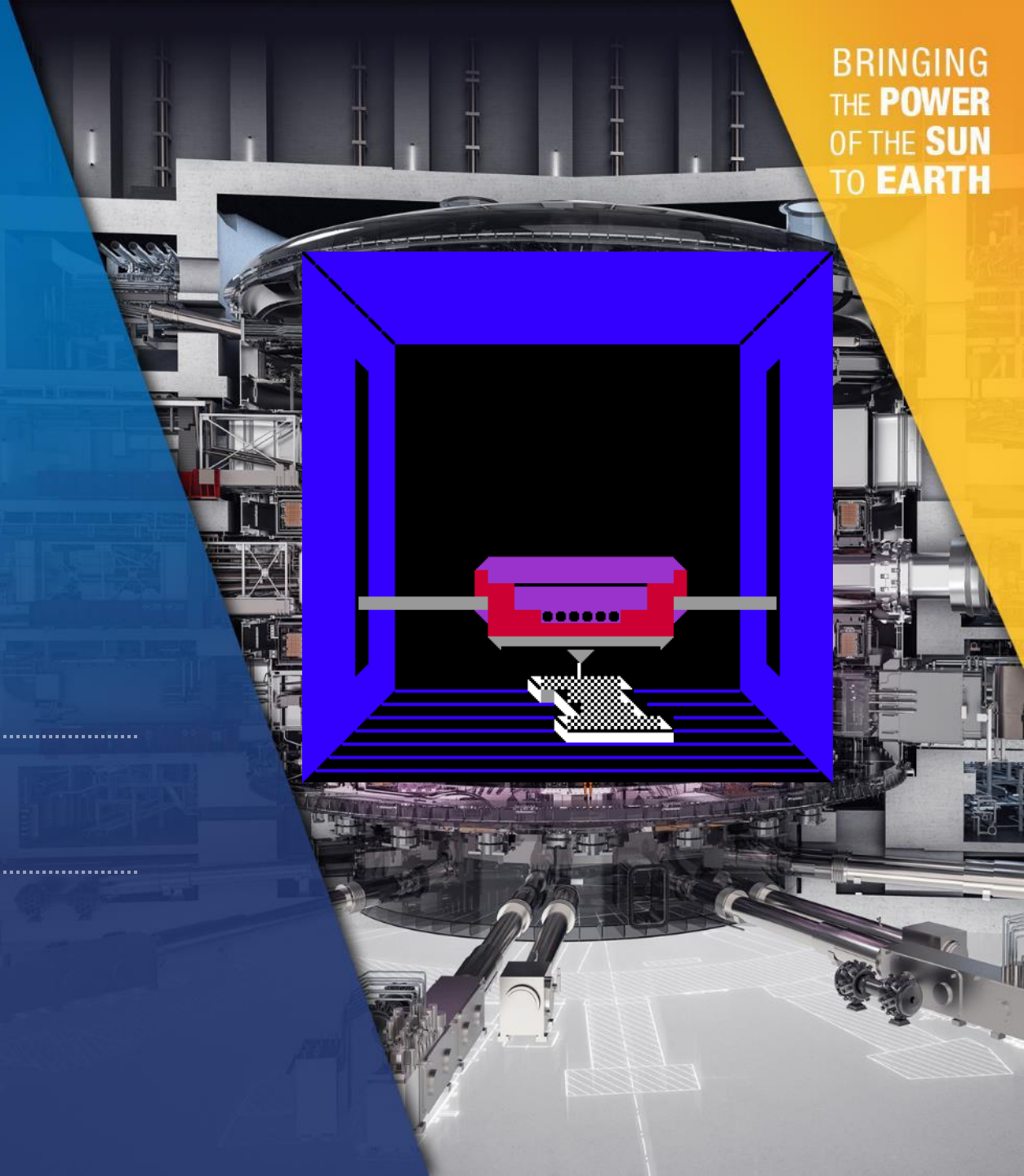
FUSION FOR ENERGY

BRINGING THE POWER OF THE SUN TO EARTH

3D PRINTER SLICER

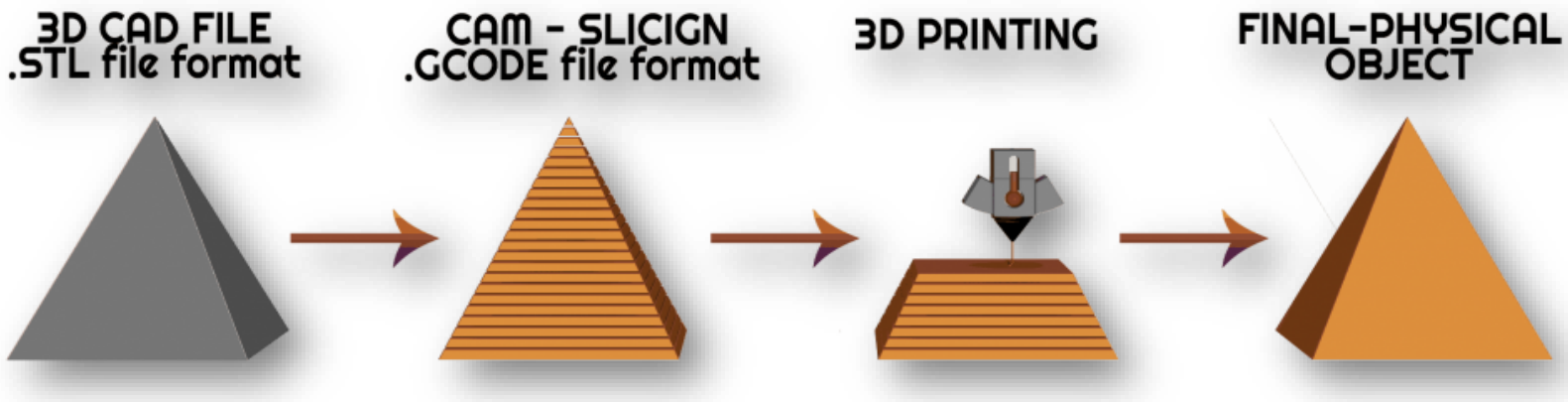
Jose PACHECO

16 January 2024



SLICER

You can't just take a 3D model and load it directly into your 3D printer. You need to convert it to a special instruction file called a **G-code**. This conversion process is called slicing, which is done in software called the **slicer**.

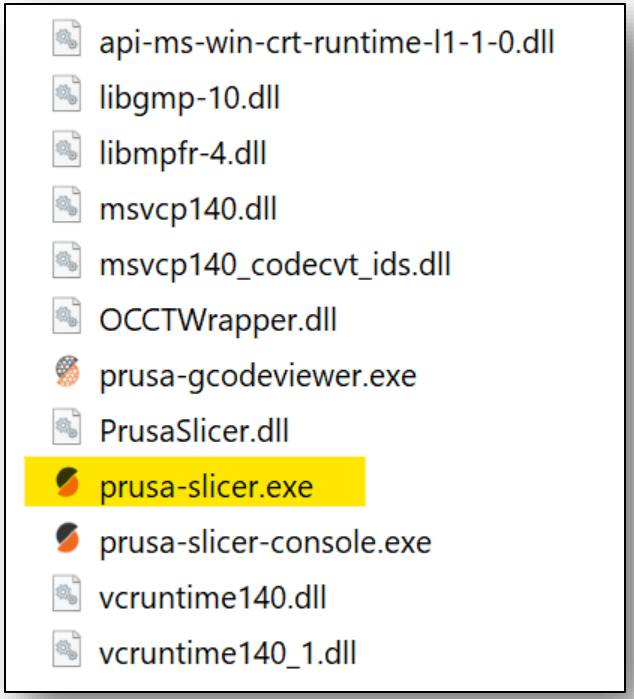


SLICER Set-up

1st Step / Download Prusa Slicer Standalone Software:
PrusaSlicer-2.8.1+win64-202409181359.zip

2nd Step / Unzip the files

3rd Step/ execute prusa-slicer.exe



https://github.com/prusa3d/PrusaSlicer/releases/

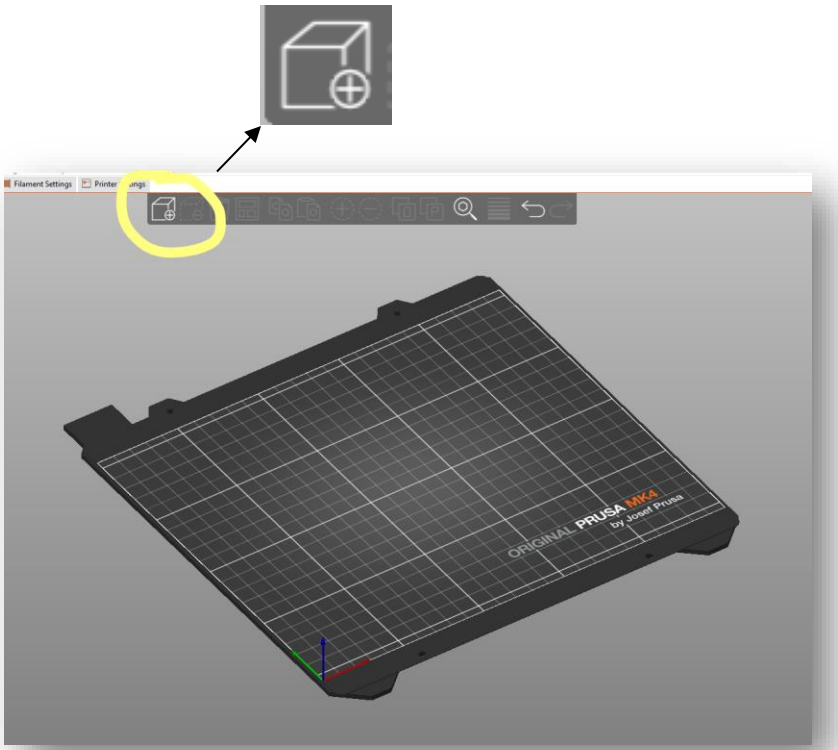
SLICER Set-up

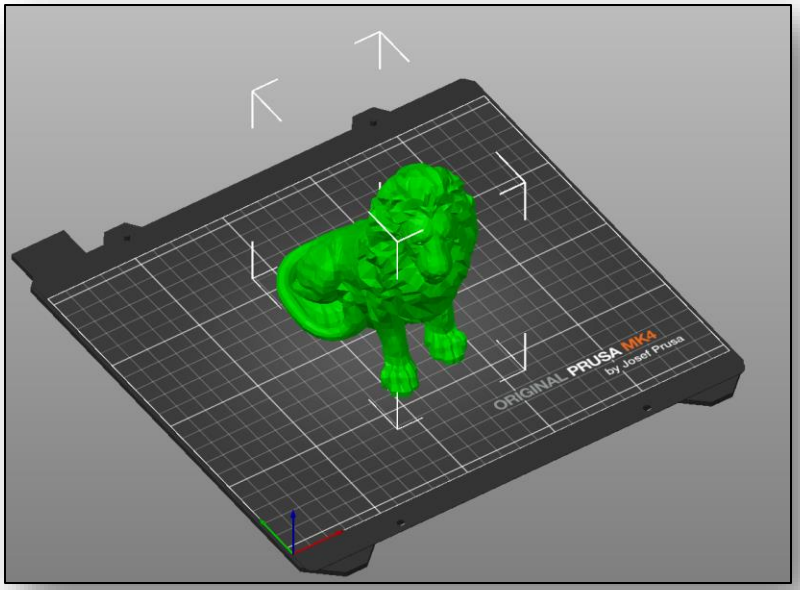
When you first launch the PrusaSlicer, there is a Configuration Wizard.

Select the **Original Prusa MK4** printer and the **0.4mm nozzle**.

Importing 3D data into PrusaSlicer

For the import, you have two main options - an **STL or an STEP file.**

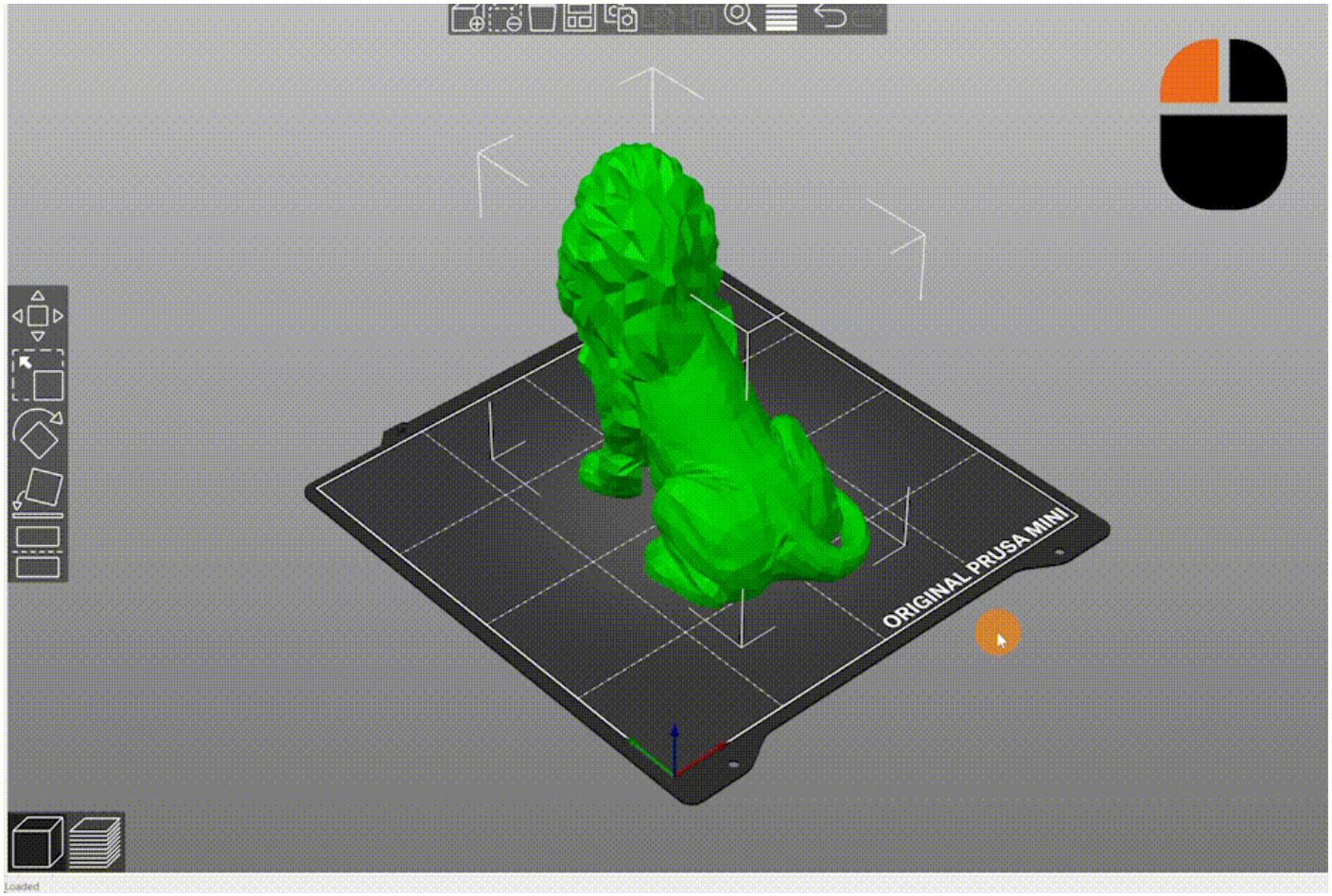




We have chosen this beautiful lion.
Download the STL file and import into the slicer

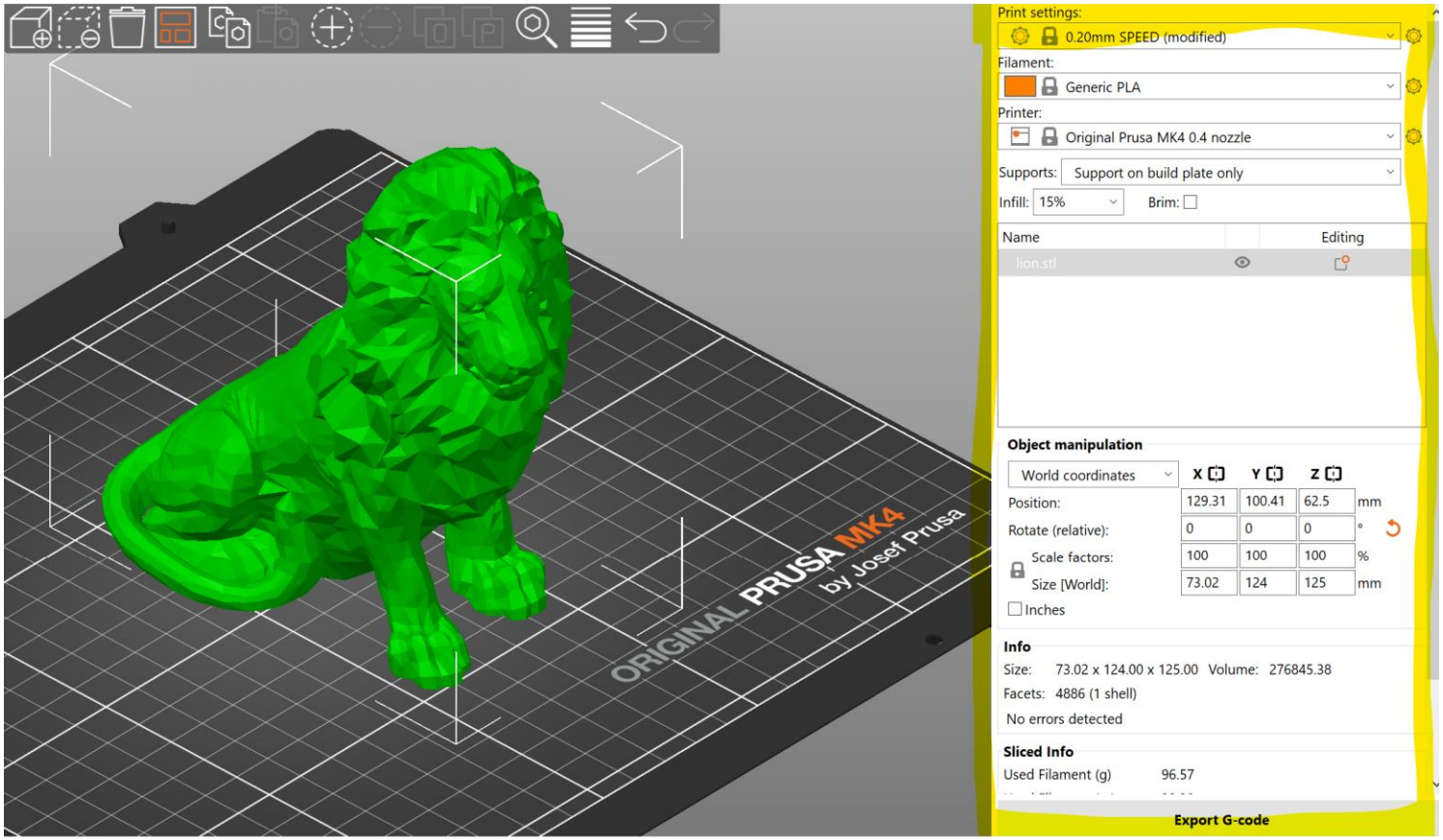
Manipulating the view

Hold the left mouse button to **rotate** the print bed, **zoom** by scrolling mouse wheel, or hold the right button or mouse wheel to **pan** the view.

