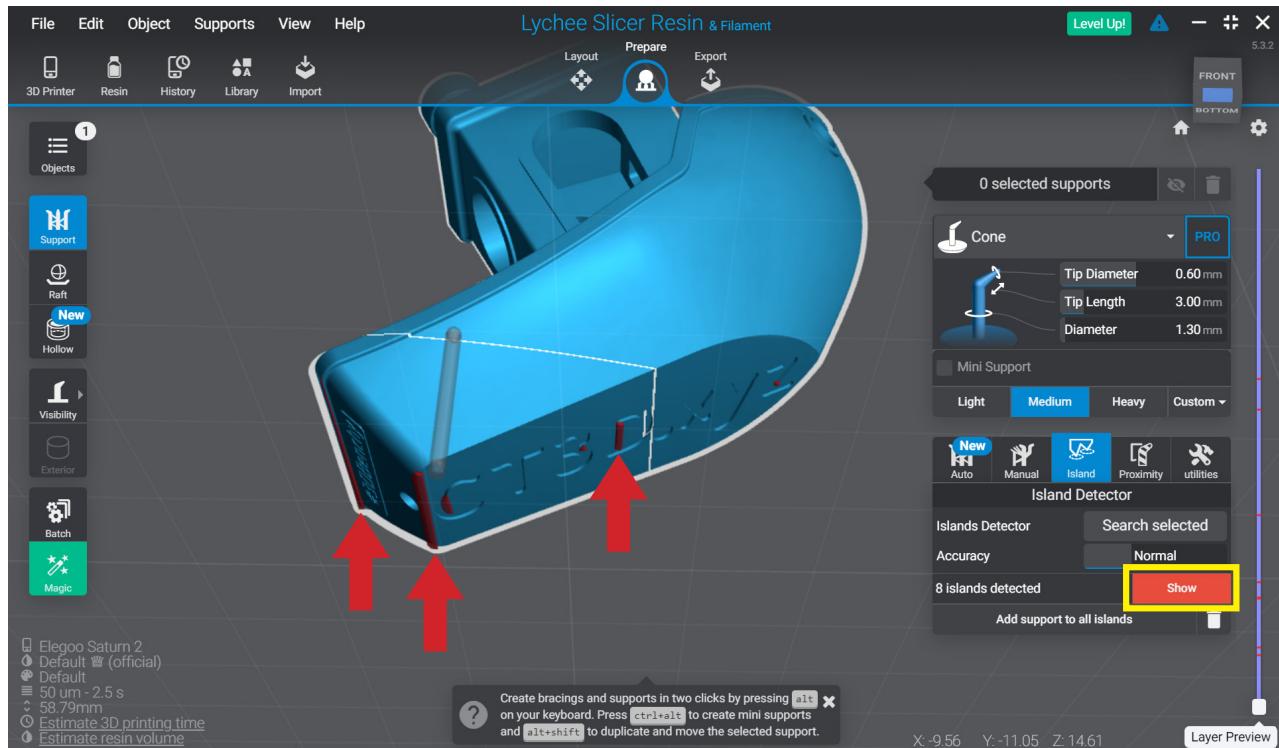


- **Parent:** Connect multiple supports to a single support pillar.
- **Bracings:** To solidify the structure of the supports (when they are too high and too thin, for example), you can bind them together.
- **Support Painting (Pro):** Create supports by holding down the *left click*, like a drawing directly on the model
- **Inline Supports (Pro):** Adds supports at regular intervals in a straight line, holding down the *shift key*.

Other useful tools :

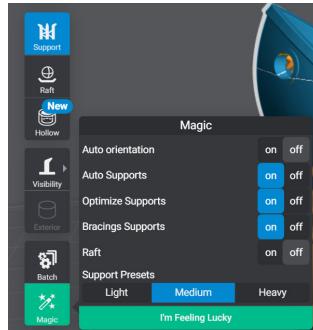
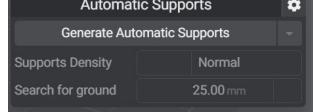
- Island detector: When in doubt, the icon  can be used to detect «essential support» islands in the 3D model. These areas will be considered by the software as potentially risky for printing and will be marked in red.

You can scroll through them by clicking on *Show*.



- Automatic support generator:

Using the *Magic icon*  or *Auto icon* , it is possible to generate supports automatically and then fine-tune them according to the needs of the model and risky areas.



