ETH STUDENT PROJECT HOUSE

PRUSA XL [DUAL-HEAD] 3D PRINTER

FDM style printer
Build volume: 36 x 36 x 36 cm
Layer height: 0.05 - 0.3 mm
Materials: PLA, PETG

USE CASES FOR XL PRINTER

❗ Use the XL 3D-printer **ONLY if one of the following cases is true**:

a. You want to print a large model **which does not fit in the regular FDM printer AND/OR ...**

b. You want to print a model **with multiple colors AND/OR ...**

c. You want to print a model **with multiple materials (only PLA and PETG).**

In any other case, we might need to restrict your access to the XL printer.

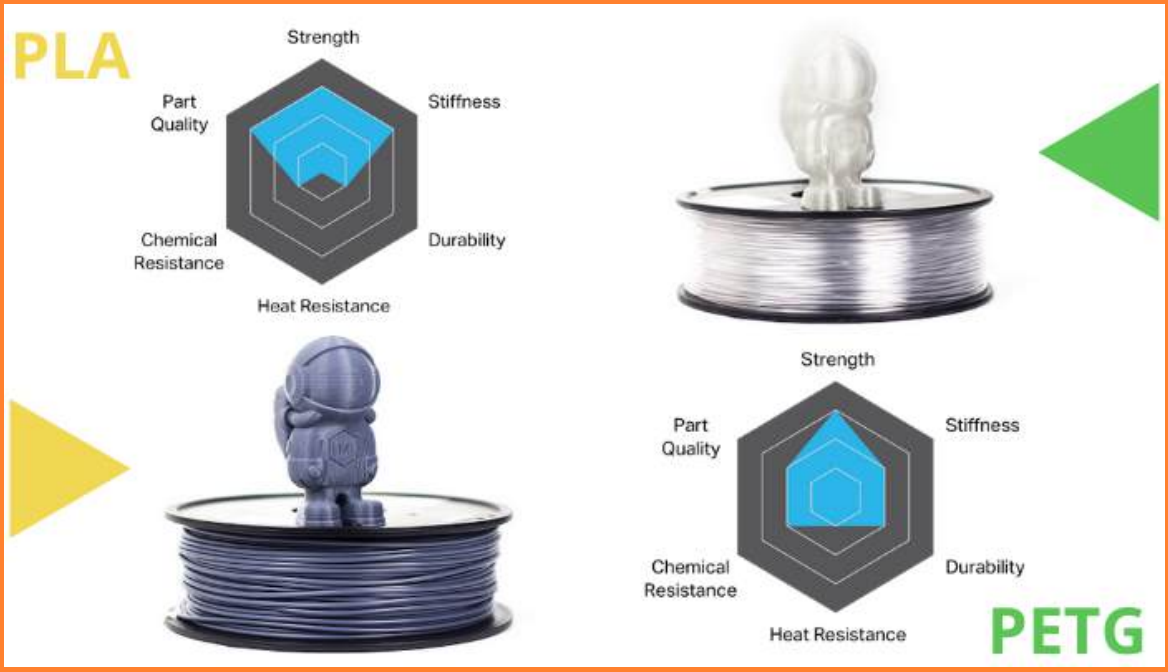
MATERIAL PROPERTIES

PLA

Biopolymer
Hotend: 180 - 210 °C
Heatbed: 50 - 70 °C
Small amount of warping
Easy to print
Sensitive to temperature

PETG

Modification of PET
Hotend: 220 - 240 °C
Heatbed: 80 - 100 °C
A bit of warping
Fairly easy to print
Not sensitive to temperature