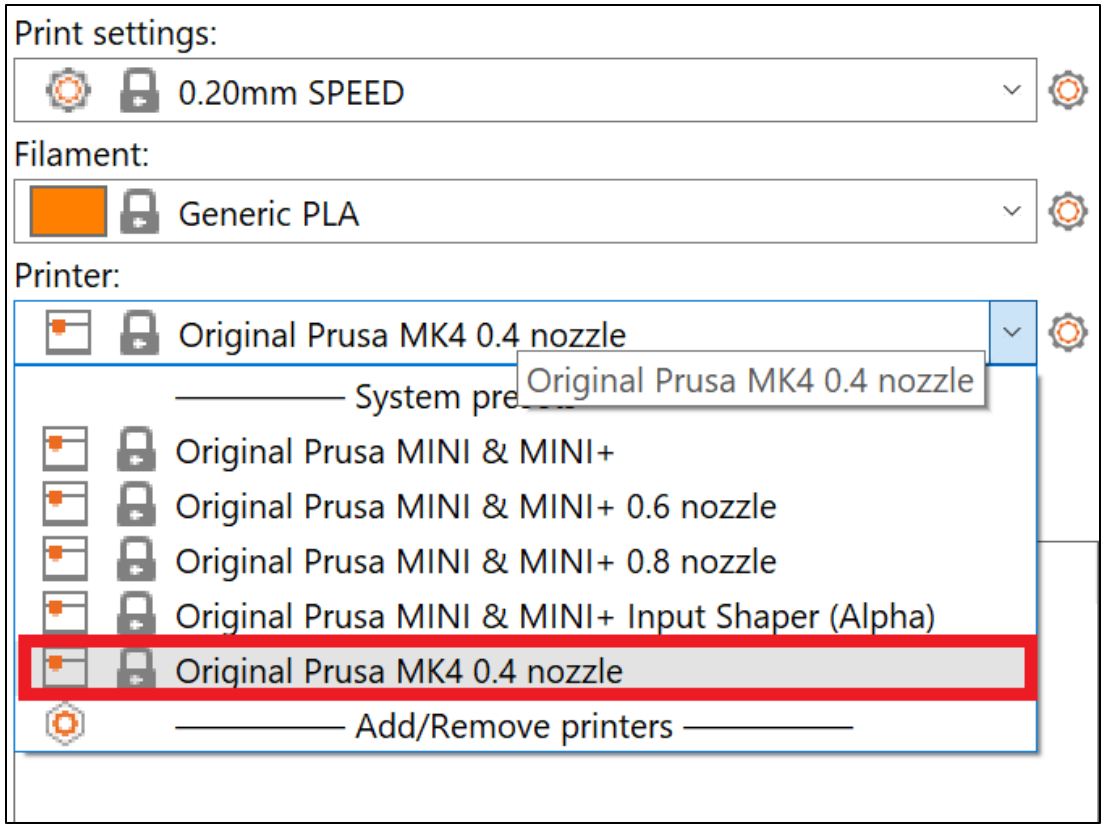


Interface: right side panel

All the basic**, must-have settings** are placed in this part of the screen: selecting the layer height, print material, printer type, enabling supports, and choosing the infill density.



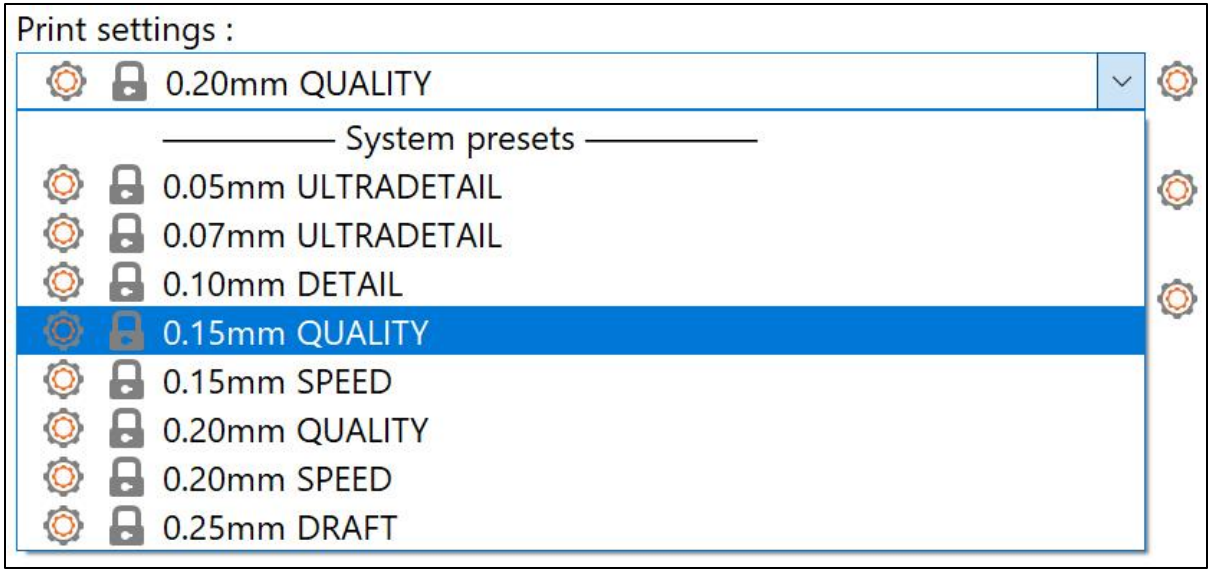
Interface: Printer



Choose the correct printer:

Original Prusa MK4 0.4 nozzle in your case

Interface: Print Settings



In this dropdown menu, you can choose from **various print layer heights.**

Interface: Layer Height

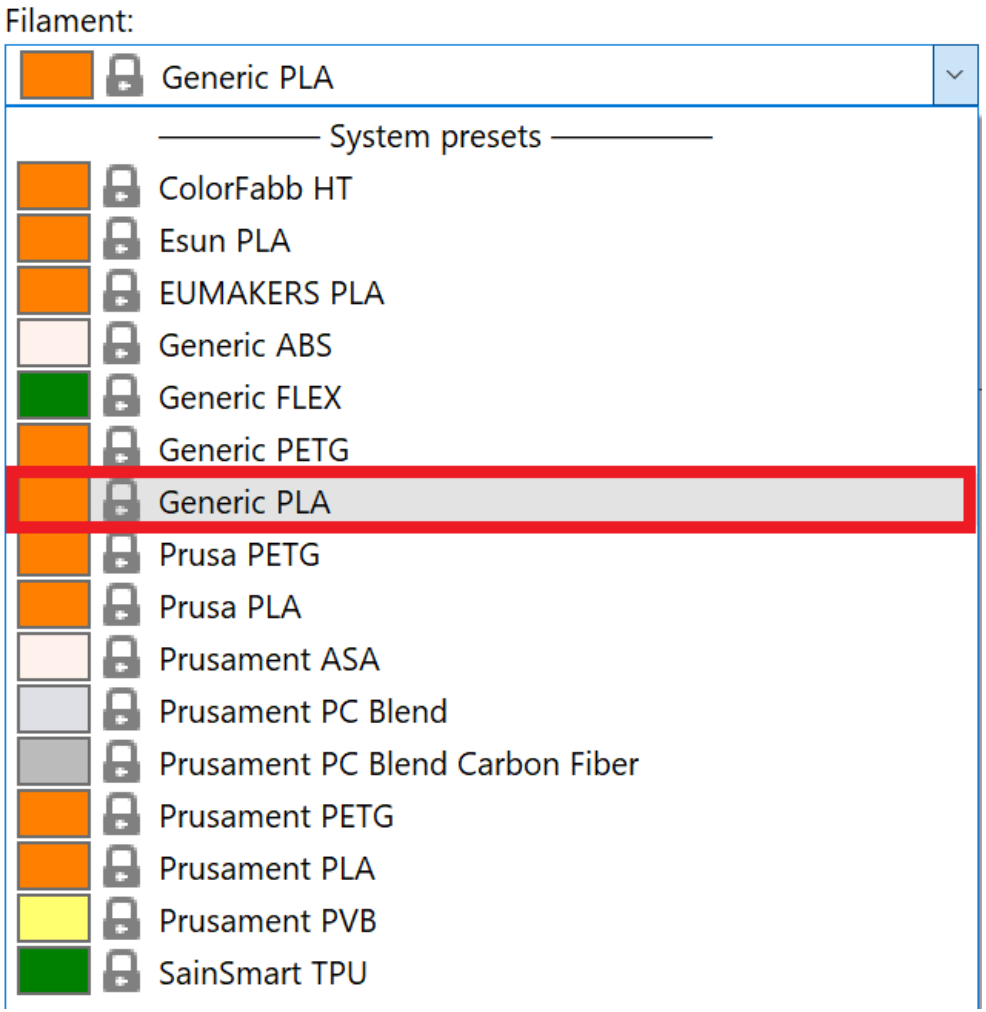
0.10mm
1h 11min

0.15mm
49min

0.20mm
37min

The lower the layer height, the better the details will be **- but the print process will take longer**. The layer height has a big impact on the quality of the surface details. However, choosing small layer heights considerably increases the printing time.

Interface: Filament



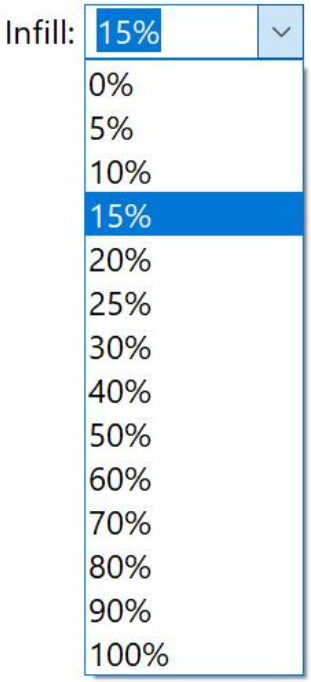
Choose your material.

Different materials require different settings (temperature, print speed, etc.).

All those parameters are **automatically adjusted by the slicer.**

Choose Generic PLA

Interface: Infill

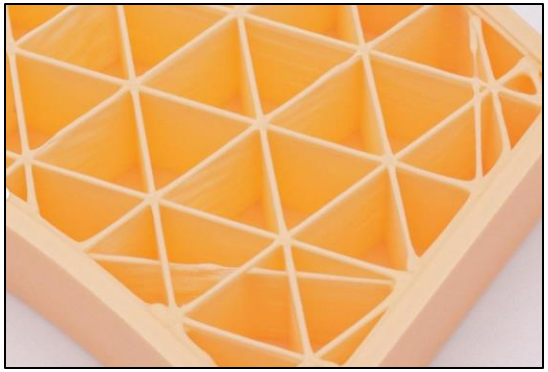


Change the **infill** density.

For now, you can leave the preset value (15 or 20%) as it is

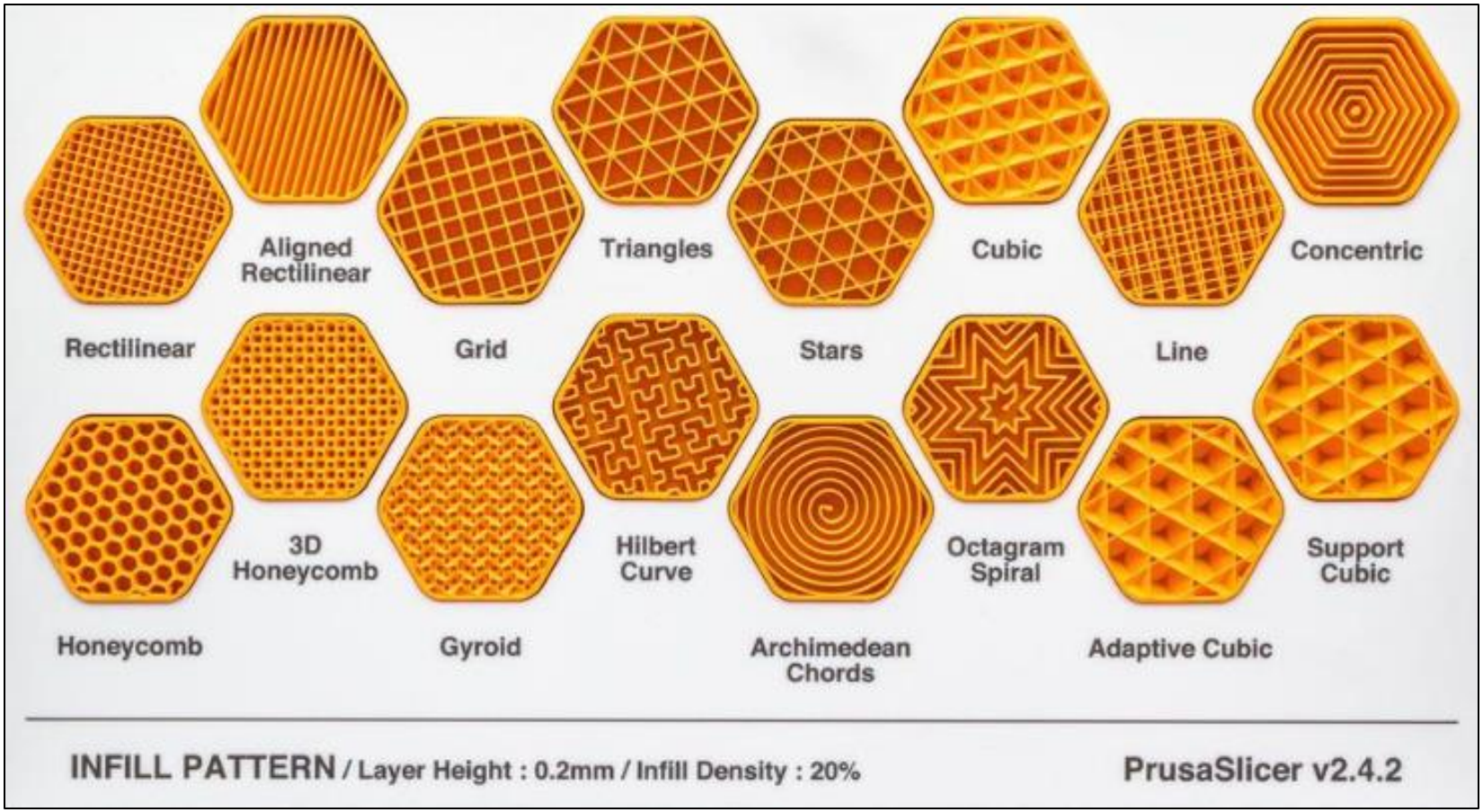
3D models are rarely printed fully solid on the inside, with 100% density. The print would take ages, consume loads of material and the resulting object would be heavy as a brick.

Instead, most prints are hollow, with supporting mesh filling the cavity - the infill.