- **Raft**

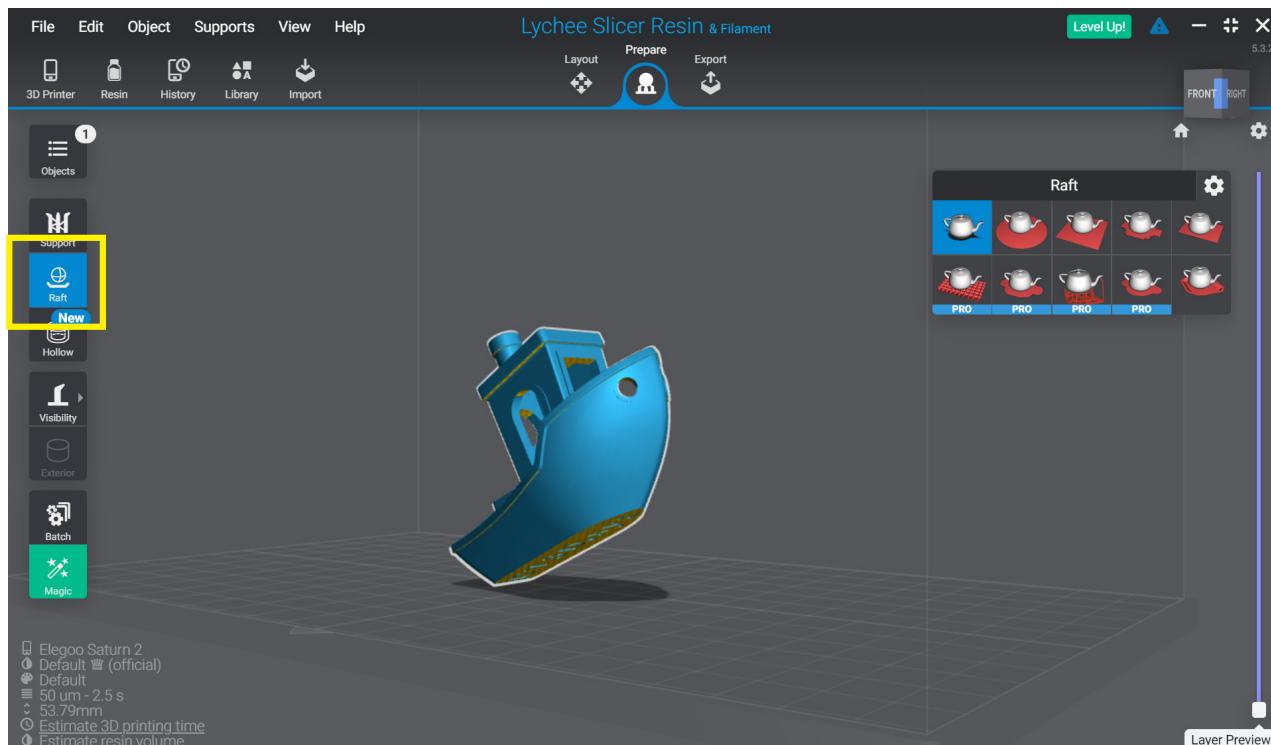


The purpose of the raft is to improve the adhesion of the model and the supports to the build plate by adding a fine structure, in order to increase the success rate of the print.

It can also help to remove the print from the build plate without breaking the supports.

Please note that depending on the type of raft used, this may increase resin consumption.

Line triangle is one of the best choices for achieving a good balance between good adhesion and minimum resin consumption.



None/Disabled



Grid: Thick grid in the form of a rectangle



Cylinder: Shape of a circle around



Pixelate Round: Big projection



Cube: Shape of a square around



Line triangle: connect supports together



Pixelate: Projection in the shape of pixels



Shape: Small projection



Rectangle: Shape of a rectangle around



Shape Wall: Small projection with curling of edges

- **Hollow / Holes**

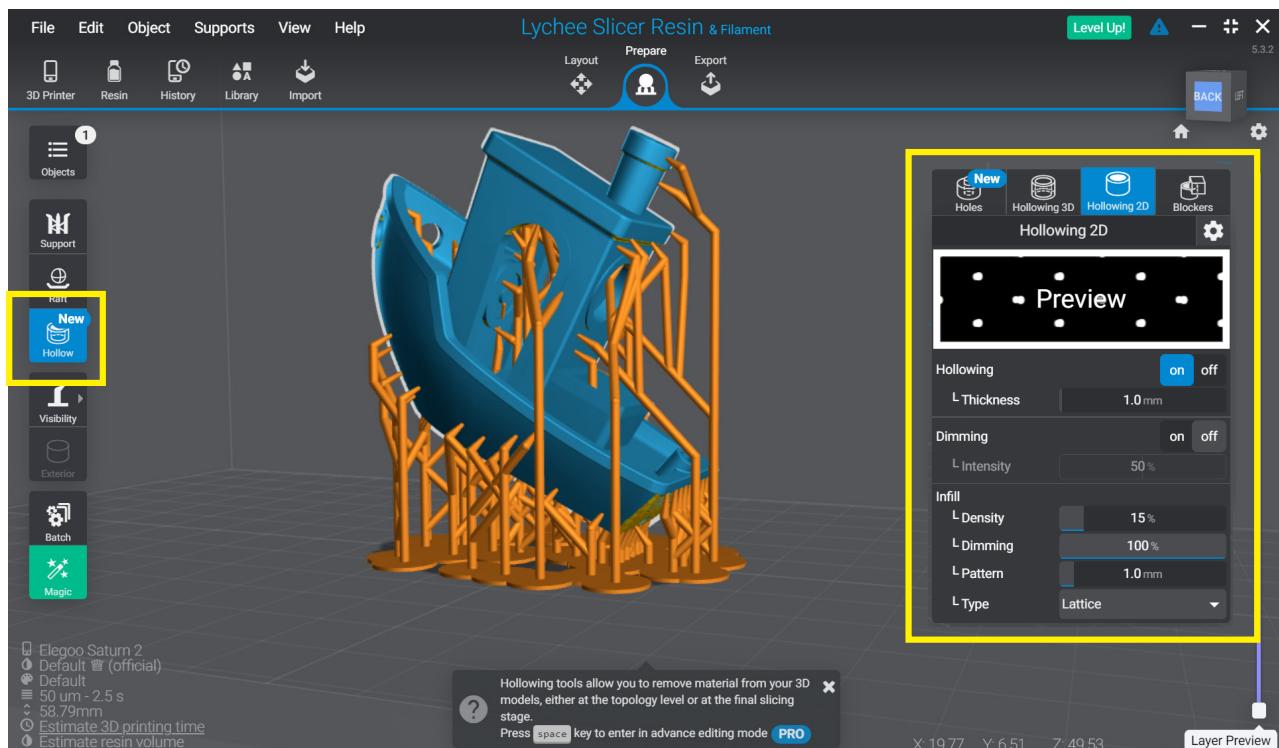


Hollowing 2D et 3D : This option creates a hollow inside a solid object.

2D Hollowing creates a hollow with a wall thickness and internal supports that can be determined in the window on the right.

3D Hollowing provides additional parameters such as more precise hollowing and an export of the new hollowed object, which can then be modified using CAD software.

It also changes the resin estimate used and the preview inside the object, which is more accurate.



- **Hollowing -Thickness**: Wall thickness
- **Dimming**: UV intensity on the internal wall
- **Infill -Density**: Internal support intensity
- **-Dimming**: UV intensity for the internal supports so that they polymerise more softly
- **-Pattern**: Thickness of the internal support