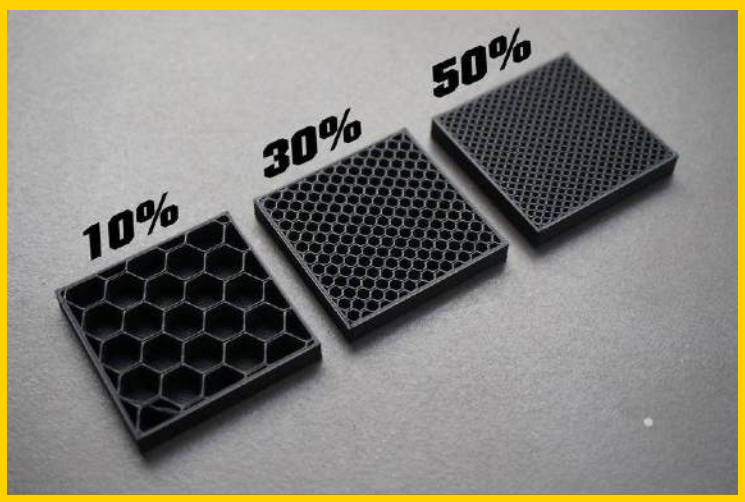


The main differences between the two are their properties, applications, and material costs. **PETG is stronger and more resilient than PLA**. PLA, on the other hand, is widely used as FDM/FFF filaments because of its better melt and cooling properties. In terms of cost, PETG is more expensive than PLA.

TERMINOLOGY

Infill

- Parts are not solid
- Most of the time 15% is enough



Supports

- Machine can’t print mid-air
- Supports the print above
- Threshold value 45°

Brim

- Needed if only a small area of contact to the bed is available to increase contact area
- Prevent lifting of the part from the print bed

PRUSA XL PRINTER

There are 2 extruder heads on Prusa XL

The Prusa XL Dual-Head 3D Printer is equipped with two extruder heads, allowing for flexible printing options.

Single Extruder

This mode simplifies the printing process by utilizing only one of the two extruder heads, ensuring straightforward and reliable prints with either PLA or PETG.

OR

Double Extruders

This mode allows the use of both extruder heads for the same print, enabling you to print with two different materials or colors in a single print job. This is ideal for creating complex models with multiple materials or color variations, enhancing the versatility and capability of your 3D prints.

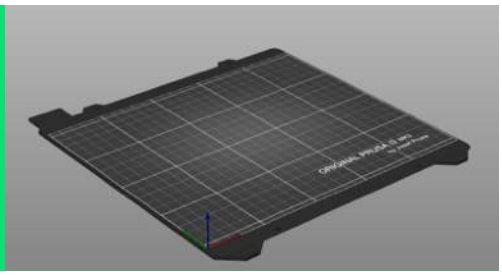
Input Shaper

Input Shaper is a new feature available on the XL printer, designed to reduce ghosting by canceling resonance vibrations. It works by analyzing the printer's movements and applying a filter to the input signals. Thanks to faster travel speed and acceleration, it minimizes stringing and enables faster printing. The pressure in the nozzle from the faster printing is compensated by another firmware feature, the Pressure Advance.

PRUSA SLICER BASICS

Base view

- Open PrusaSlicer
- Make sure the **“Advanced/Normal”**mode is activated
- Drag and drop your .stl file on the print bed



TRANSFORMATIONS

Move

Use this to move your part around the build plate

Scale

This tool can be used to make a part bigger/smaller

Rotate

With this tool you can rotate your part around x,y and z axis

Place on face

Select a face and the software will place it on the build plate

Cut

This tool lets you cut your part into two

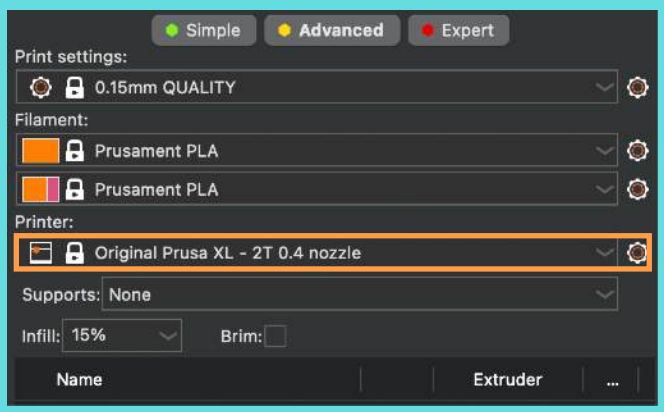
Multimaterial Painting

This tool allows you to paint a model with multiple colors and materials (only on “Advanced/Normal” and “Expert” mode)