INFILLS

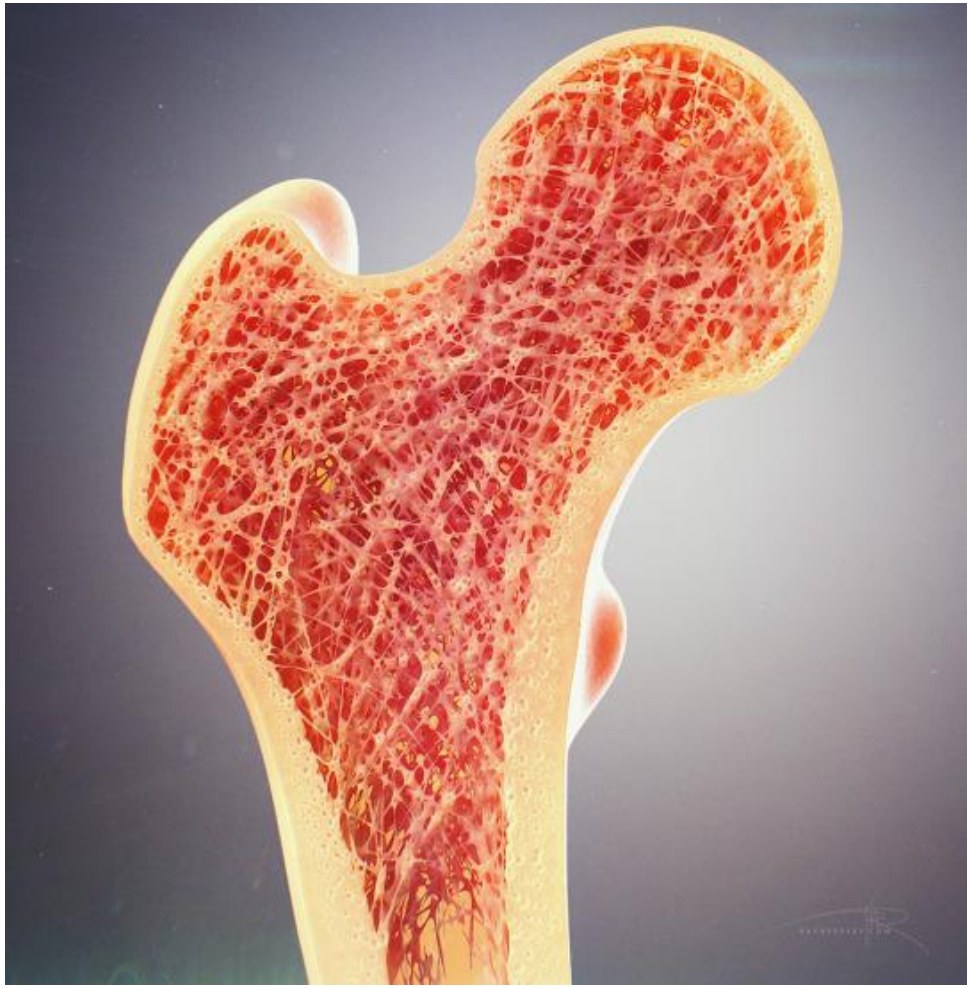
The infill can have **various patterns** with either aesthetic or mechanical special benefits (e.g. honeycomb, stars, or a curved mesh). **The main purpose of infill is to add an internal support structure for the top layers of the print.**



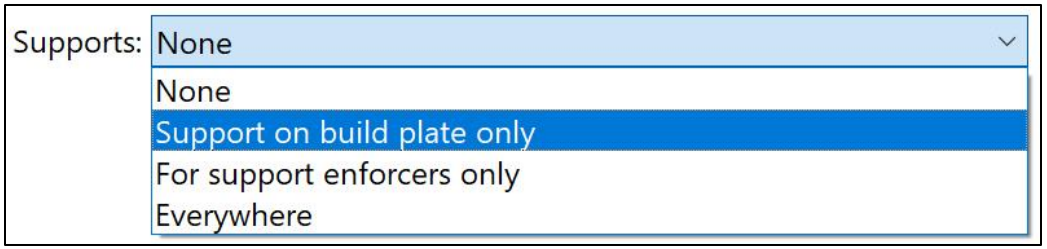
INFILLS

Objects with **higher infill density are also somewhat tougher.**

15-20% is usually a good tradeoff between toughness and reliability vs. print time and material consumption.



Interface: Supports

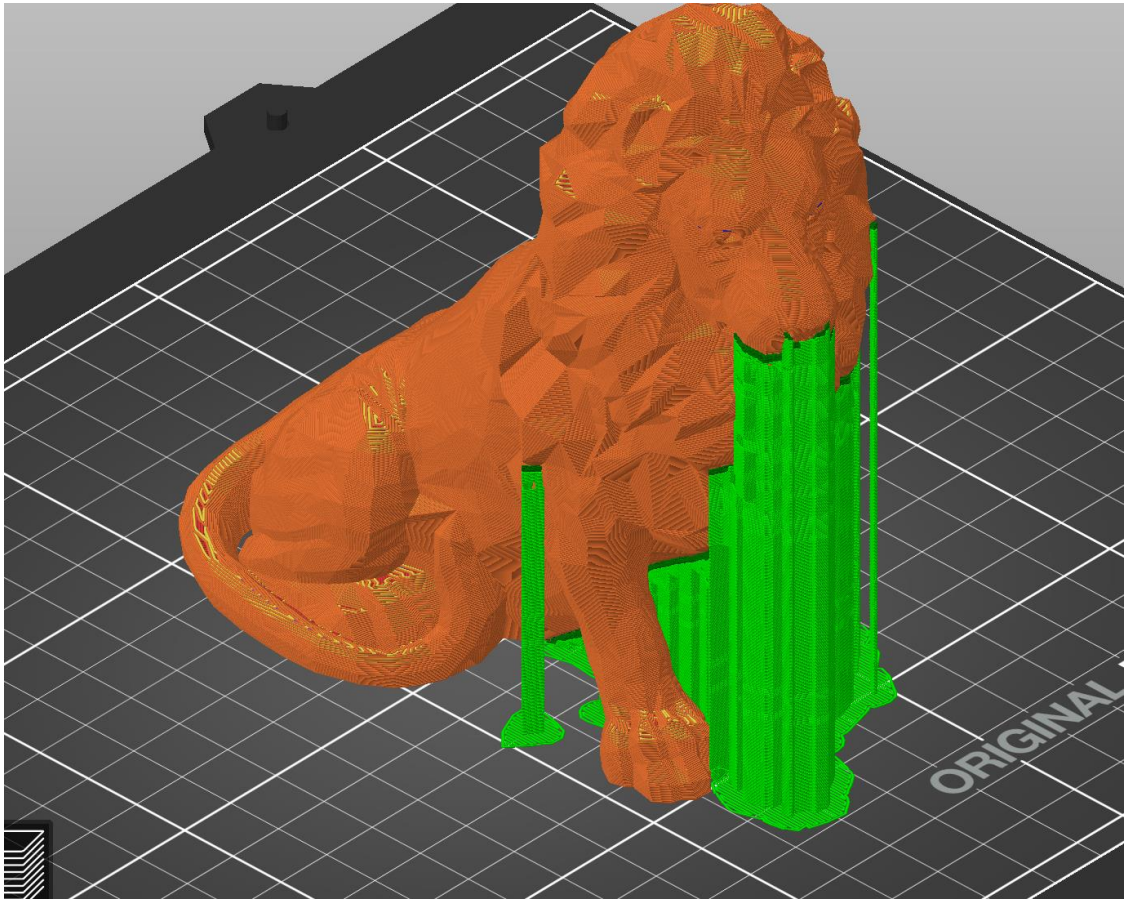


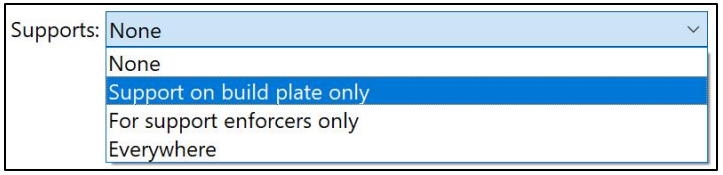
The parts of the model which are hanging or protruding “in the air” are called **overhangs**. Whether these overhangs are printable depends on their angle. 45° is OK (imagine arms of the letter “Y”), 90° (arms of the letter “T”) is not.

Choose whether slicer should generate **supports.**

When the print is finished, you can just snap the supports off and throw them away.

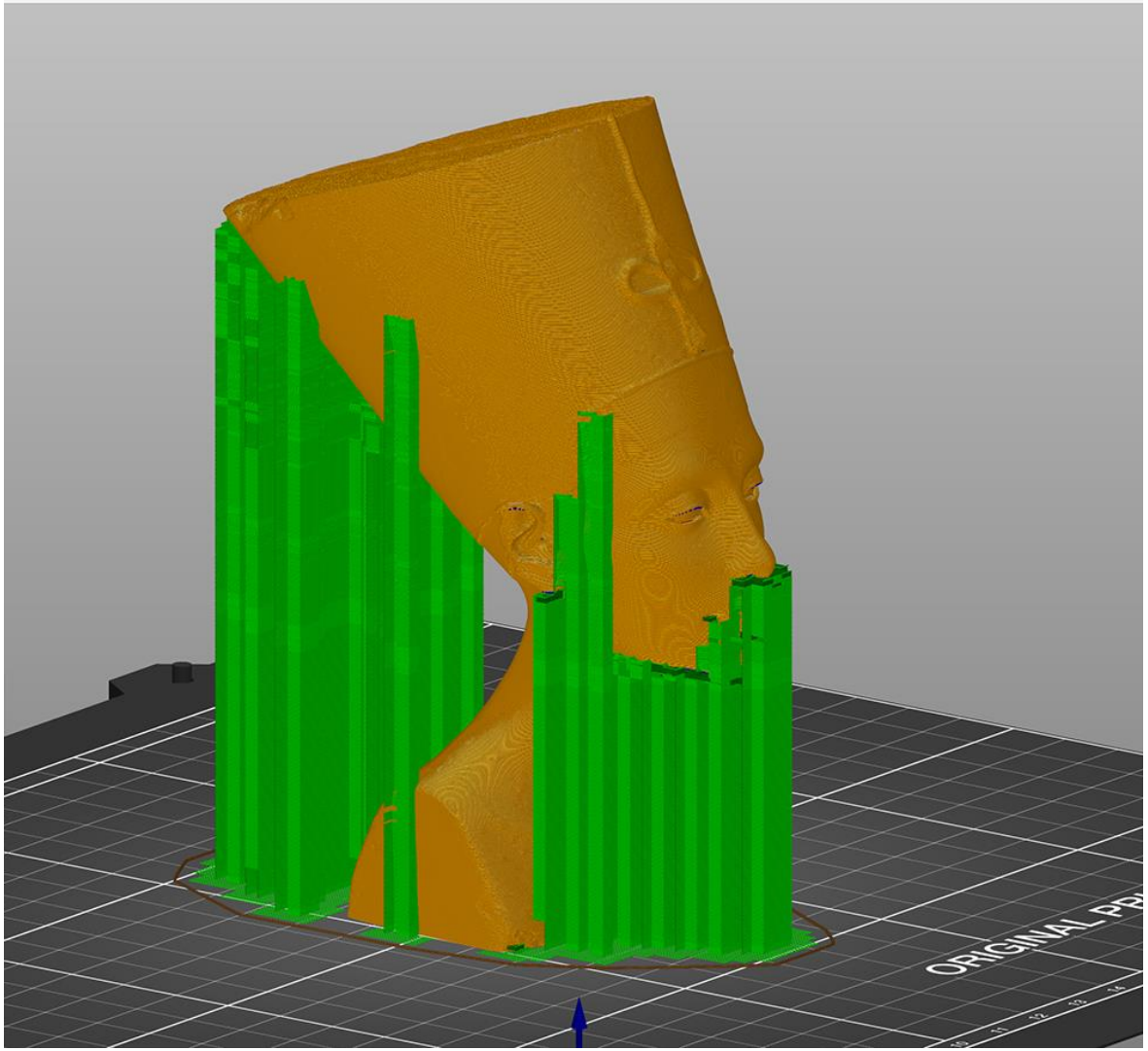
Interface: Supports





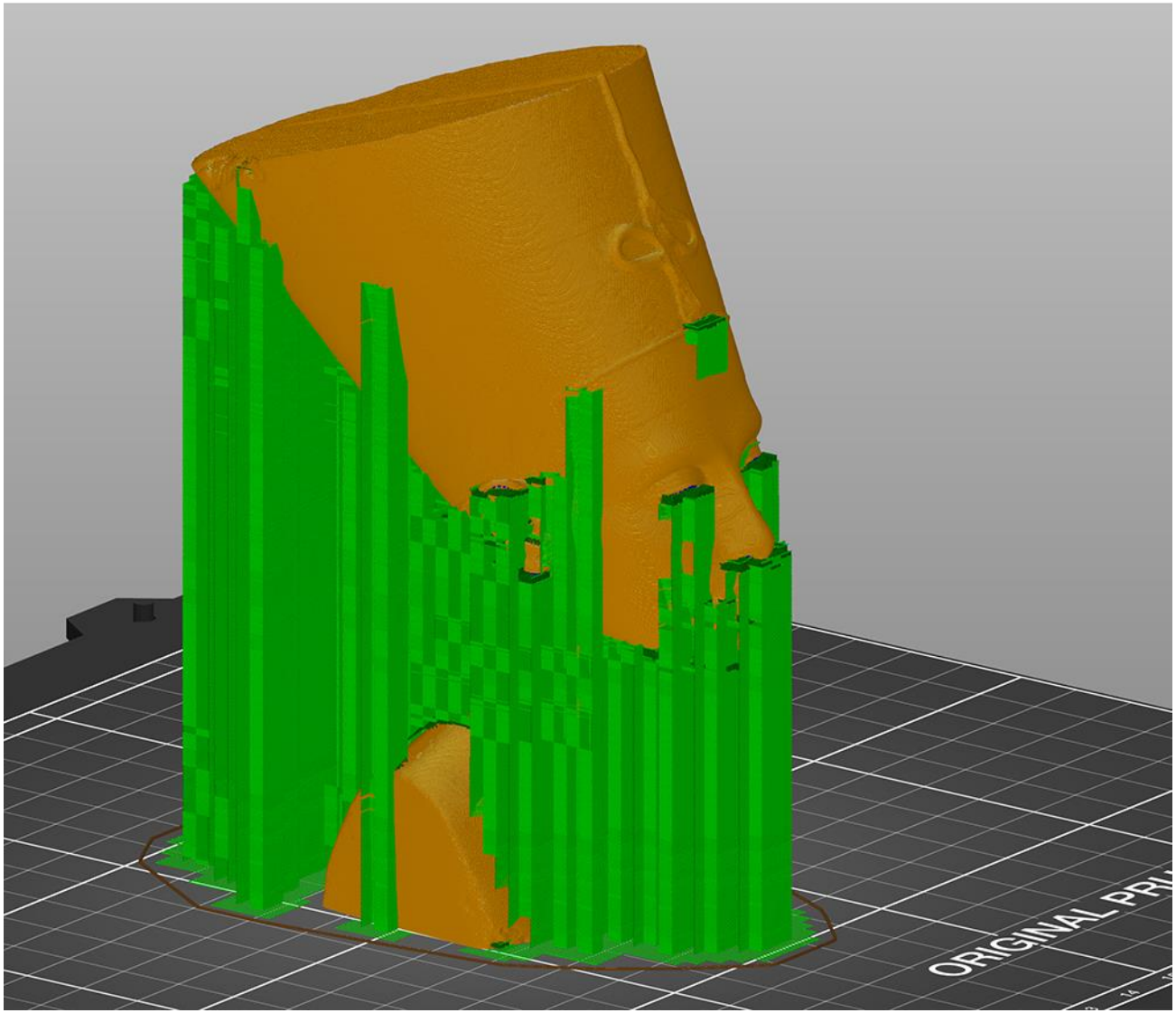
If enabled the slicer will **automatically calculate the needed supports** considering the overhangs on the 3D model.

Supports on build plate only



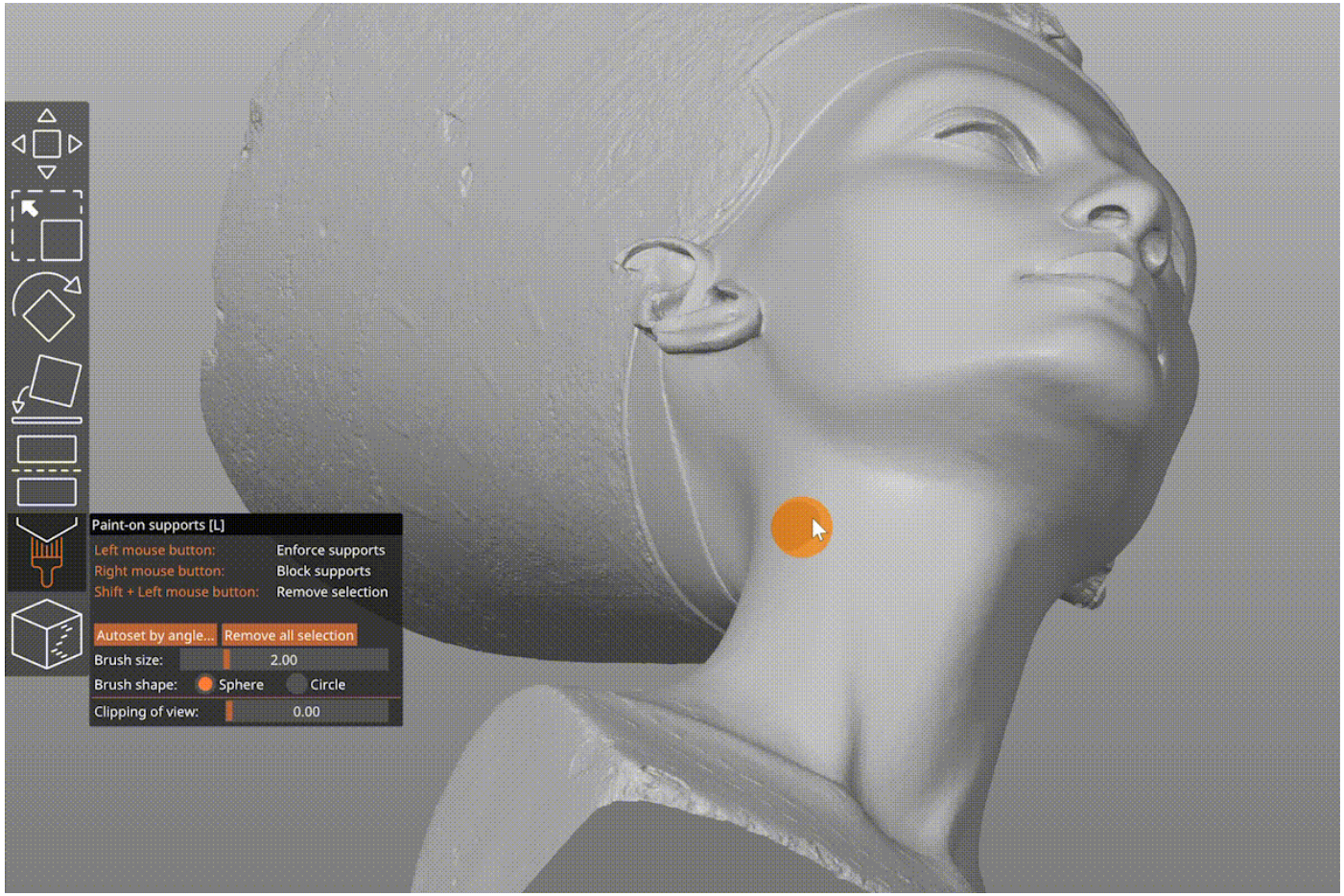
Keep in mind
that printing all the supports takes extra time and
material and **degrades the surface quality** in places where the supports touch the model.