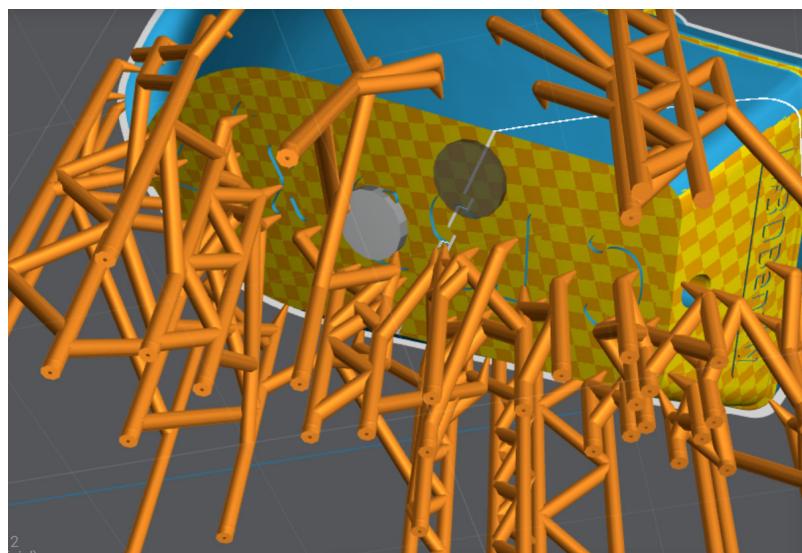
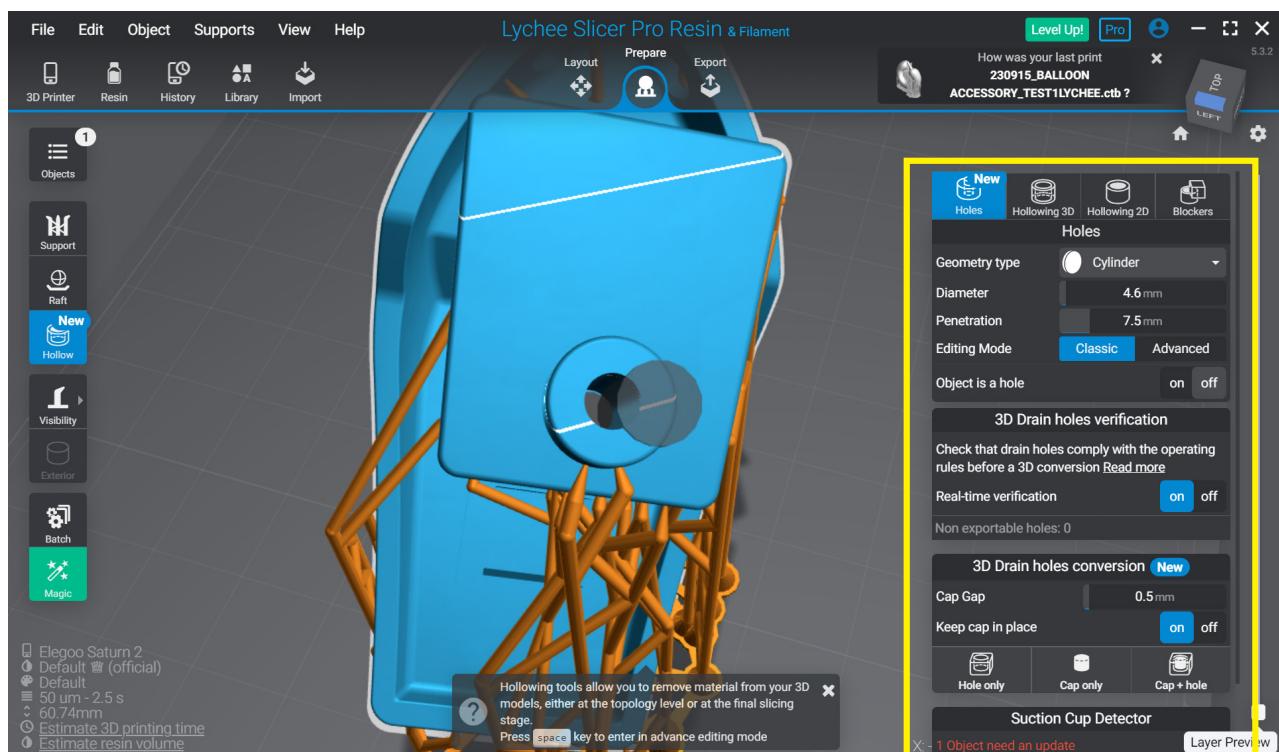
Notes: Blockers: Removing the hollowing in specific places

With Hollowing 3D, you will need to create internal supports manually (islands detector and bottom on the layer selector).

Holes : It is necessary to release internal pressure, but also to allow uncured resin to escape through holes:

- At the bottom: To avoid the «suction effect» and let out air. And for UV rays to reach the inside of the model during curing.
- At the top: «Drain holes» to let out resin trapped inside the object. The more viscous the resin, the larger the hole should be.

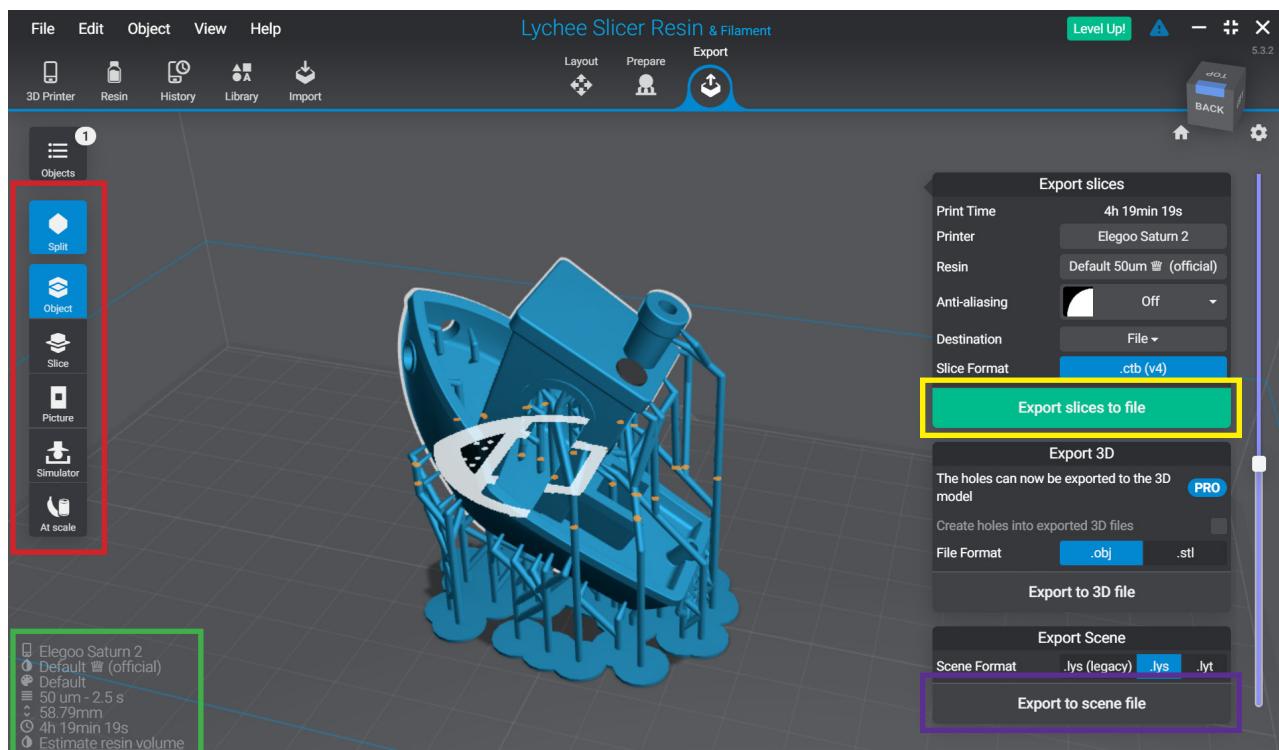
Note that if your object does not allow for discreet drain holes, you can evacuate the resin once you have finished printing by turning the object slightly.