A.PRUSA SLICER

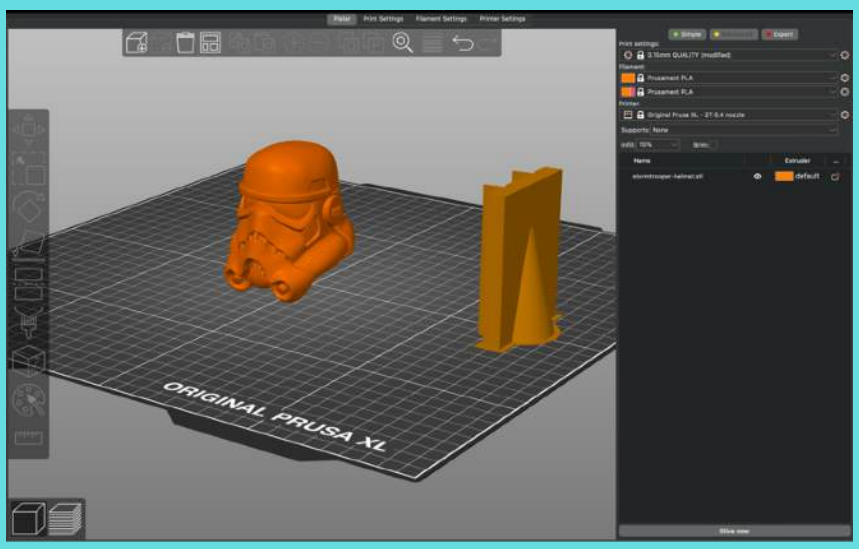
1. Select the correct printer

- Make sure to select the correct printer type: **[Original Prusa XL - 2T 0.4 nozzle] or [Original Prusa XL - 2T Input Shaper 0.4 nozzle].**



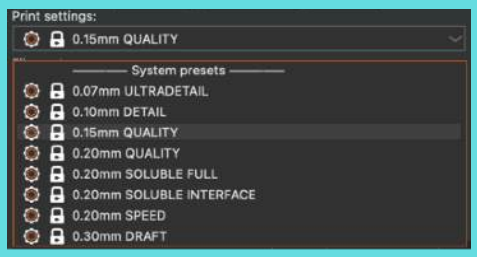
2. Drag your print into software

- Drag your stl / f3d / obj / other file format onto the printbed as shown on the screen.
- Note: The structure you see next to your model is an automatically-generated structure called the **priming tower**. It is a block that will be 3D-printed next to your object to help stabilize the pressure inside the nozzle once an extruder is unparked from the dock. There will be one next to every 3D model you are trying to print, so do not worry about it, it just ensures better printing quality!



A.PRUSA SLICER

3. Select layer height



- Select the layer height to decide the print quality.
- Smaller layer heights take longer to print.
- Smaller layer heights print with better quality.
- Check example frogs in the 3D-printer area.
- When using **both PLA and PETG, use the [0.20mm SOLUBLE FULL] option.**

4. Multimaterial painting

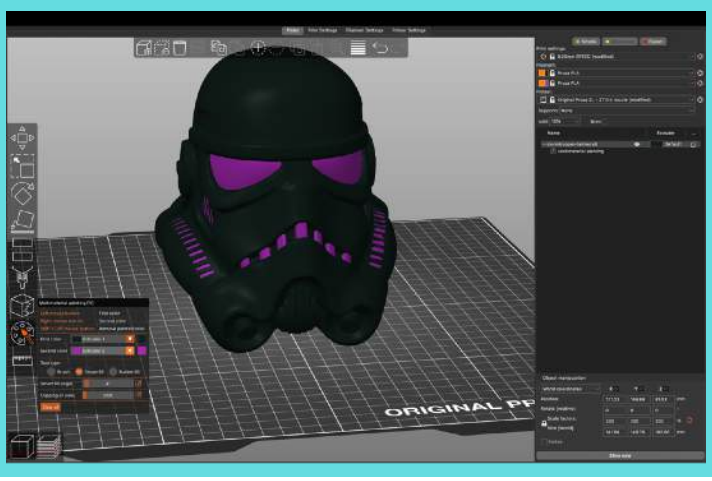
- Select your 3D model.
- Switch to “Advanced/Normal” or “Expert” mode on the top right menu.
- Choose the “Multimaterial painting” icon on the left bar menu.