Supports everywhere



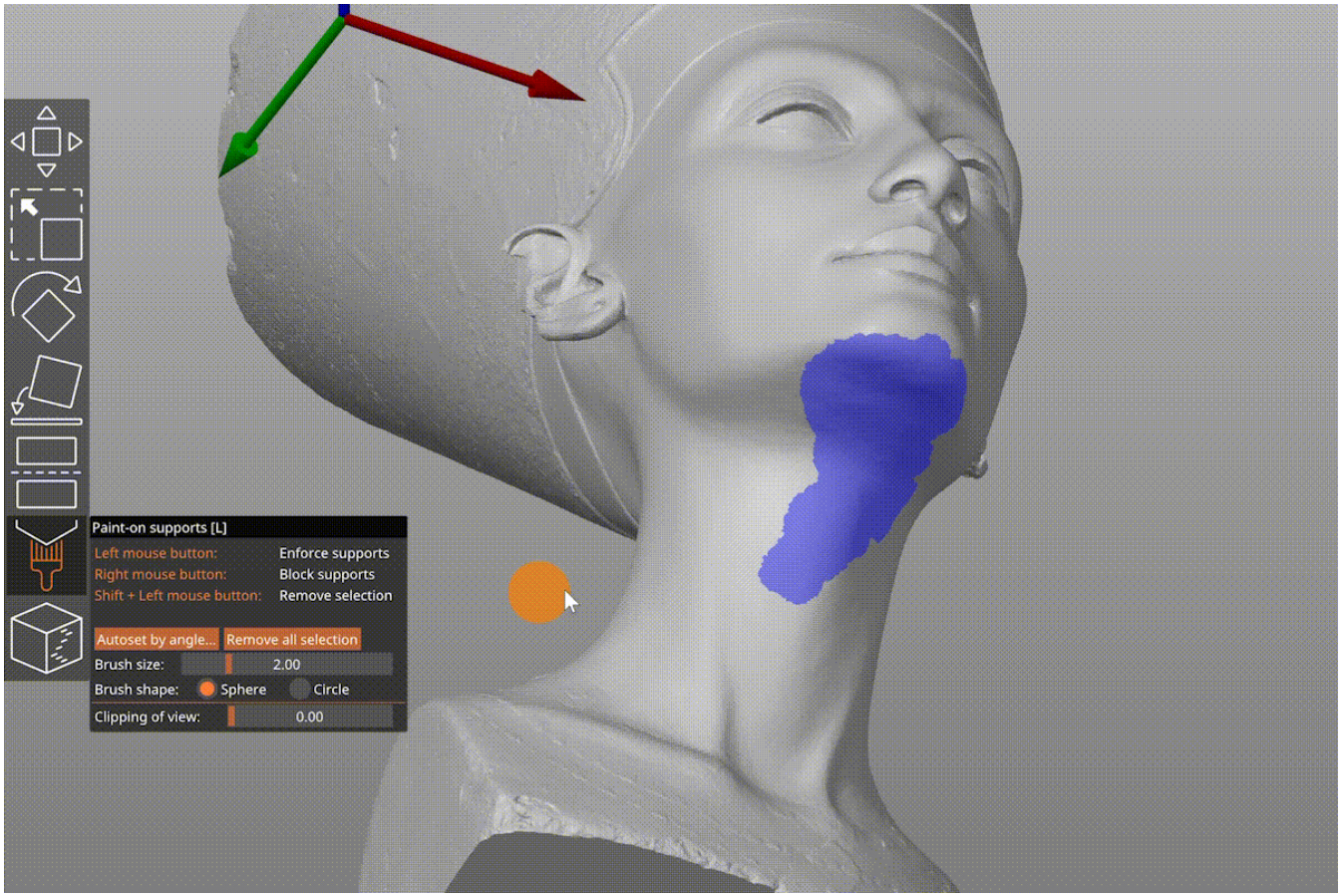
Now, the supports can grow not only from the print sheet, but **also from other printed parts of the model**

For support enforcers only



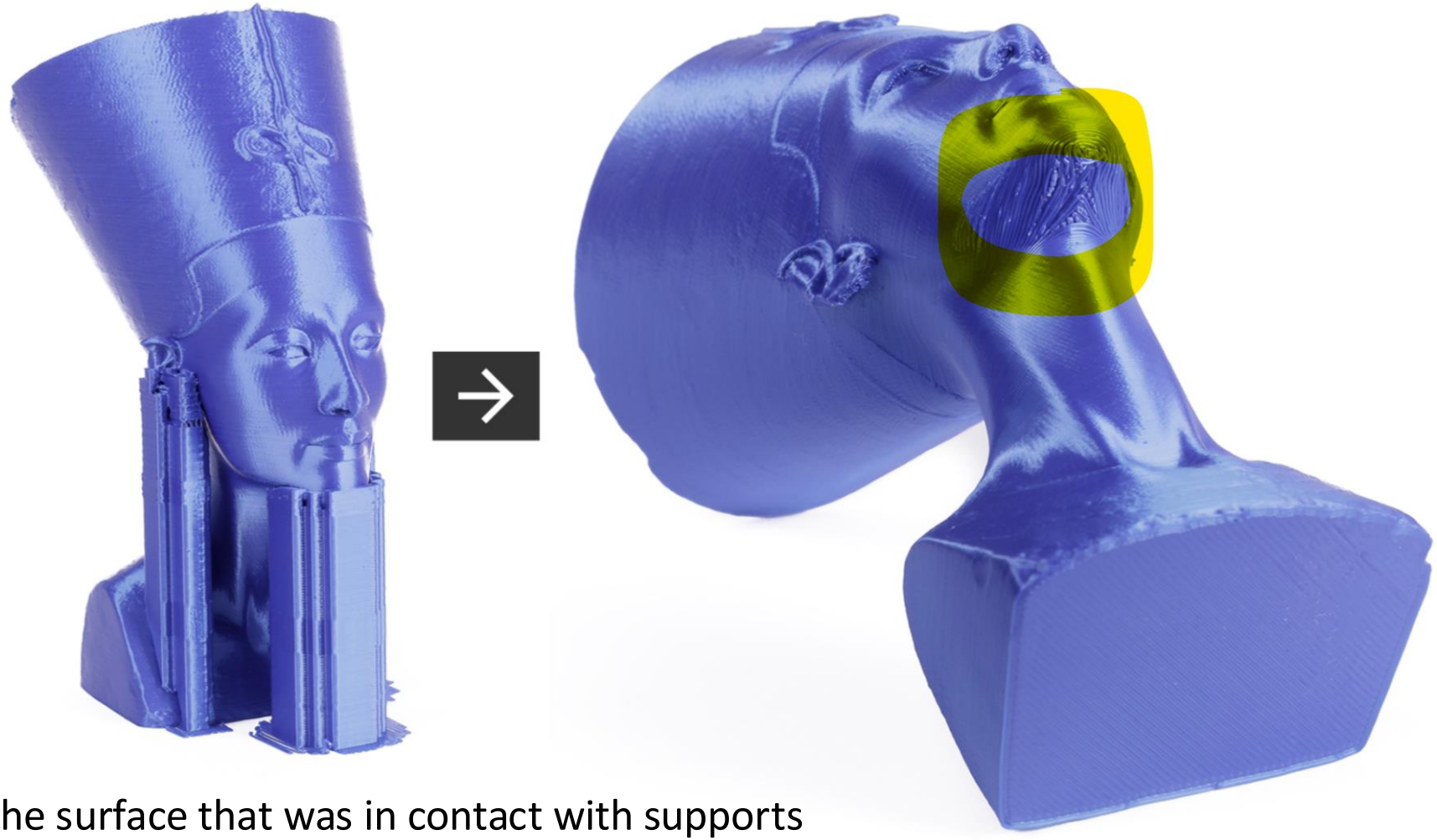
Paint-on supports: select the exact places (marked blue) where the supports should be.

For support enforcers only



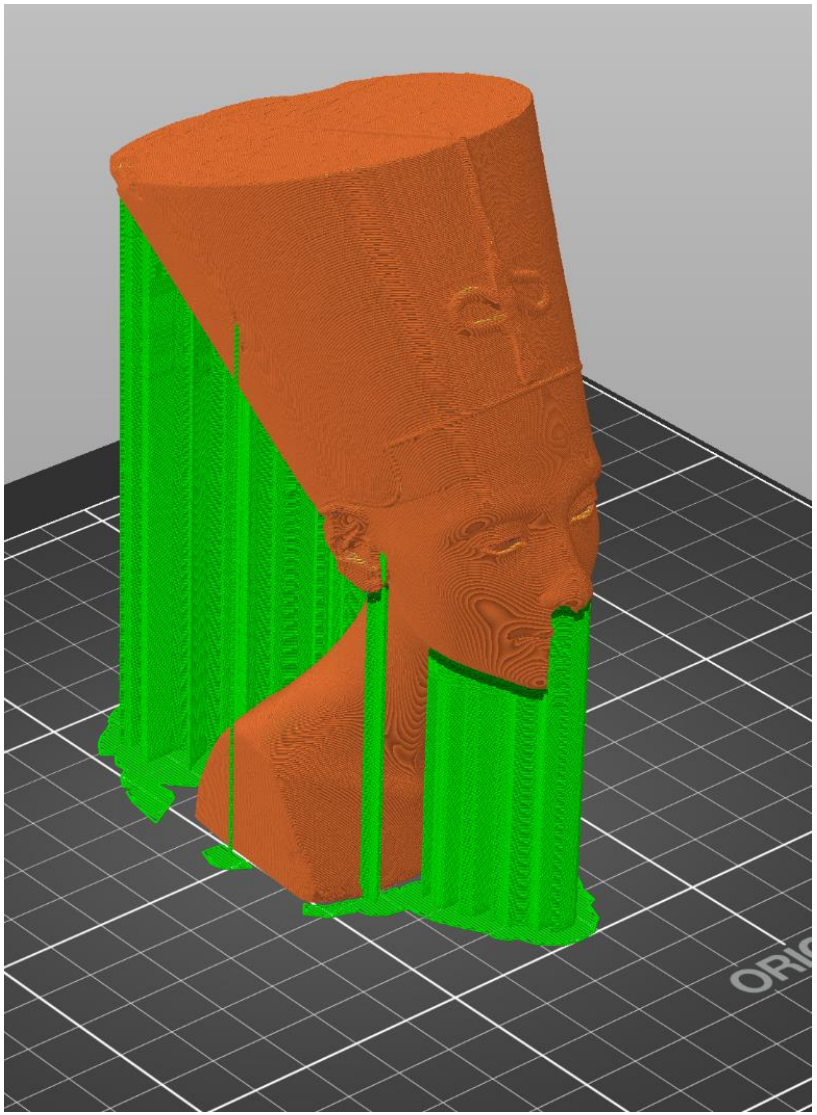
Paint-on supports: right mouse button, you add blockers instead of enforcers - those areas will be marked red.

Supports detrimental aspect

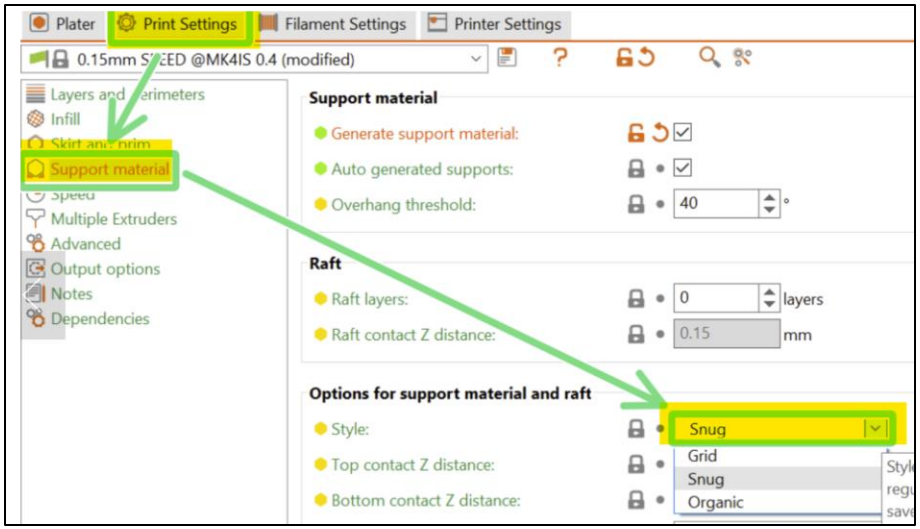


The surface that was in contact with supports will **never look as smooth as the rest** of the model.

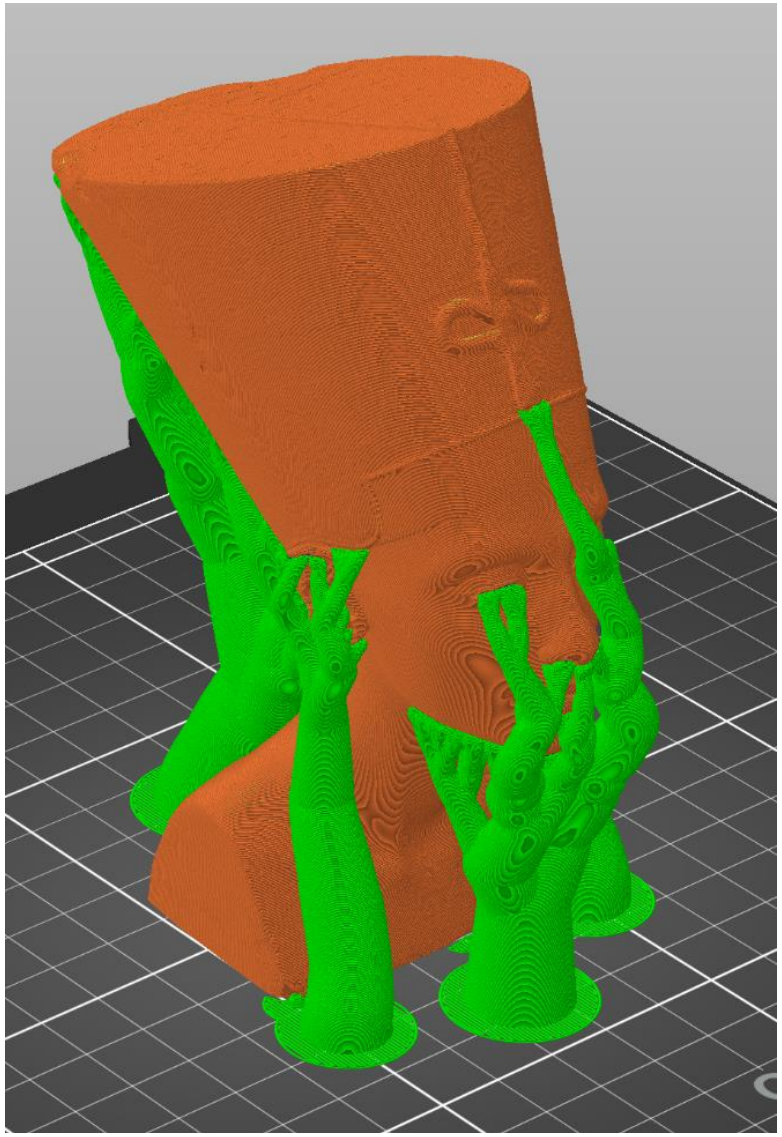
Different styles of supports



The **snug** supports are relatively **subtle**, therefore they spend **less material**, have a **less detrimental effect on the surface quality**, and are **easier to remove** from the model than the grid supports.



Different styles of supports



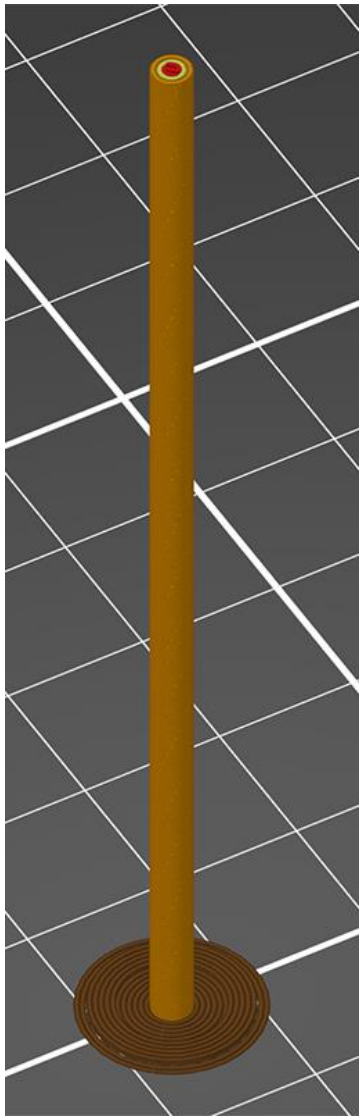
The **organic supports** can twist and bend to bypass the body of the model.