4. Onglet «Export»

In this area, the user can simulate his/her printout and obtain several important information about it. It is also the area where you export your file for the machine.

- **ANTICIPATE** your printing thanks to the preview mode: 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 highly recommended as it will allow you to anticipate errors, save material and time.
- 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.



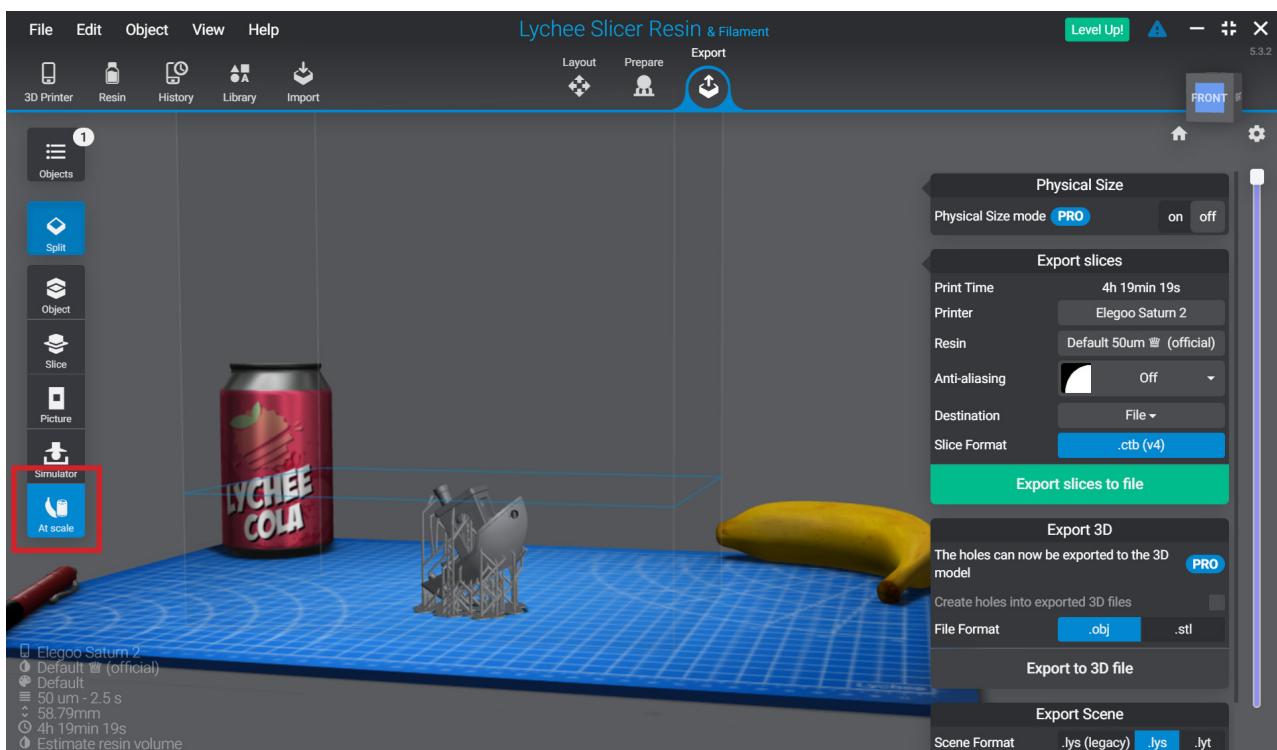
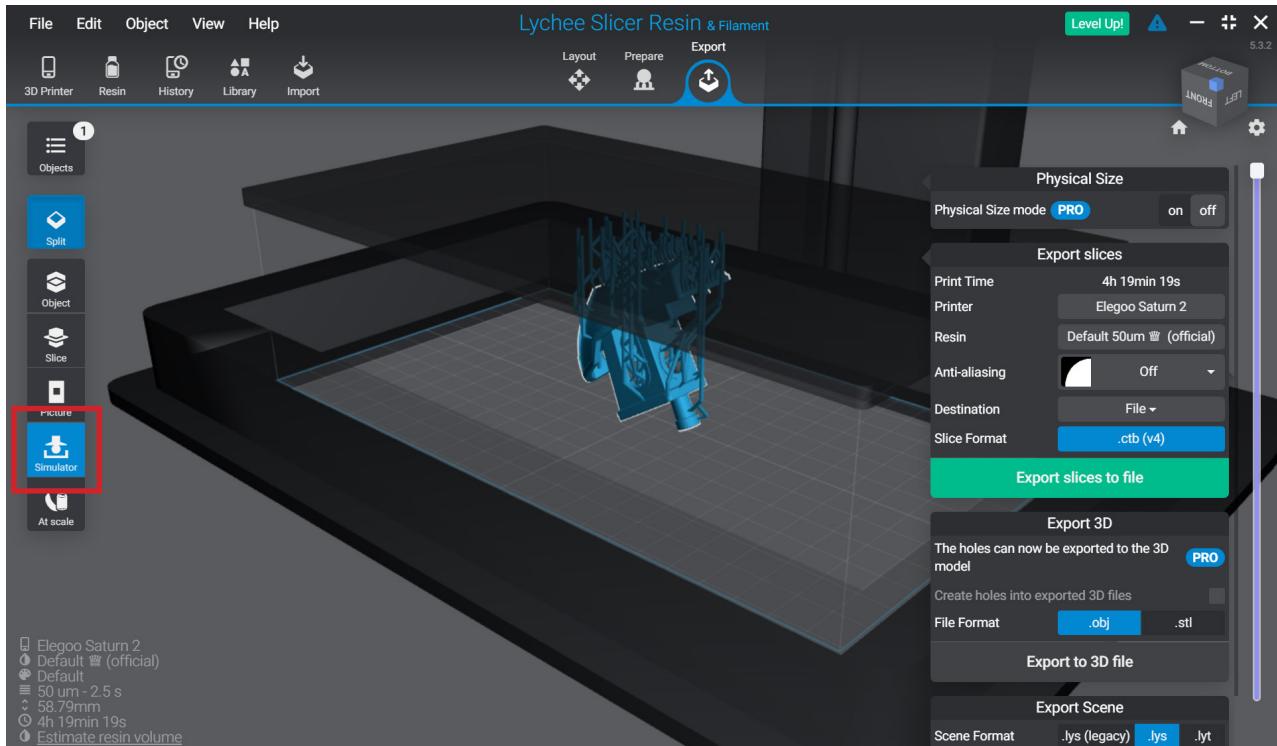
Estimate resin volume

Export to print

Preview mode

Save your file

- **Estimate resin volume:** it gives you the amount of resin the print will use.