Extruder 1

When you select "Extruder 1" you are choosing the first color and setting the material color for this extruder. First, choose your desired material color. Then, select your “Tool type” and we recommend using “Smart Fill.” Adjust the “Smart fill angle” to increase or decrease the sensitivity for grouping polygons (smaller angle means you will be coloring smaller areas). To apply color, hold down the “**left** mouse button” and paint the desired area. To remove the painted color in an area, hold down the “Shift” key while left-clicking.

Extruder 2

Repeat the same initial steps for “Extruder 2” to select the color, the tool type and the angle. To apply color, hold down the "**right** mouse button" and paint the desired area. To remove the painted color in an area, hold down the “Shift” key while left-clicking.

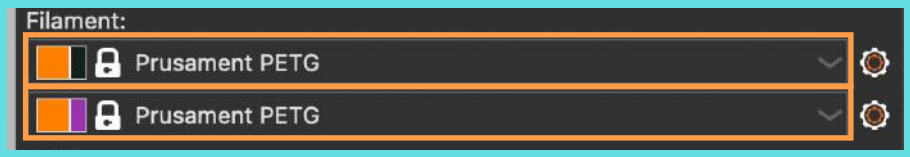


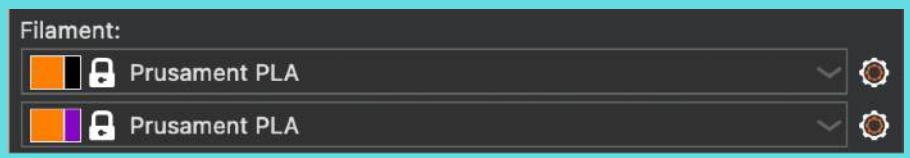
A.PRUSA SLICER

5. Select materials

- Now you can select your print material(s): it is written on the side of the spools
- The colors you select on the software are only for viewing purposes! It does not make a difference for the actual print.
- **Please use only Prusament PLA or PETG for each extruder.**

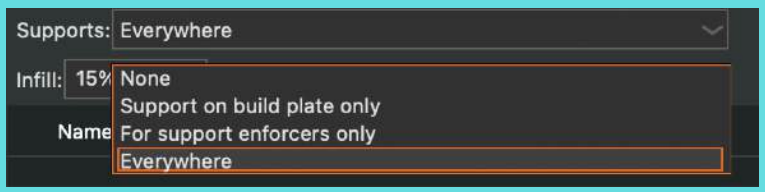
Extruder 1 ➡
Extruder 2 ➡





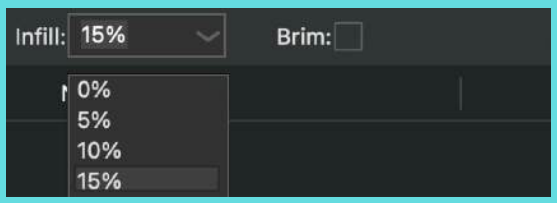
6. Adjust the supports

- Set your desired support setting.
- For beginners only use “None/Everywhere” (select “Everywhere” to be sure).



6. Select the infill

- Set your infill percentage, leave it to 15% if not sure.
- More infill means more weight, longer print times and higher costs.