BRIM

Infill: 15% Brim:

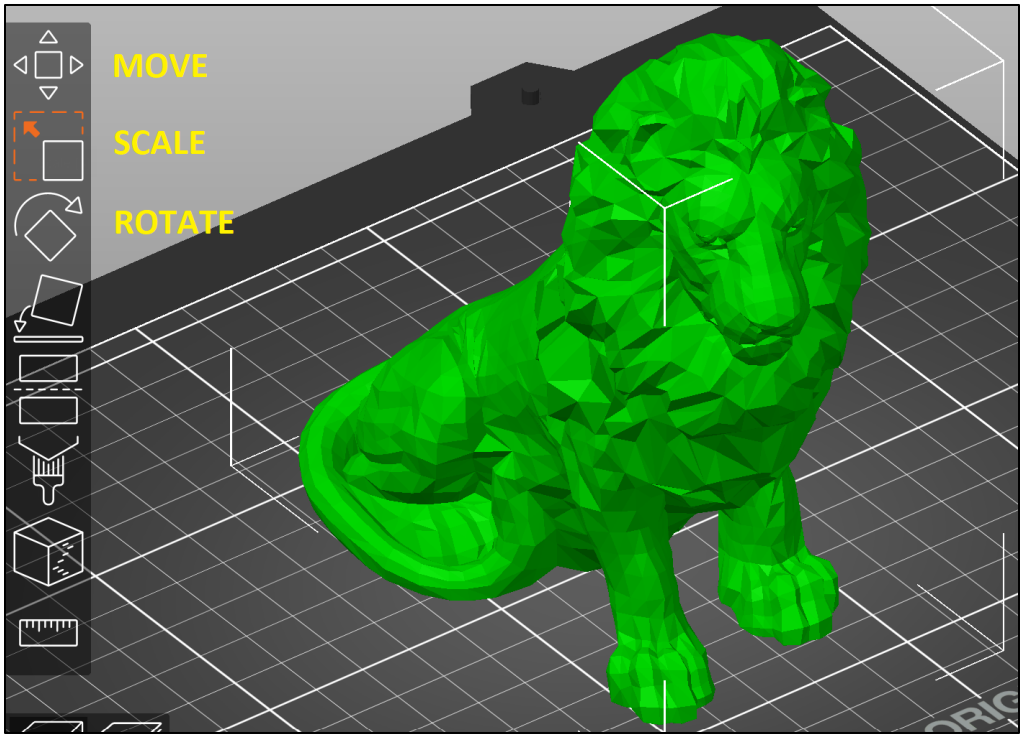
This option will generate a thin layer around the base of the model, **which will improve the adhesion to the print bed.**

This is an option for models that don't have enough of a grip in proportion to their size.

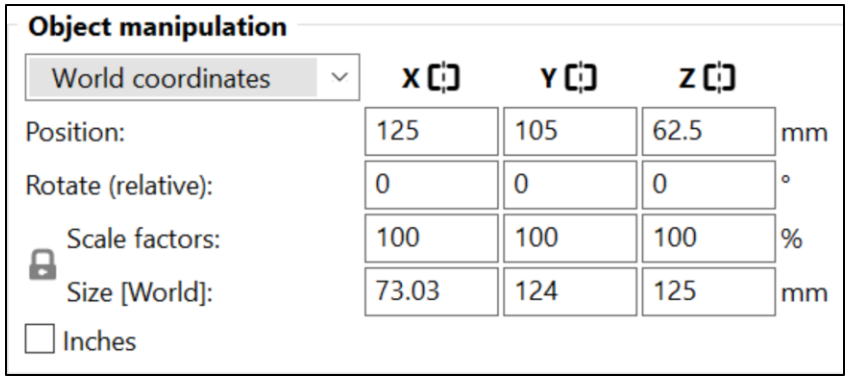


An example of **needed** Brim

Object manipulation



This menu basically duplicates the left icon bar functions **Move, Rotate, and Scale.**



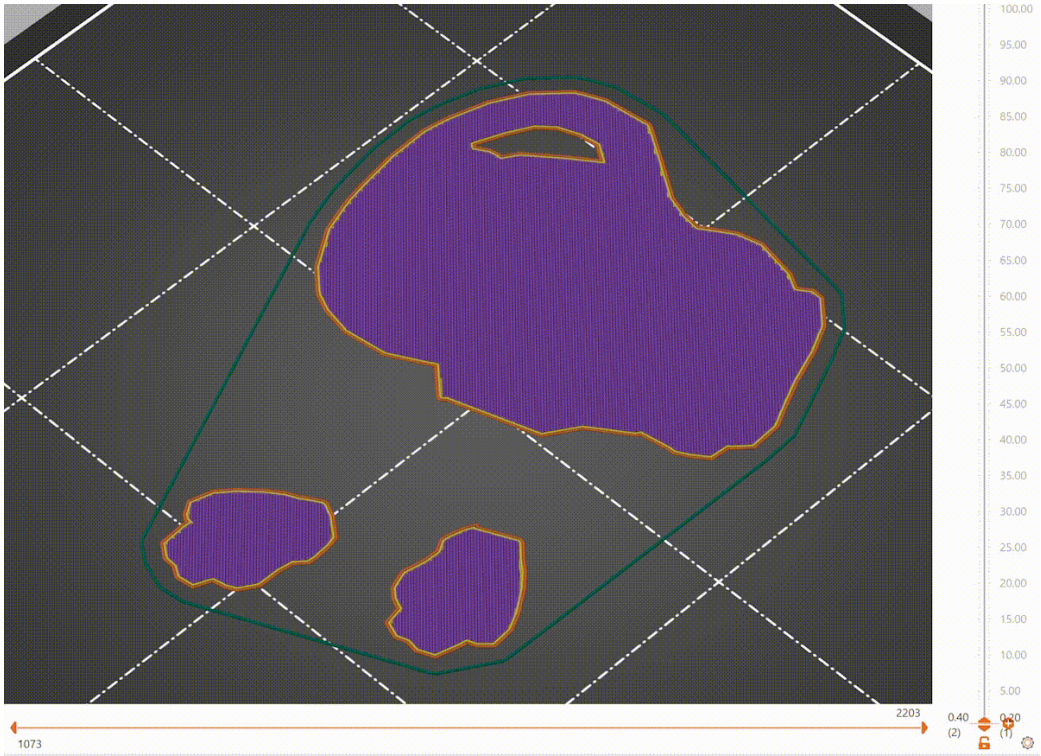
Object manipulation

World coordinates	X	Y	Z	
Position:	125	105	62.5	mm
Rotate (relative):	0	0	0	°
Scale factors:	100	100	100	%
Size [World]:	73.03	124	125	mm

☐ Inches

However, instead of manipulating the model with your mouse, here you can dial in the exact numbers

Slice now BUTTON



Finally, you can now slice and export your first model.

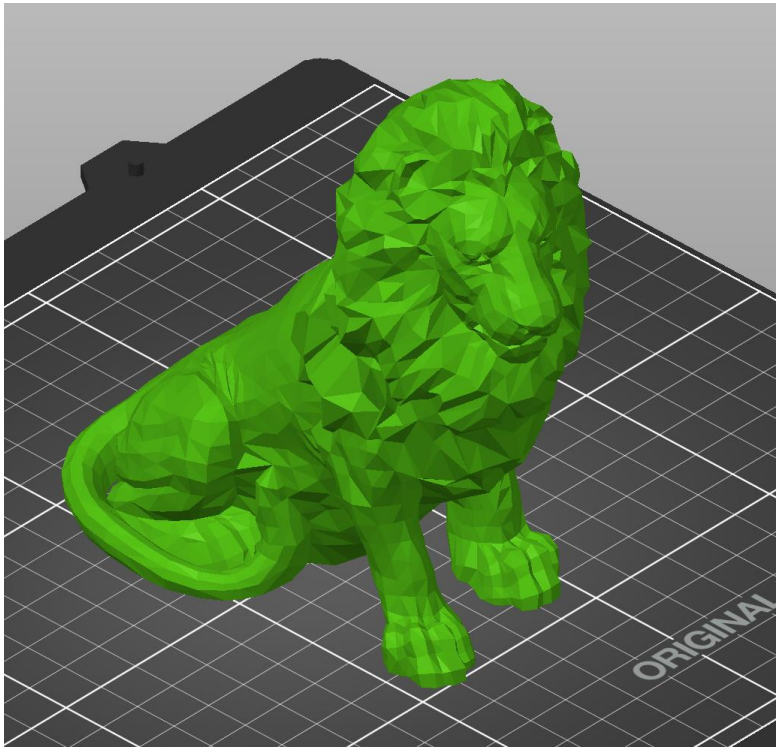
Click the "Slice now" button in the lower right corner.

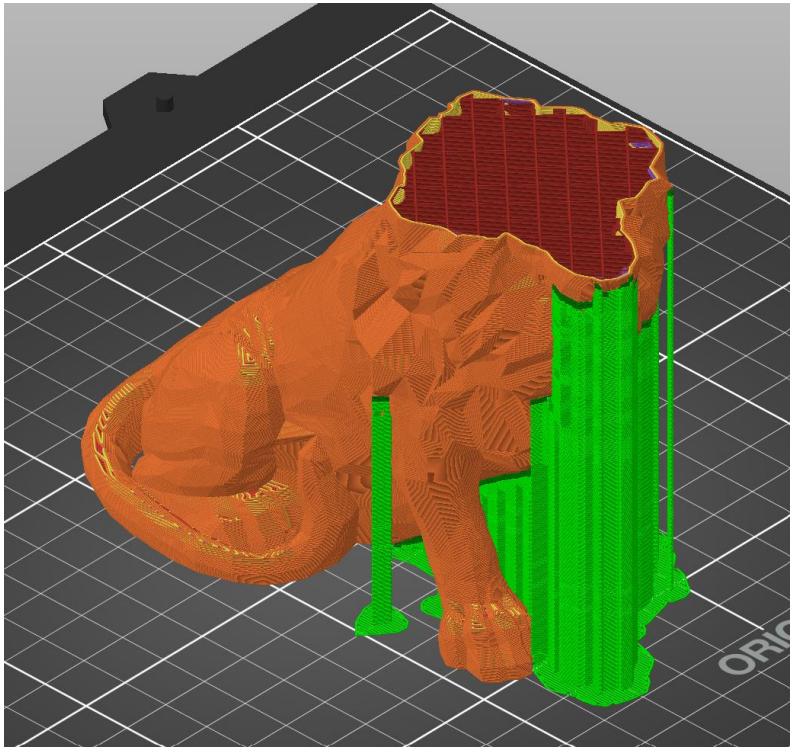
Voilá, you have sliced the lion and generated the G-code.

If you want to print right away, just click on **Export G-code**, download the file on your SD card and you are ready to go

LEFT BAR ICONS: View Toggle

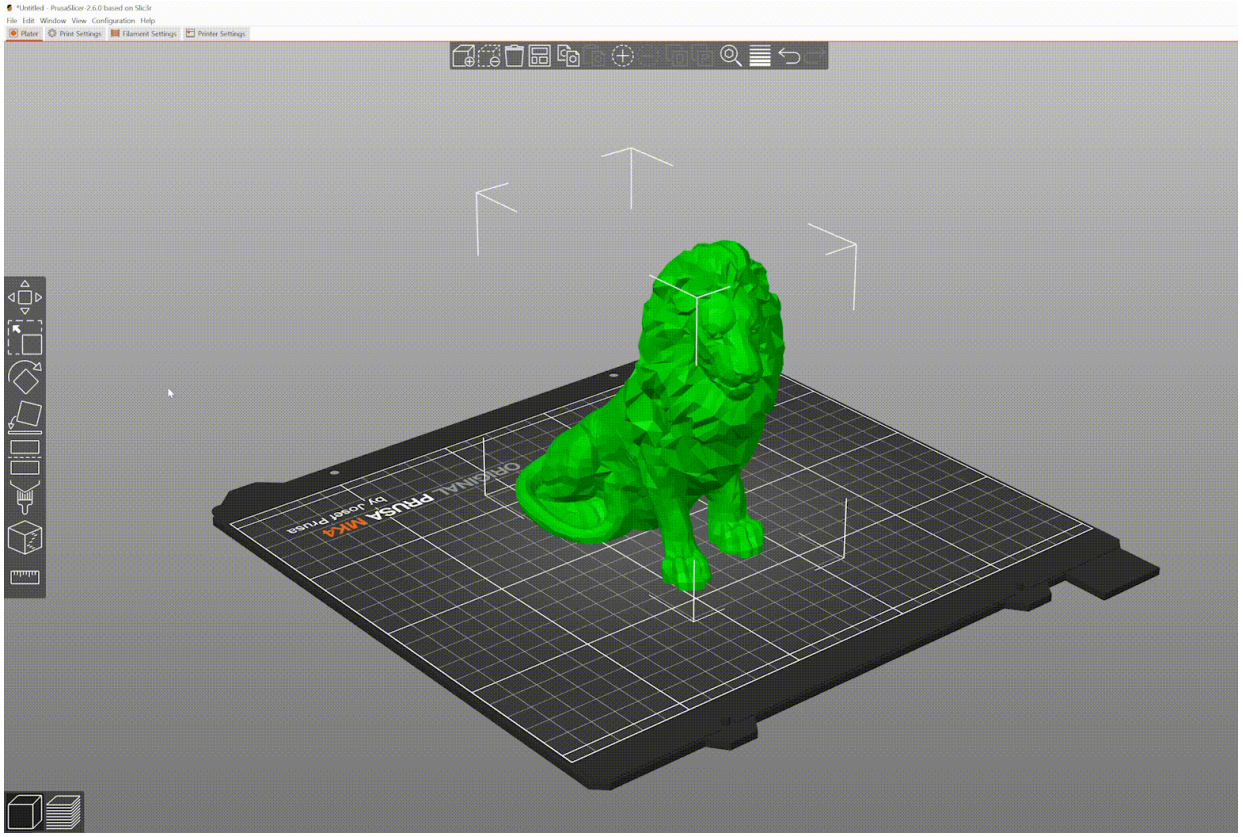
Using these icons in the lower-left corner of your screen, **you can toggle between the 3D editor view** (i.e. how did the scene look like before clicking **Slice Now**) and the sliced **G-code Preview**.



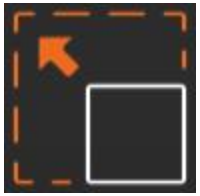


LEFT BAR ICONS: MOVE

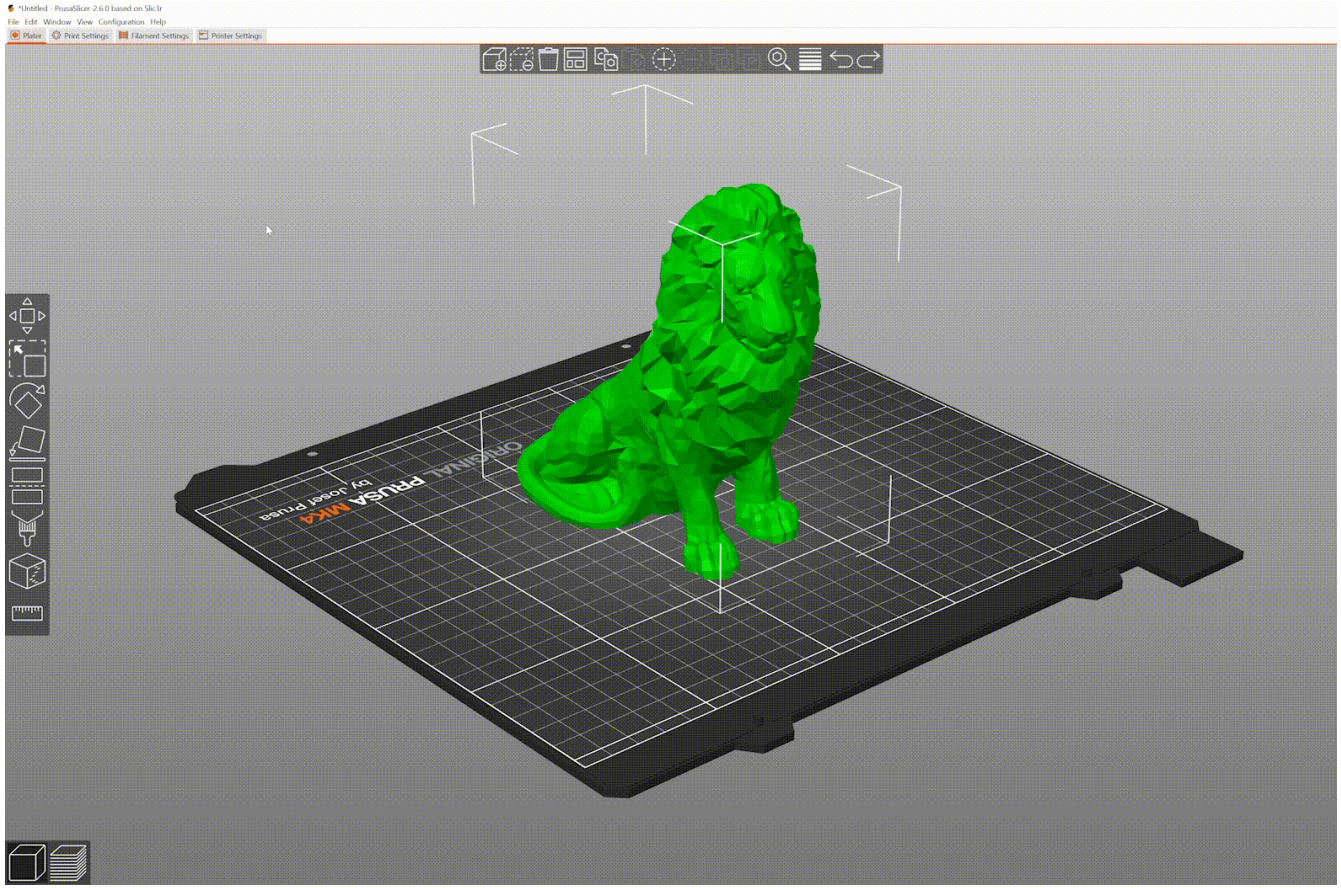
Simply drag the little boxes to move the model in each of the three axes. You can also move the model simply by dragging it with mouse directly.