Note: If you need more information about the 3D Lychee Slicer, you can find them here.
<https://docs.mango3d.io/docs/lychee-slicer-resin/>

Printing and post-printing

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:

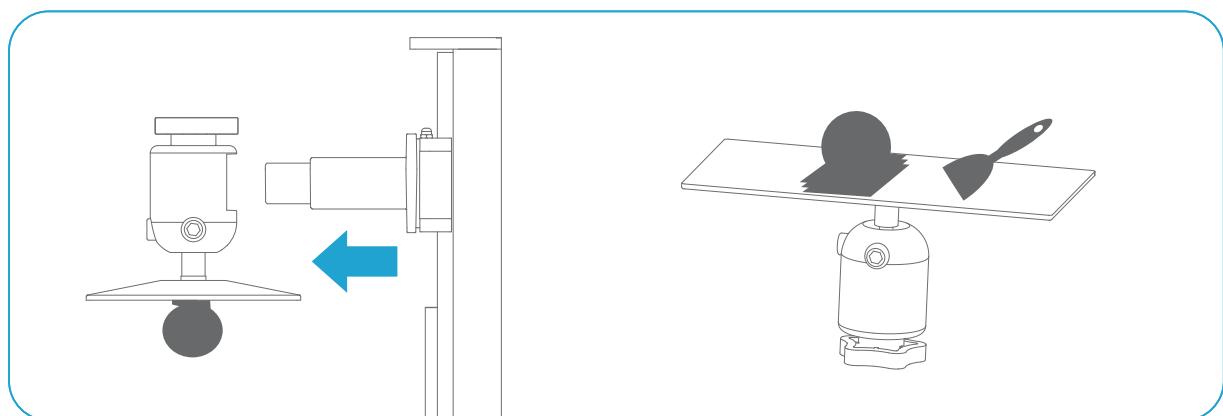
- **Damage to the machine** : if another person touches the machine while it is in use, if an external element is present inside the machine enclosure, if there are residues in the tank or the print fails and falls off the plat; the machine may be damaged and will not be able to operate until it is repaired, 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 your print fell but the UV light at the bottom of the machine will continue to polymerise some of the resin in the tank. Polymerised resin is not recyclable and must therefore be thrown away. If you monitor the printing process, you can stop if there is a problem more quickly and save 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

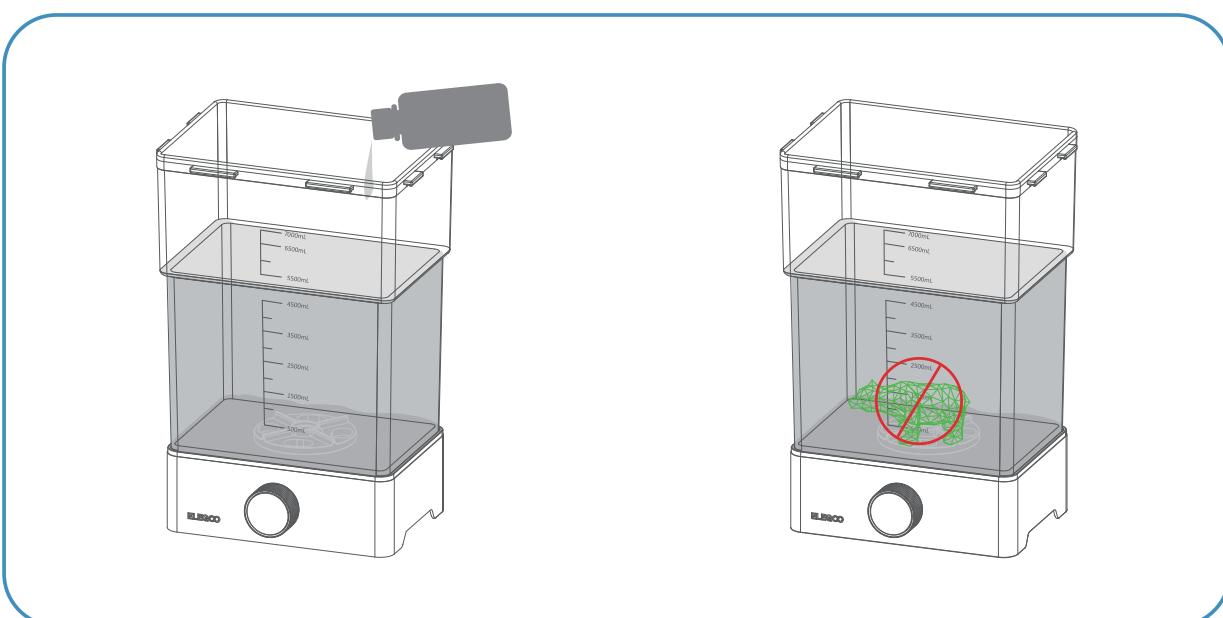
- **Wait** until the residual resin on the build platform doesn't drop any more, and try to take most of it out with the scrapper.

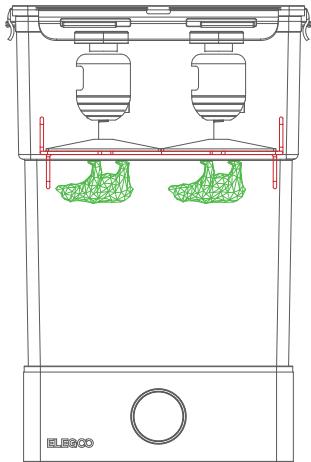


- **Loosen the rotary knob:** carefully remove the build platform. Use the scrapper to remove the model and wash it with ethyl alcohol if you are using standard resin. If you use water washable resin, you can directly wash it with running water. (rinse water must be stored in a container).



- **Clean the model using Mercury Wash:** Submerge the model in a suitable way in the washing liquid of the cleaning bucket.
- **!** It is forbidden to put the model directly into the cleaning bucket for cleaning, otherwise, the model and the parts of the cleaning bucket will be damaged.