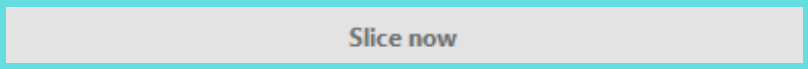


7. Select brim / no brim

A.PRUSA SLICER

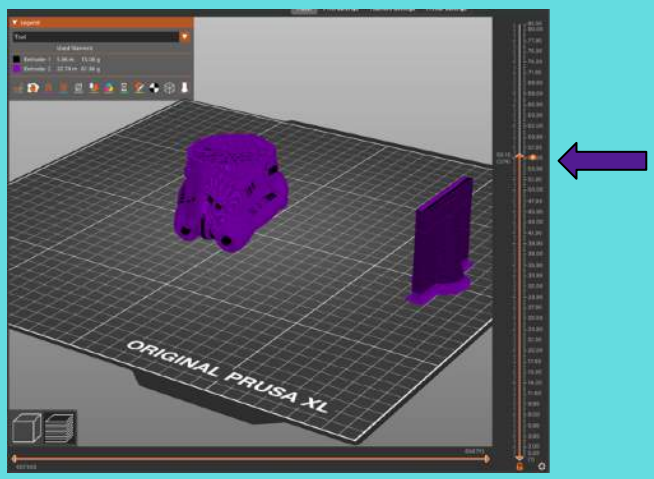
8. Slice your print

- Use the ‘Slice now’ button to prepare your part for printing.



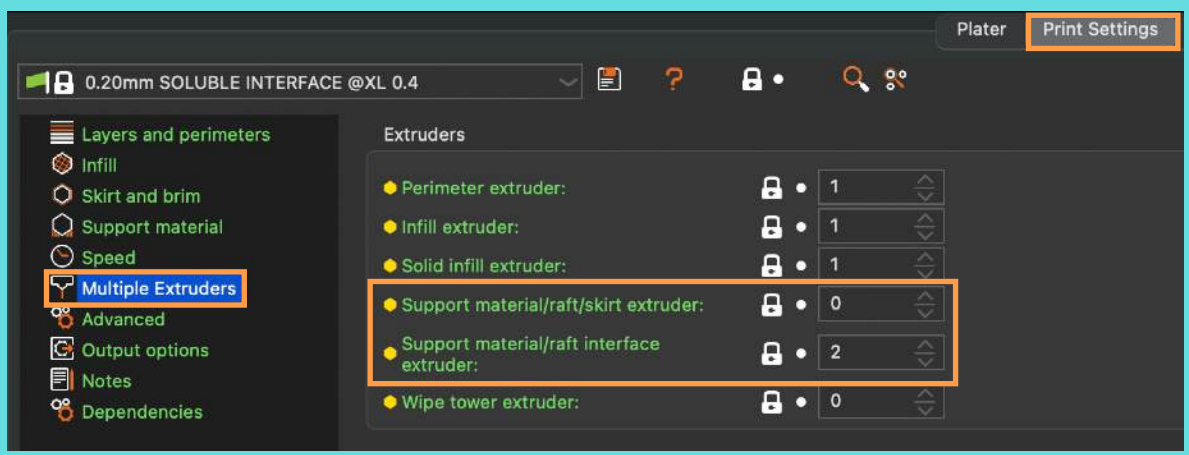
9. Preview check

- Use the slider on the right to go through the different layers.
- Check that at no point any layers appear out of nowhere.



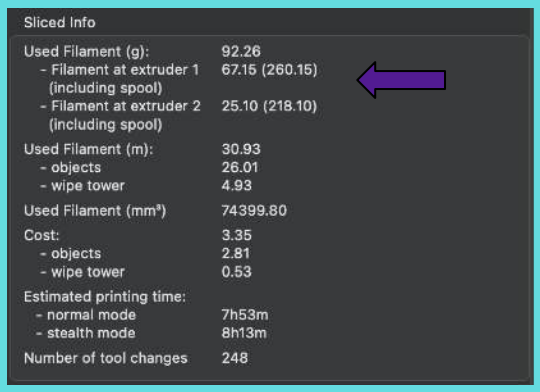
10. Change material of supports and support interface

- Navigate to “Print Settings” > “Multiple Extruders” > “Support material/raft/skirt extruder” or “Support material/raft interface extruder”.
- The number you indicate is the number of the extruder that will be used to print those structures.



11. Check if you have enough filament

- On the lower right corner you can find the expected weight on your print
- Find the empty spool and use it to tare the scale. Weigh your spool and ensure you have the **predicted weight + 10%** for every spool you want to use.
- Note: The costs are not correct.



The XL printer is equipped with two filament sensors, hence it will not break in case of filament runout but you will be left with a half-finished print that you will have to bill.

12. Export your file

- Select ‘Export G-code’ in the bottom right corner and save it onto the dedicated USB stick of the printer.