LEFT BAR ICONS: SCALE

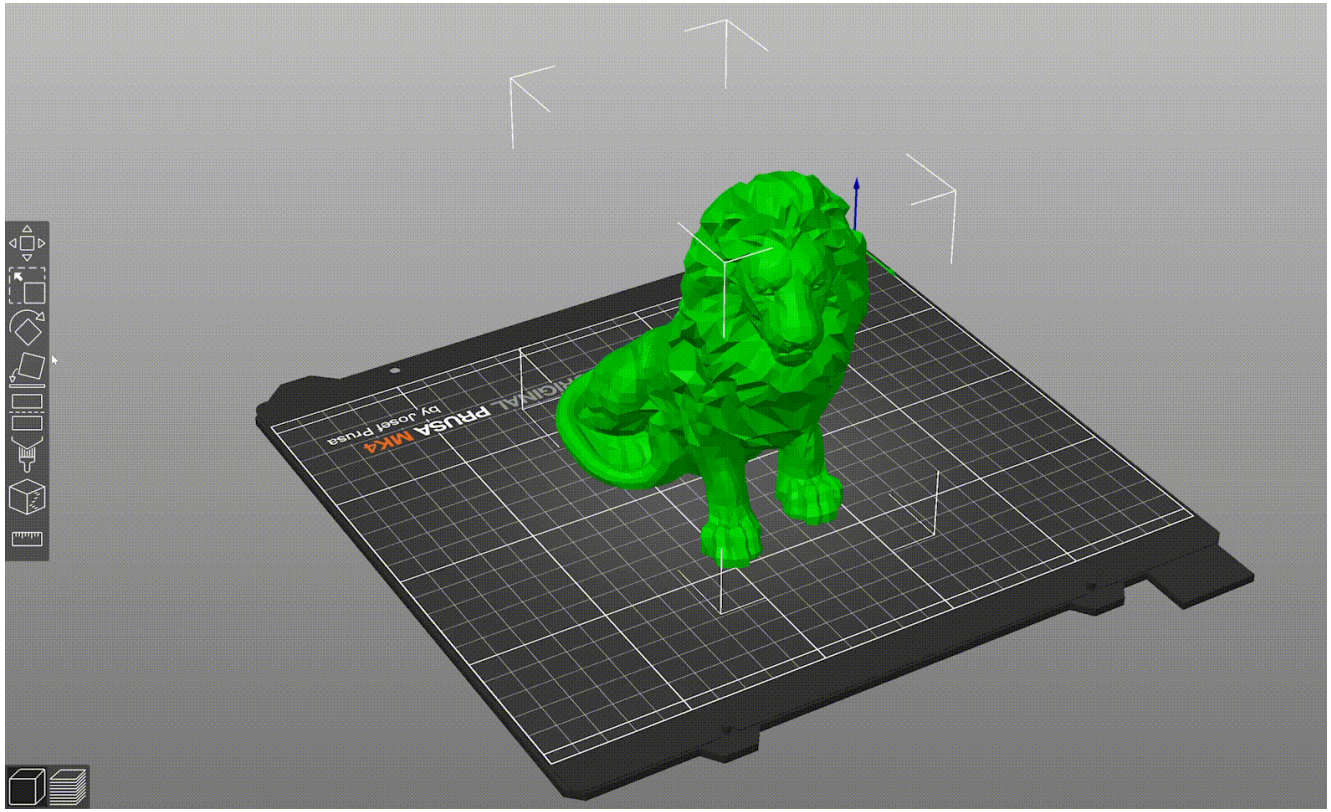


Drag the boxes to change size (or proportions). **Use the orange boxes to keep the same proportions.** All the other boxes will stretch or squeeze the model.



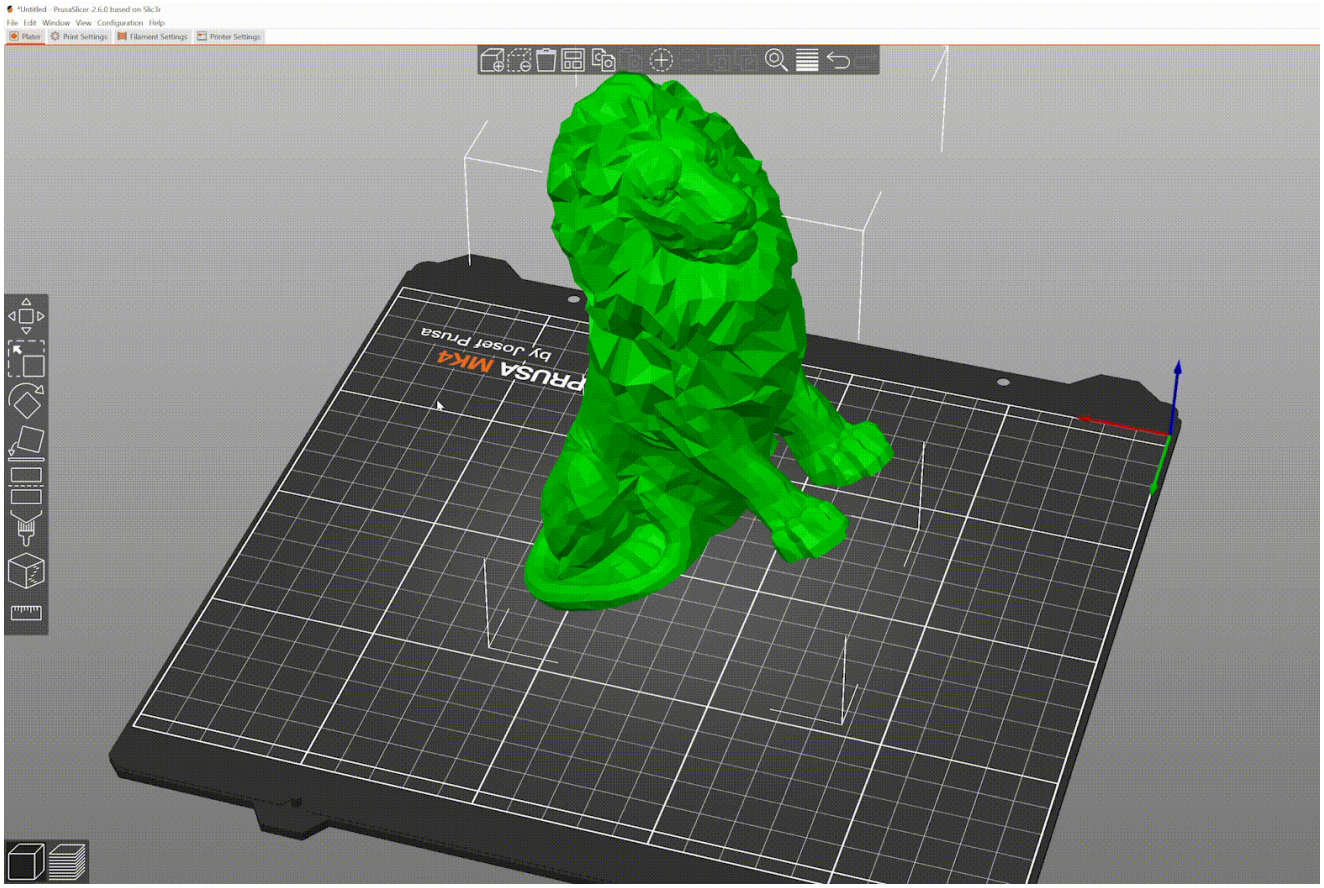
LEFT BAR ICONS: ROTATE

Rotate the model. Use the longer white beams to rotate in larger increments (it will "snap" through 45°, 90°, etc.), or the shorter beams for smaller increments.

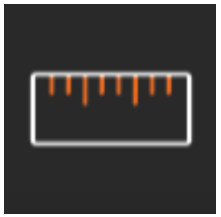


LEFT BAR ICONS: Place on face

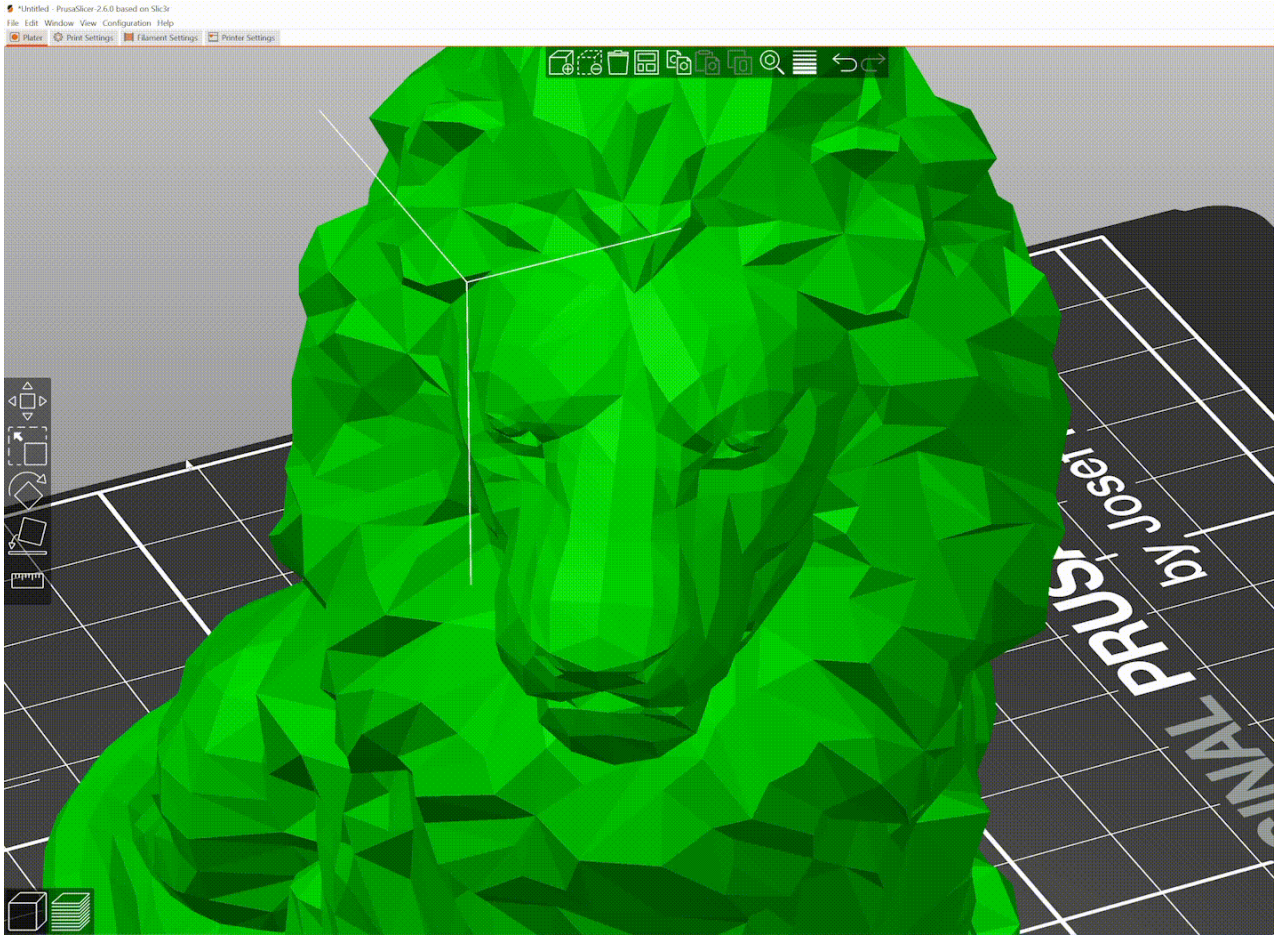
Place on face: Choose which surface to align with the print bed. Click on the bottom of the model to put the lion back on his feet.



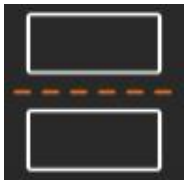
LEFT BAR ICONS: Measure



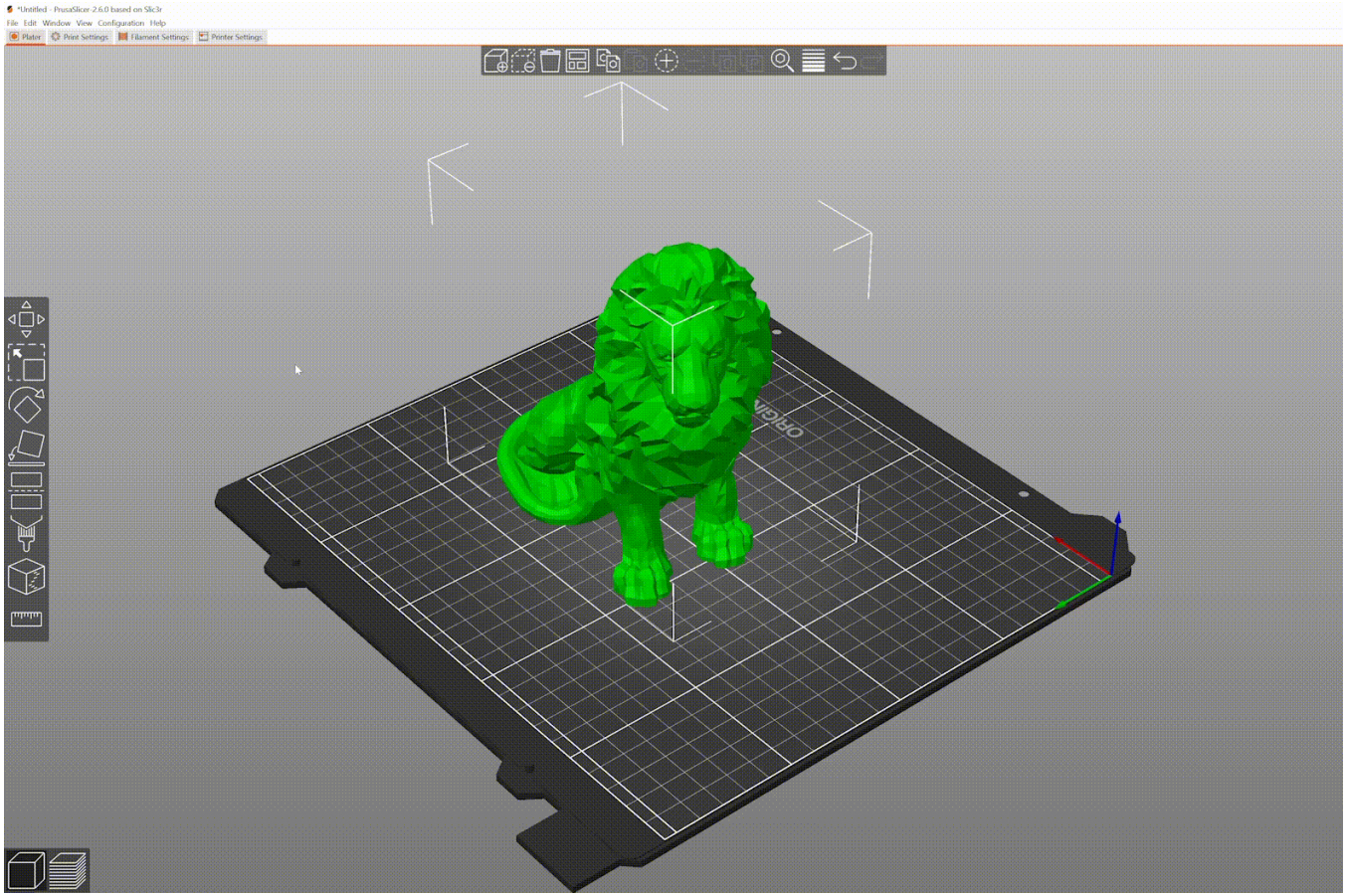
Measure: You can select two points, edges, or faces of the model, and the program will measure their distance..



LEFT BAR ICONS: Cut

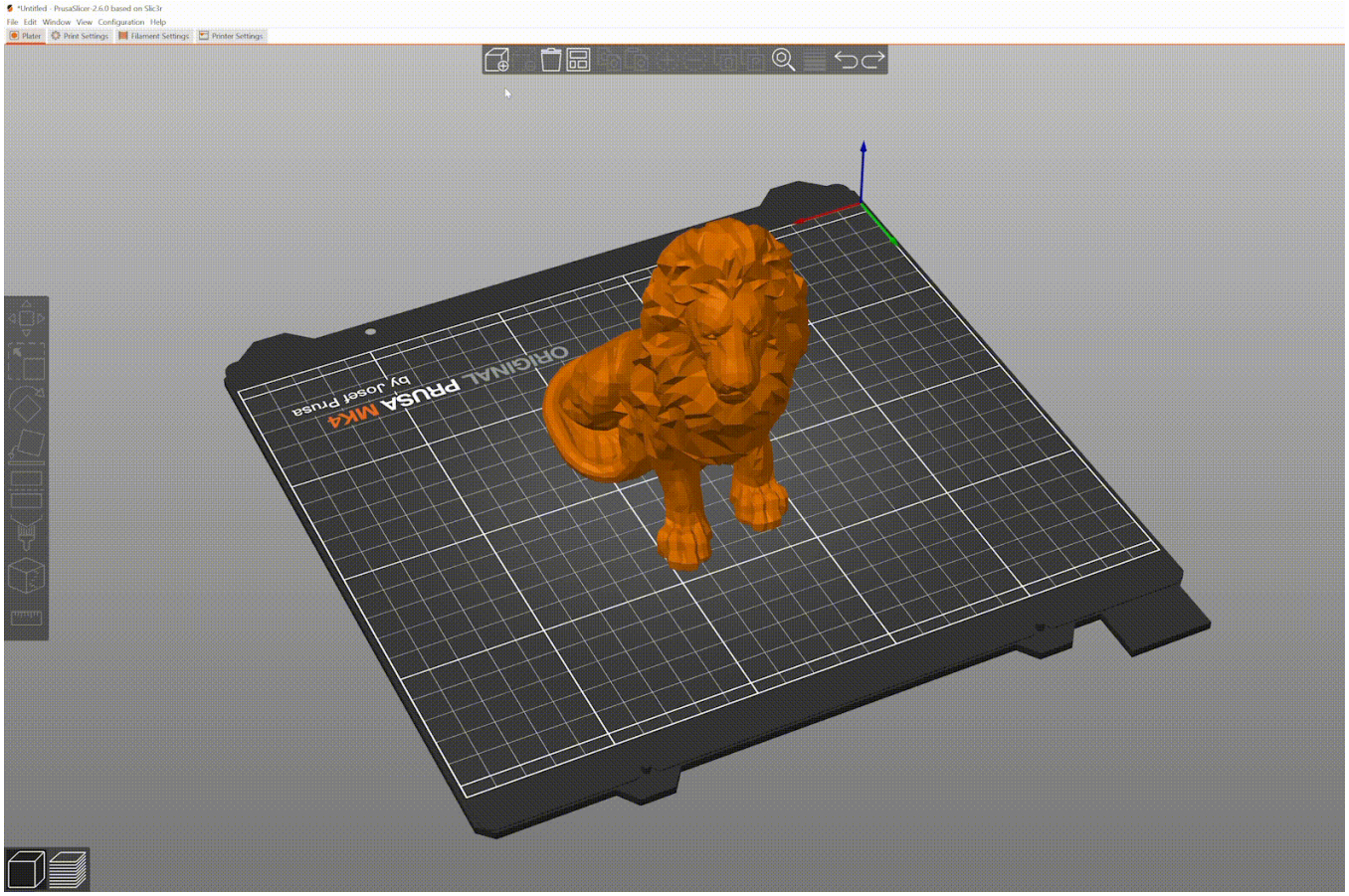


Cut the model into more parts. if you need to split a model too large for the print volume or too difficult to print as one piece. Here we show the most basic use of the tool - simple cutting by a horizontal plane.