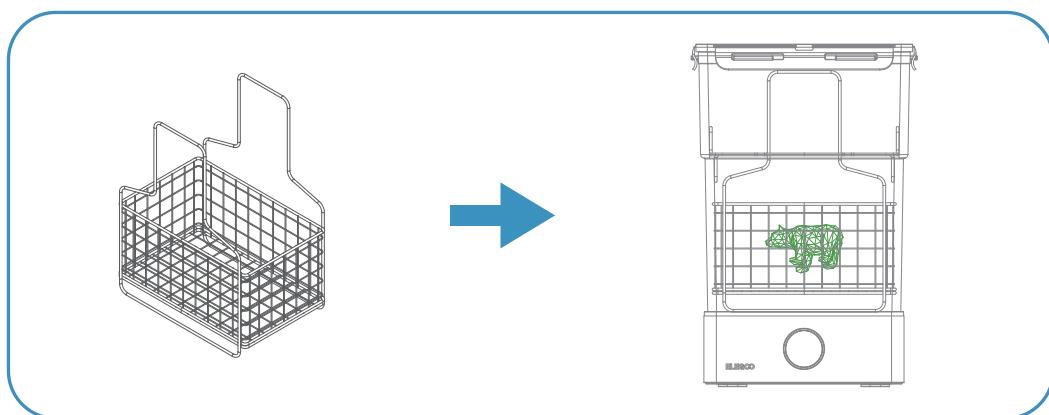


When the model *is within the size* of 180 x 121 x 153 mm, you can wash the model with the build platform. Place the platform steadily without the cleaning basket on the side of the bracket, so that the model is totally immersed in the alcohol and seal the lid.

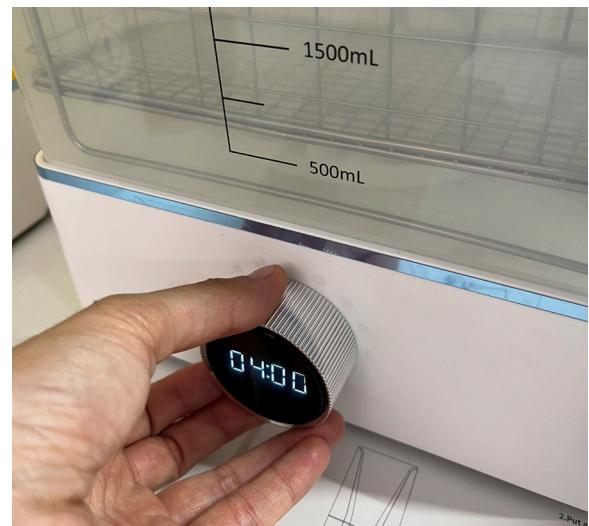
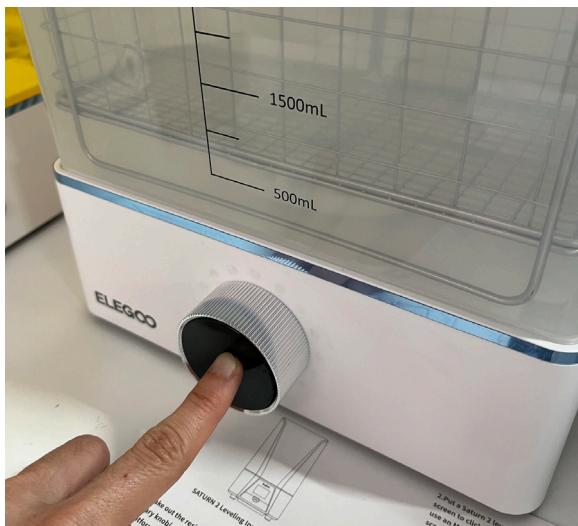
⚠ Be careful not to bump the model during placement.

When the model *is out of the size* of 180 x 121 x 153 mm, then you should clean the model without the build platform. Remove the lid of the cleaning bucket first, put the cleaning basket into the cleaning bucket, take the model off the platform directly and put it lightly into the cleaning basket and finally seal the lid.

Be sure that the model is placed as flat as possible in the cleaning basket to prevent the sharp corners of the model from going over the grid at the bottom of the cleaning basket and hitting the wheels.

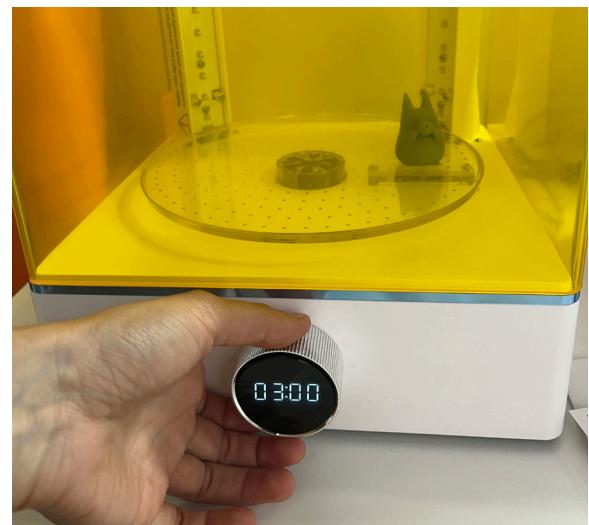
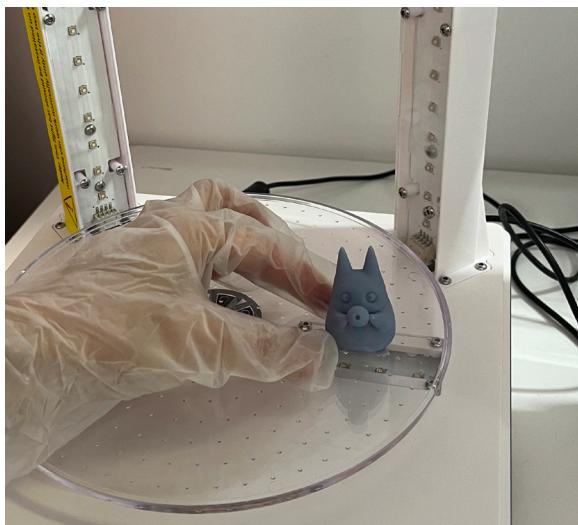


- Turn the knob clockwise to set the cleaning time, and press the screen briefly to start cleaning the model. The machine will beep when the cleaning is finished, you can take out the model for air-drying before curing.