B. AT THE MACHINE

13. Turn on your printer

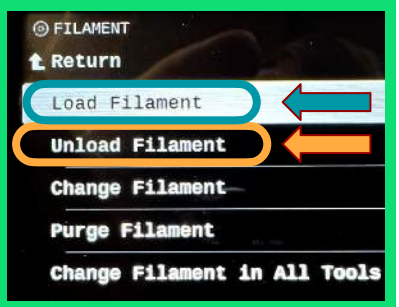
- Turn on the machine on one of the terminals (select machine → use machine).

14. Check that you have enough filament

- Select the spools with the filament of your choice and make sure that there is enough for your print (see previous page).

15. Load each type of filament in the correct extruder

- Press the knob on the printer and navigate to [FILAMENT].
- Select the “Unload Filament” option.



- When the nozzle is up to temperature the printer will unload the filament. Immediately pull the filament out and roll the free filament back on the spool (do not just pull it out because it might break).

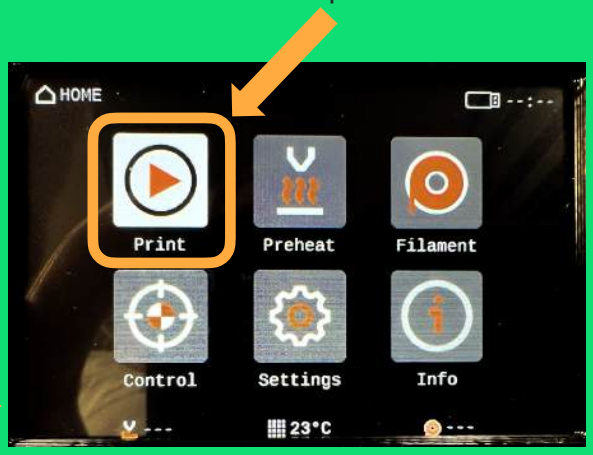
- Cut the new filament at an angle and insert it into the correct tool/extruder as you specified it in page 8.
- Select the “Load Filament” option on the printer menu and press the knob.
- The filament should immediately start to be pulled in.

- When asked if the filament is of the correct color: if you see that the filament is not of the correct color select [PURGE MORE], more filament will be extruded. When you have the correct color extruding select [YES] and you are good to go.

B. AT THE MACHINE

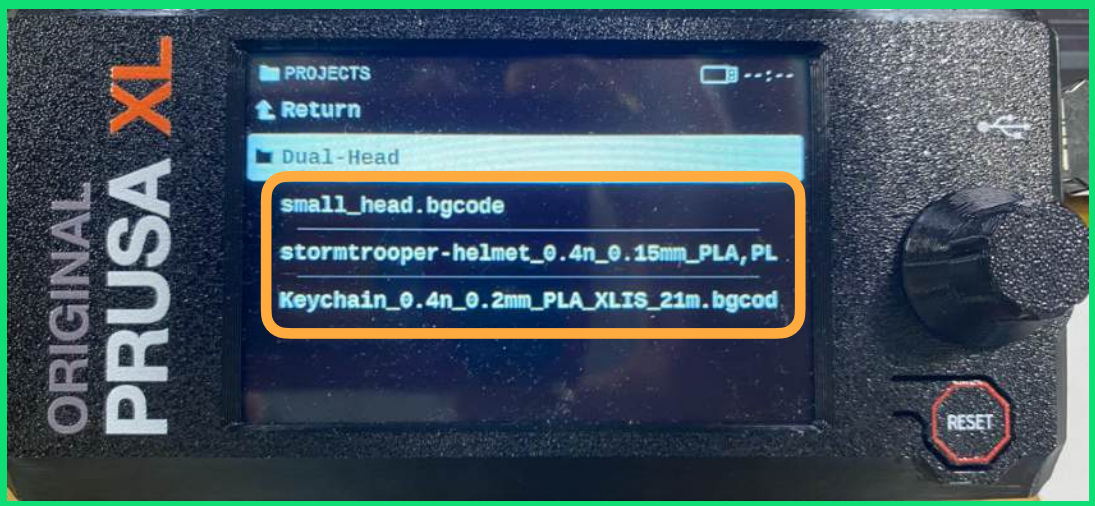
16. Select your file

- Insert the USB stick as shown below.
- Press the knob once on the printer and navigate to “Print”.
- Files will be shown with the most recent files located at the top of the list.