Interface: Central icon bar – Add Button

Add another model (yes, you can print models from more different files at once). To demonstrate, we added a new model, the **Nefertiti bust**. It interferes with the lion, but don't worry about that for now.

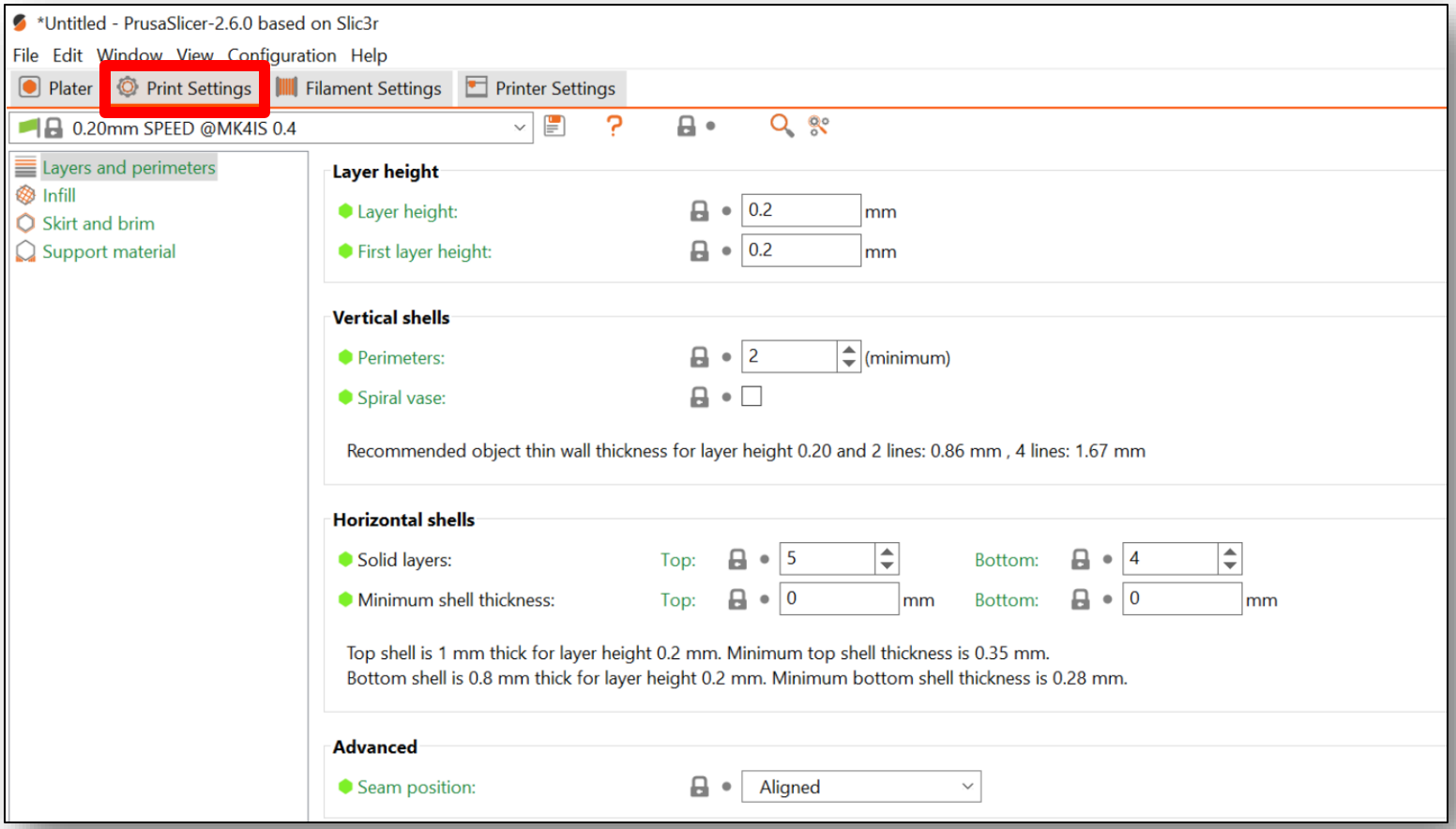


Interface: Central icon bar – Arrange

With this button, you can automatically arrange both models so they don't obstruct each other.

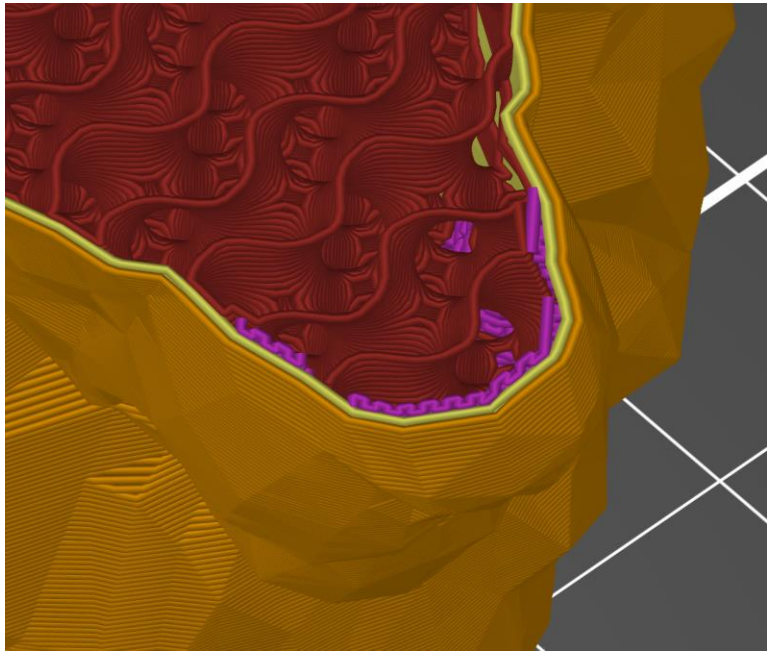
Upper menu tabs: Print Settings

On previous slides we covered all the menus and icon bars on the main screen. Now we will go through the settings that are available in the upper menu tabs. If you want to return to the main screen, click on the **Plater** tab.

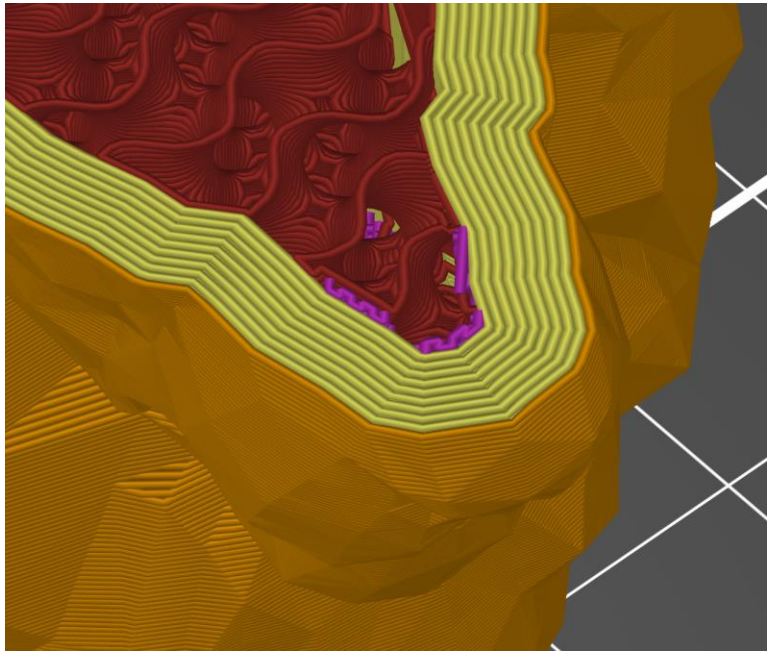


Print Settings: Perimeters

How many layers of filament form the outer shell of the model.



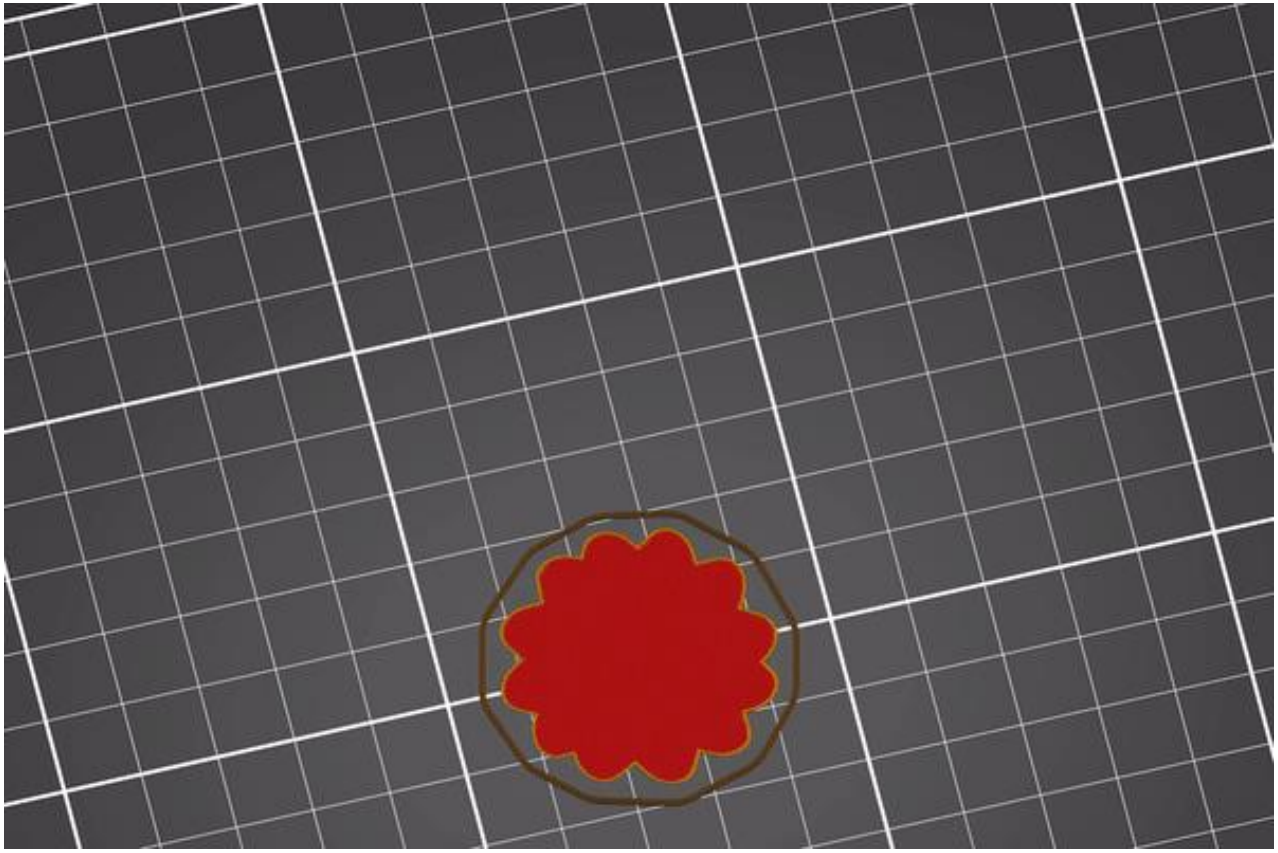
The usual minimum is 2 perimeters (just one would be too fragile).



10 perimeters - rather an impractical extreme, shown here as an example.

Print Settings: Spiral vase

The spiral vase mode means that the nozzle does not print individual layers one by one, but executes the whole print **with one continuous spiral movement.** This results in a nice smooth surface.

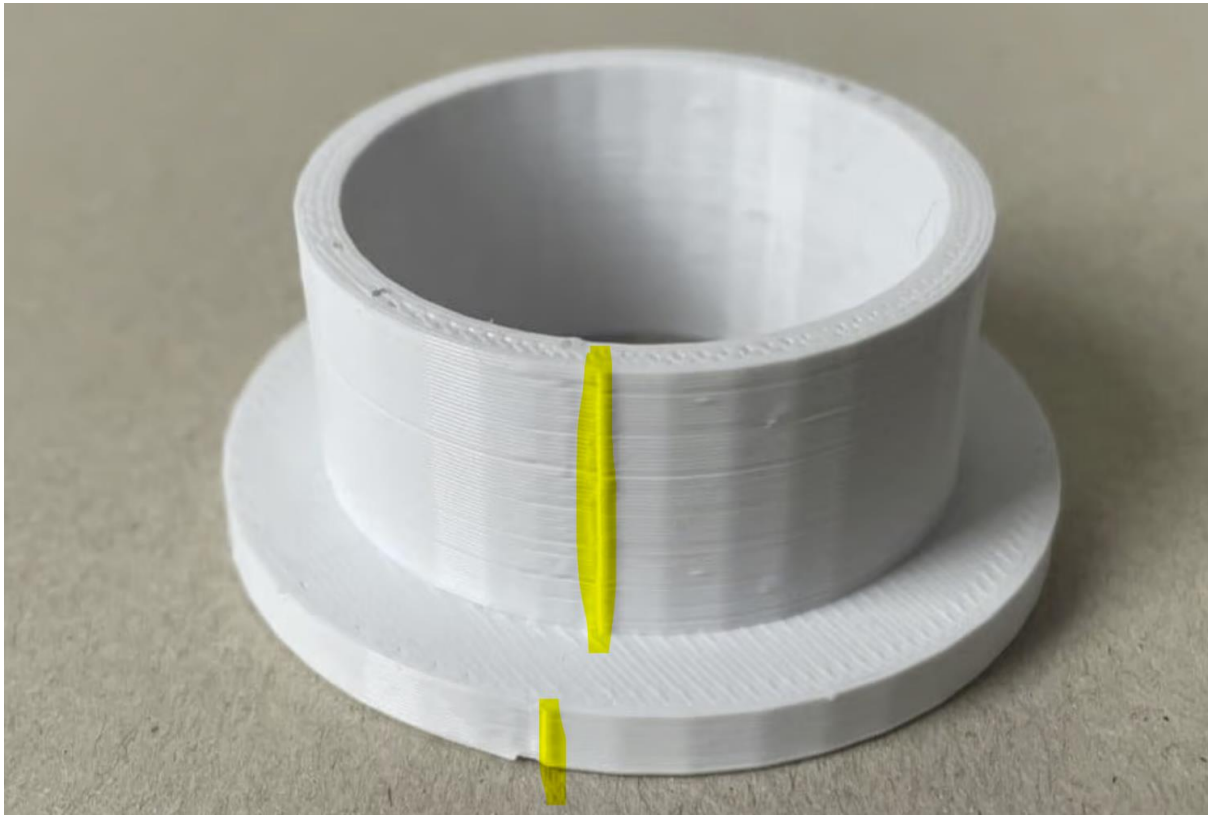


Print Settings: Spiral vase

However, when using the spiral vase mode **the walls of the print are very thin** (only a single perimeter) with no use for structural parts. Nevertheless, is the fastest of the print modes.