- **Cure the model using Mercury XS CURE:**

- Place the air-dried model in the center of the curing turntable and put the anti-UV lid on.
- Turn the knob clockwise to set the curing time and press the screen briefly to start curing the model. (Due to the slow rotating speed of the turntable, there is slight shaking, which is a normal phenomenon). The machine will beep when curing is completed.



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 resin and clean the workspace.
If the plateform was not clean with the model in the Mercury WASH machine, CLEAN THE PLATEFORM using alcohol before putting it back in the machine.

2. Storage

- **STORE TOOLS** (scraper, spatula, gloves, etc.) in their respective places.
- Be sure to **LEAVE THE WORKING AREA** as you found it when you arrived.

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

TROUBLESHOOTING

Diagnosing a print failure

Sometimes prints don't come out the way you expected. Compare common symptoms to narrow in on the underlying source of the problem and find a solution.

1. Missing parts

- 1.1. Raft silhouetting
- 1.2. Non-adherence



2. Incomplete or damaged parts

- 2.1. Delamination
- 2.2. Cuping blowout
- 2.3. Undeveloped features
- 2.4. Pinholes or cuts



3. Poor surface quality

- 3.1. Ragging
- 3.2. Rashing



4. Dimensional accuracy

- 4.1. Overcompression

1. Missing parts

Les défauts d'impression suivants entraînent l'absence totale de toute pièce imprimée. La plateforme de fabrication est laissée presque entièrement vide.

1.1. Raft silhouetting

“Raft silhouetting” describes a print defect in which the early layers adhere to the build platform, creating a silhouette of the raft or initial layers. After these initial layers that create the raft silhouette, the remainder of the part does not print.

Contamination, dust, or debris on the printer’s optical surfaces can diffuse or weaken laser light, causing a print failure where only the early layers of the print adhere to the build platform.



Visual symptoms:

- Only the first few layers adhere to the build platform.
- The rest of the part is missing from the build platform, and may be present as cured resin in the tank bottom or absent entirely.

Common causes:

Raft silhouettes are usually caused by optical surface contamination that blurs or weakens the laser light before it reaches the resin.

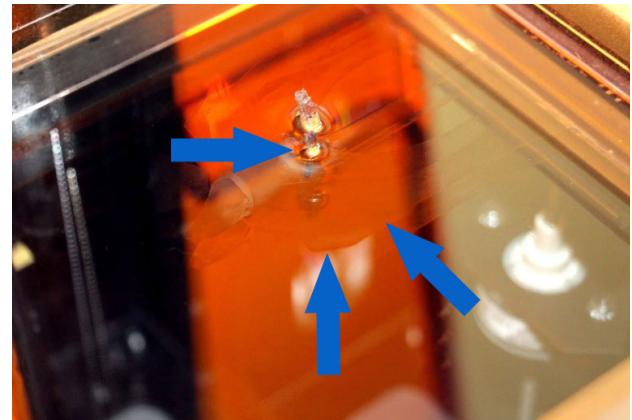
The early layers of a print receive additional laser curing, so these layers can cure successfully even if contamination or clouding weaken the laser. Further layers may fail to cure, either partially or completely.

Any of the following factors, or a combination of them, can blur or weaken laser light:

- Debris/clouding/damage in resin tank
- Contaminated optical surfaces
- Resin tank acrylic window
- FEP film
- Glass optical window

1.2. Non-adherence

“Non-adherence” describes a print defect in which prints are either partially or completely detached from the build platform. This occurs when the initial layers of a print fail to adhere to the build platform.



Right: Arrows point to remains of a part left inside the resin tank.

Left: Print missing completely from the build platform.

Visual symptoms:

- Complete non-adherence (at print start):
 - The print is missing from the build platform and there is a flat area of cured resin on the bottom of the resin tank.

- Partial non-adherence :
 - The print's raft partially separates from the build platform.
- Complete non-adherence later in print :
 - The print has formed partially, but has fallen into the tank during the print, leaving nothing on the build platform.

Common causes:

Any of the following factors, or a combination of them, can cause non-adherence in prints :

- Printing without a raft, when the part has no suitable flat surface to grip the build platform.
- The first layer of the part on the build platform is too small to withstand peel forces.
- Debris/clouding/damage in the resin tank weakening the printer's laser.
- Contaminated optical surfaces weakening the printer's laser.
- Excessive space between the build platform and the elastic layer or the flexible film in the resin tank.

2. *Incomplete or damaged parts*

The following print defects result in a “completed” print, but the printed parts often show damage or have chunks missing.

2.1. *Delamination*

“Delamination” describes a print defect in which the cured layers of a print separate from one another. Delamination can occur on any type of 3D printer, though the causes are unique to each print process.



Visual symptoms :

- Peeling or separation between printed layers
- Pieces of cured parts floating in the resin tank

Common causes :

Any of the following factors, or a combination of them, can cause delamination in prints :

- Model orientation, layout, or support issues
- A print that has been paused for an hour or longer
- An older resin tank
- Loose build platform
- Contaminated optical surfaces

2.2. Cupping blowout

“Cupping” is a concern unique to upside-down stereolithography (SLA) printing. A print failure caused by unresolved cupping is often referred to as a “blowout.” This occurs when a hollow or convex portion of a part acts as a suction cup and traps air while printing. As the build platform pulls away from the tank during the peel process, the empty space within the cup increases. This reduces the air pressure within the cup to create a pressure differential that pushes inward on the surrounding cup walls. If the walls are too thin and unable to withstand the pressure, then they buckle inward to equalize the pressure. Depending on the location of the rupture, the part can potentially continue to print without further errors.

The risk of a blowout is dependent on the resin type, size of the cup, shape of the model, and thickness of the cup wall. Small cups, or cups with thicker walls, may print successfully. We recommend adding a drainage hole and orienting the model in PreForm to minimize cupping.



Visual symptoms :

- A “hole” in the wall of your part
- A large, ragged “rupture” in your part
- Poor surface finish on cup surfaces

Common causes:

Any of the following factors, or a combination of them, can cause cupping blowout in prints :

- No drainage holes in hollow or convex parts
- Model orientation issues

2.3. Undeveloped features

An «undeveloped feature» is a print defect in which a void of missing material forms within a print and widens as the print builds away from the build platform. These voids can look like a crater and frequently exhibit sharp edges and rough surfaces. In some cases, it can look like the missing material exploded” outwards from the build platform.

This type of defect is a print failure unique to stereolithography (SLA). If a piece of the print sticks to the bottom of the resin tank, that piece partially blocks the laser from reaching the next layer of the print. The piece stuck to the tank expands with each additional cured layer and prevents other features from forming.