17. Select your print

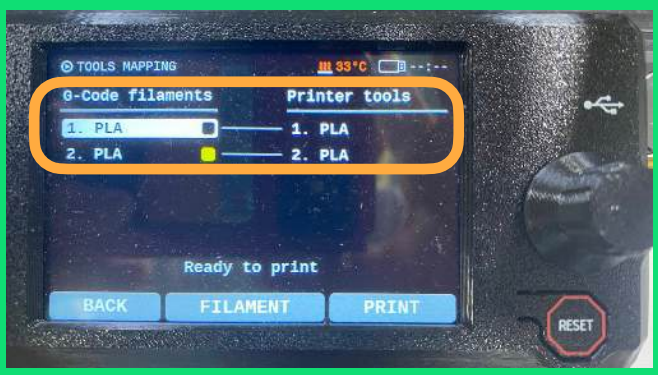
- The folder [Dual-Head] includes some example prints that you can try if you would like.
- Your file should be the one immediately after the folder.



B. AT THE MACHINE

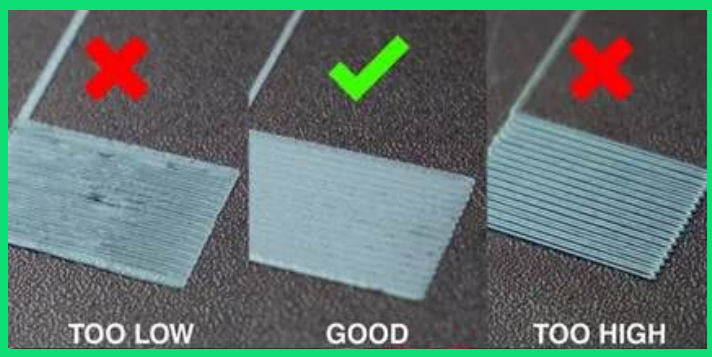
18. Verify the material type for each extruder

- In the following screen verify that each extruder number is assigned to the correct material from the G-code file. Compare the materials you selected in the software part on page 8 with the materials that you have inserted into each extruder of the printer. Make sure that also the colors are your desired ones before you proceed.



19. Stay to check the first layer

- Check the height of the first layer.
- If it is too high/low please speak to a Makerspace Manager.
- **If we see your print failed in the first layer and you did not see it, we will charge the final weight of your print to your account. If it failed later because of our printers, we will not charge you for the material that was used.**



C. TAKING OFF YOUR PRINT

Taking your print off the bed

- The print sheet should be LIFTED(!) (not dragged) from the XL 3D-printer.
- DO NOT use metal spatulas as it can scratch the surface of the print sheet.
- DO NOT wash the print sheet under running water!
- Bend the print sheet to easily remove the print.

Weighing your print

- Place the print AND all supports on the scale.

Logging off of the printer

- Use your legi to log into a terminal, and select the XL printer.
- Select “Free Printer”.
- **If you have used PETG as one of the materials, then choose PETG as the material. Otherwise, select PLA (if both extruder materials were PLA).**
- Costs
  - **PLA: 0.05 CHF/g**
  - **PETG: 0.06 CHF/g**
  - Note: this will be taken off of the money in your account. If you go under 0, then all machines will be blocked for you until you re-fill your account to at least 0.
- If one of our staff gets to your print first, we will do this process for you and you will find the print in one of the collection boxes. You will receive the invoice in your email.

WARNINGS

1. Stick to the Usage Guidelines

Use the XL 3D-printer **ONLY if one of the following cases is true**:

a. You want to print a large model **which does not fit in the regular FDM printer**
   AND/OR ...
b. You want to print a model **with multiple colors**
   AND/OR ...
c. You want to print a model **with multiple materials (only PLA and PETG).**

In any other case, we might need to restrict your access to the XL printer.

2. Unbilled Print Fee

If you remove your print without weighing it and paying for the used material, we will **assume you used 500g** of the material on EACH spool.

ETH STUDENT PROJECT HOUSE