Visual symptoms:

- A void within a print that expands outwards from the earliest layer to the most recent layer, usually with sharp edges and rough surfaces like a crater
- A layer of cured resin on the bottom of the resin tank

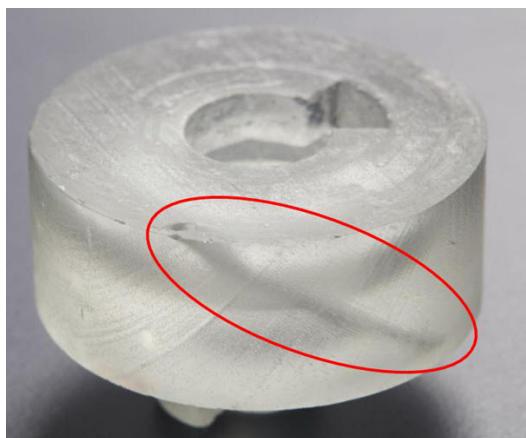
Common causes :

Any of the following factors, or a combination of them, can cause prints to have undeveloped features :

- Debris or damage in resin tanks
- Clouding in resin tanks
- Model orientation, layout, or support issues
- Contaminated optical surfaces

2.4. Pinholes or cuts

A “pinhole” or “cut” describes a print defect in which a small region of the part does not form. This void can take the form of a small hole, a tunnel that runs through the part, or a slit that cuts through the part. If other sections of the part would have been supported directly by the missing material, those sections may also fail to cure to the part.



Visual symptoms :

- A void that passes through the entire print, including the raft (if one is used). The void may take the form of a small hole in one portion of the part or a linear slit that extends across the part.
- A hole or cut may lead to one or more undeveloped features, depending on the geometry and orientation of the part.

Common causes :

Either of the following factors, or a combination of them, can cause a hole or cut in a print :

- Debris or scratches on the bottom surface of the resin tank
- Debris or scratches on the optical window or another optical surface

If, while curing a layer, the laser is blocked before it reaches the resin, that portion of the part will not form properly. If the obstruction is on a stationary surface, a hole may develop directly above that point. If the blockage is on a moving surface, a linear cut may develop instead, following the path of the blockage.

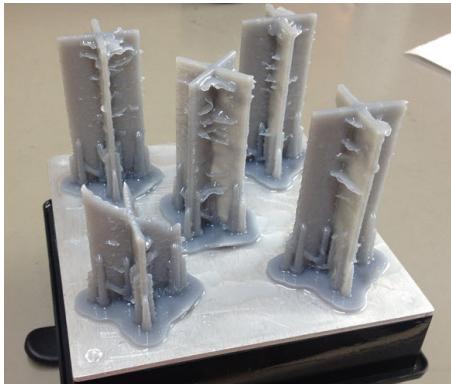
3. Poor surface quality

The following print defects result in a completed print, but the printed parts often show a rough or uneven texture upon close inspection.

3.1. Ragging

“Ragging” describes a print defect in which cured or partially cured resin forms thin, shelf-like structures that hang horizontally from a print. These structures, also called “flaps,” can detach from the part during printing.

Once these flaps are floating in the resin tank, they can block the laser’s path and cause print failures in subsequent layers. Ragging occurs when diffused laser light spreads the cure area beyond the appropriate limits for each layer outline.



Visual symptoms :

- Flaps of cured or partially cured resin on the print that degrade the overall cosmetic aspects of the print
- Thin flakes of cured or partially cured resin from the print floating in the resin tank, which may or may not cause a print failure

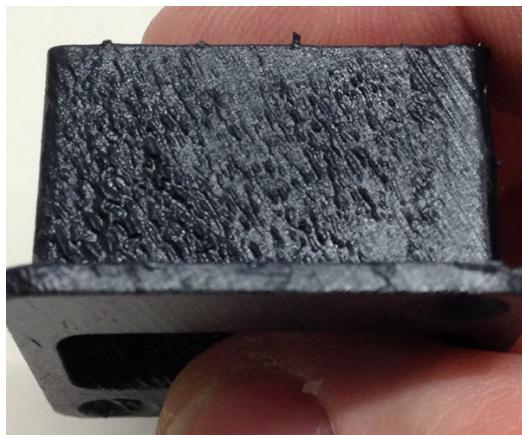
Common causes :

Any of the following factors, or a combination of them, can cause ragging in prints :

- Expired resin
- Debris/clouding/damage in the resin tank
- Contaminated optical surfaces
- Restricted resin flow due to improper model orientation or overly dense supports

3.2. Rashing

“Rashing” describes a print defect in which a rough surface texture forms on one or both sides of an otherwise successful print.



Visual symptoms :

- For prints created on Formlabs printers, rashing produces an uneven or bumpy surface finish on one or both sides of a part.

Common causes :

Any of the following factors, or a combination of them, can cause rashing in prints :

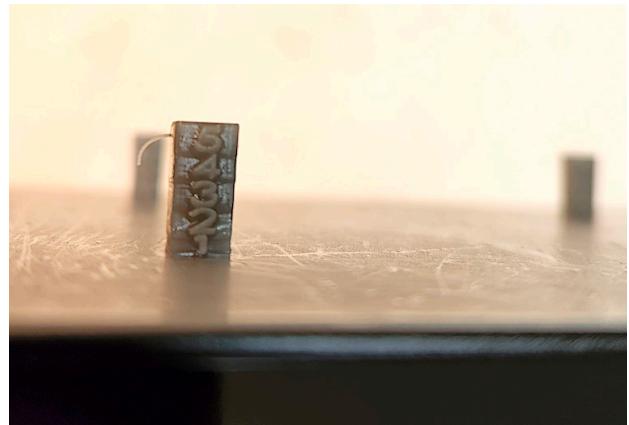
- Expired resin
- Debris/clouding/damage in the resin tank
- Contaminated optical surfaces
- Restricted resin flow due to improper model orientation or overly dense supports

4. Dimensional accuracy

The following print defect results in a completed print, but the height of the printed part is shorter than expected.

4.1. Overcompression

“Overcompression” describes a print defect in which reduced space between the build platform and resin tank causes the initial layers to be very thin. The rest of the part prints, but the initial layers are flattened and may be very difficult to remove from the build platform.



Visual symptoms :

- The raft is compressed and thinner than expected.
- Parts are especially difficult to remove.
- The raft is completely flat and supports are significantly shorter than expected.

Common causes :

Overcompression is caused by insufficient space between the build platform and the elastic layer or the flexible film in the resin tank.

Note that some compression is normal. The first few layers will always be thinner than the rest of the part.