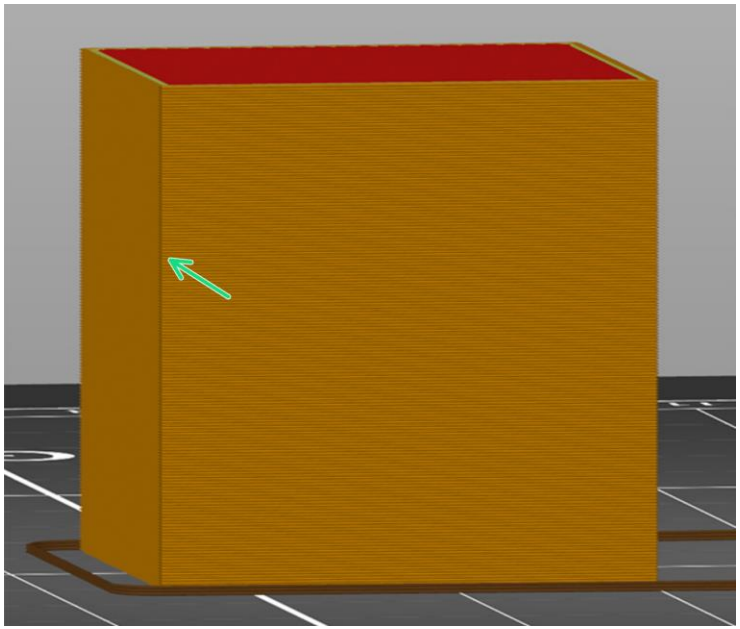
Print Settings: Seam position

Each printed layer ends with a seam - which looks like a zit on the model surface. It marks the place where the nozzle lifts off after finishing a layer. It's an inevitable feature, **apart from the Spiral vase mode.**

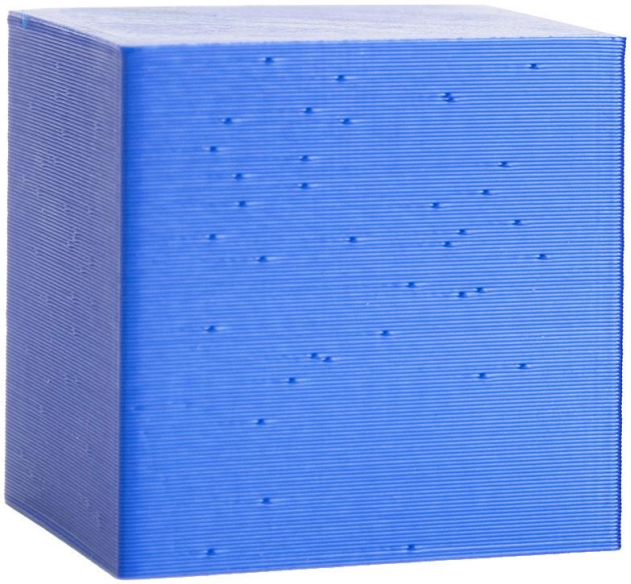


Print Settings: Seam position

PrusaSlicer can attempt **to hide** the seams into model edges (aligned) or by spread them randomly on the surface (random)



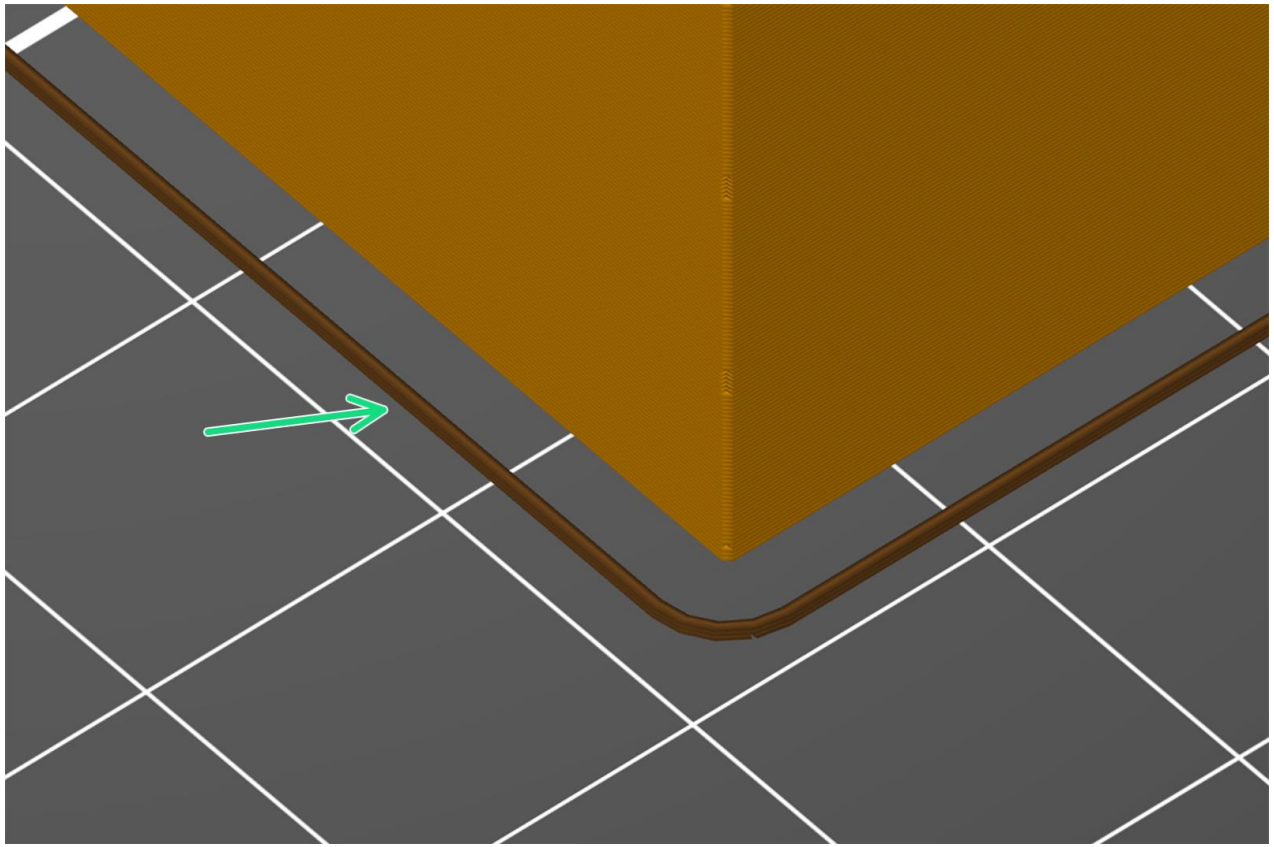
Aligned



Random

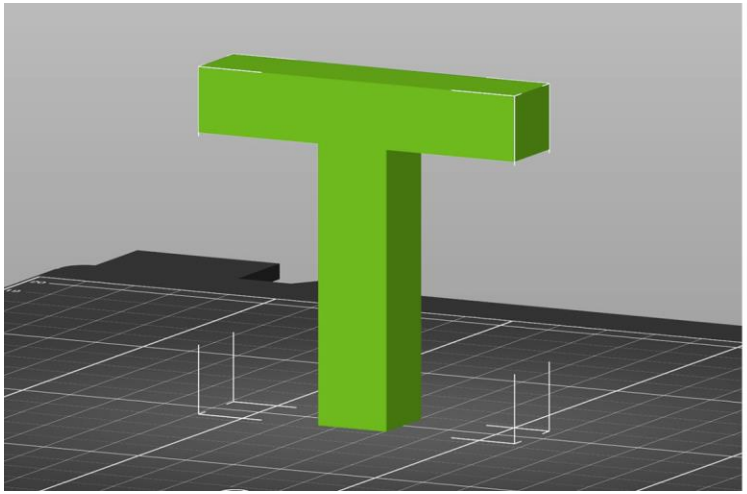
Print Settings: Skirt

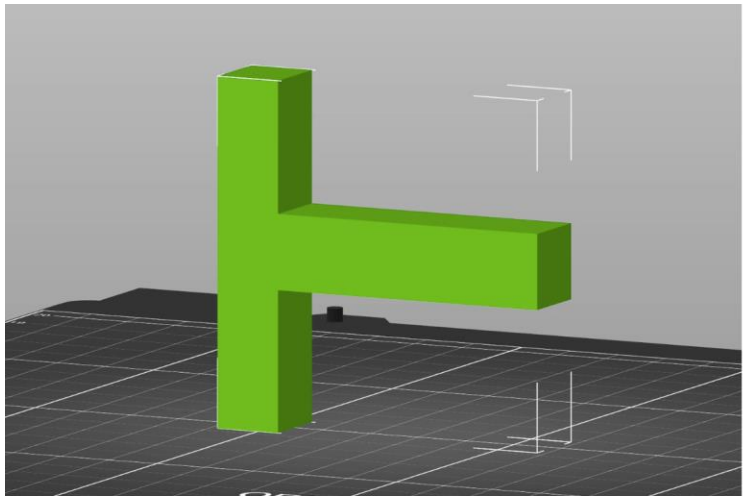
The skirt is a thin printed outline around all the models on the print bed. **Its purpose is to stabilize the filament flow through the nozzle.**

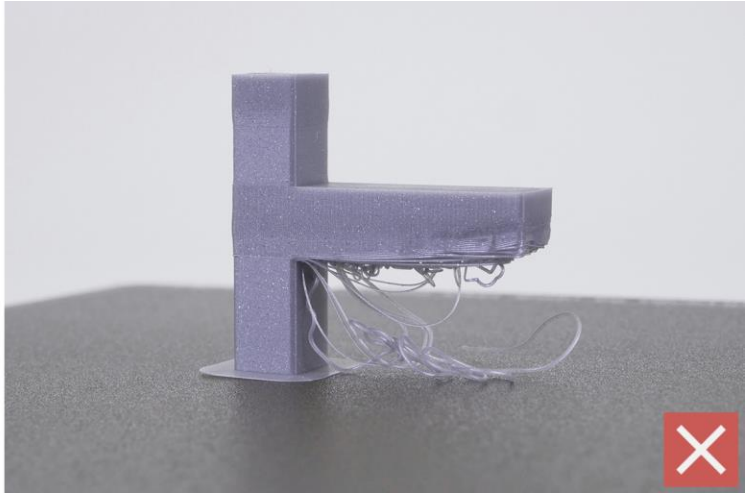


Print Orientation

The orientation of the model on the print bed is very important - by finding the ideal position you can avoid the supports.



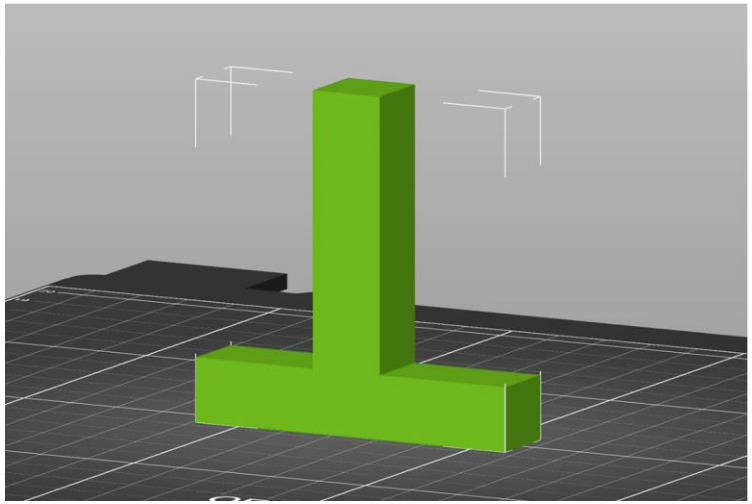


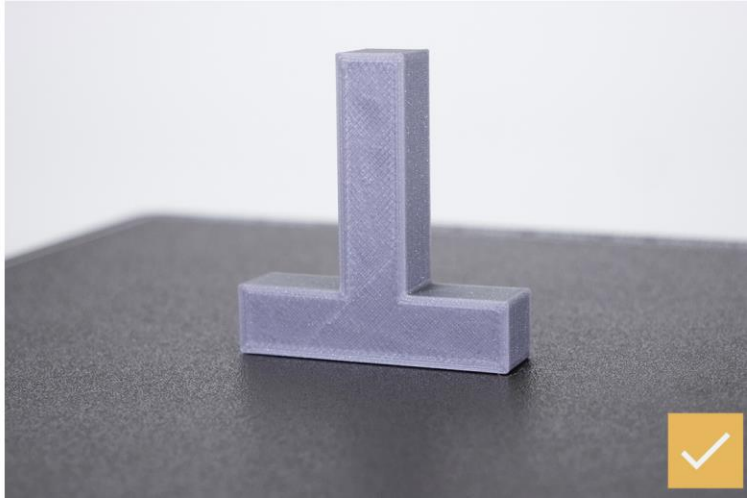


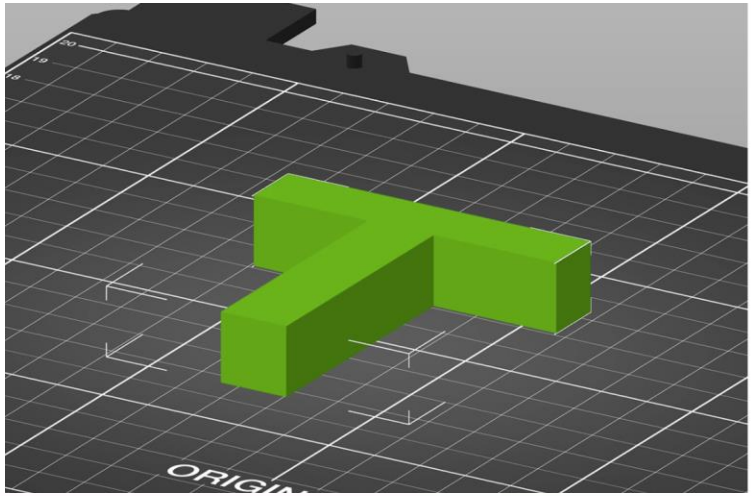
Examples of **poor** print stability and poor adhesion to the print surface.

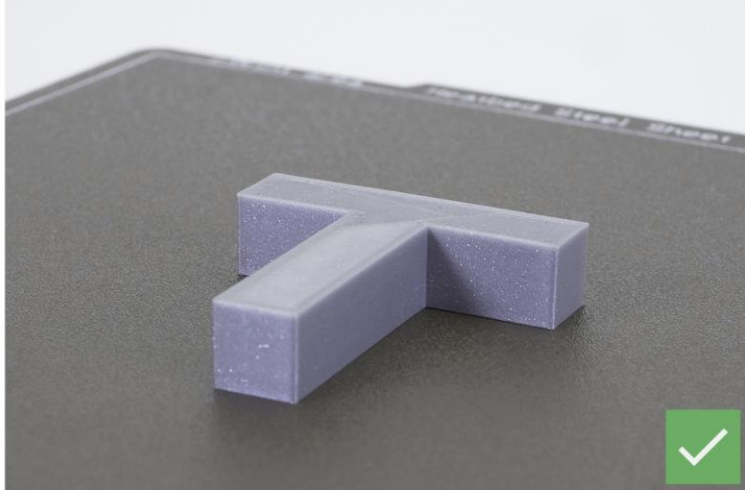
Print Orientation

Proper orientation can have a big impact on aesthetic qualities as well.



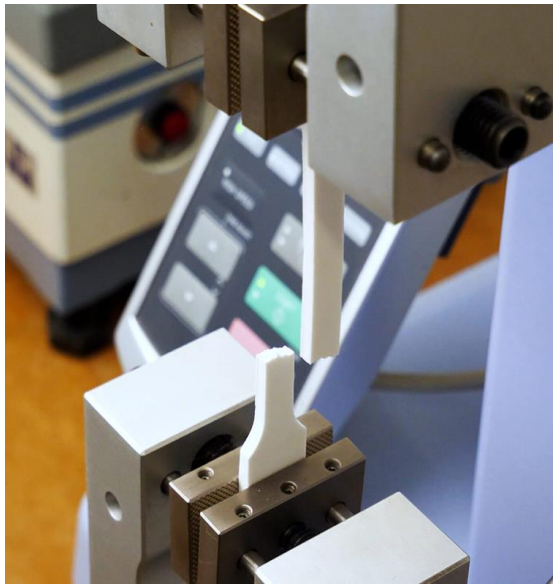


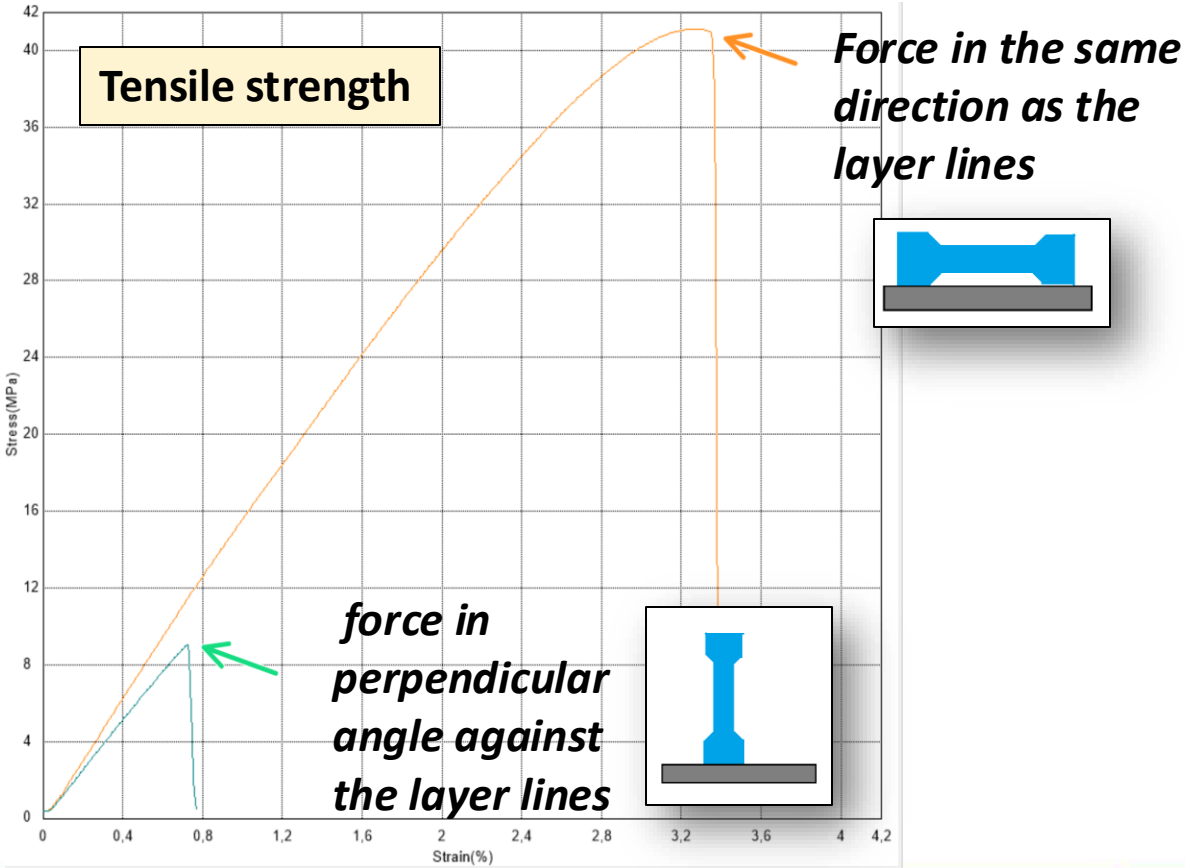




Print Orientation and toughness

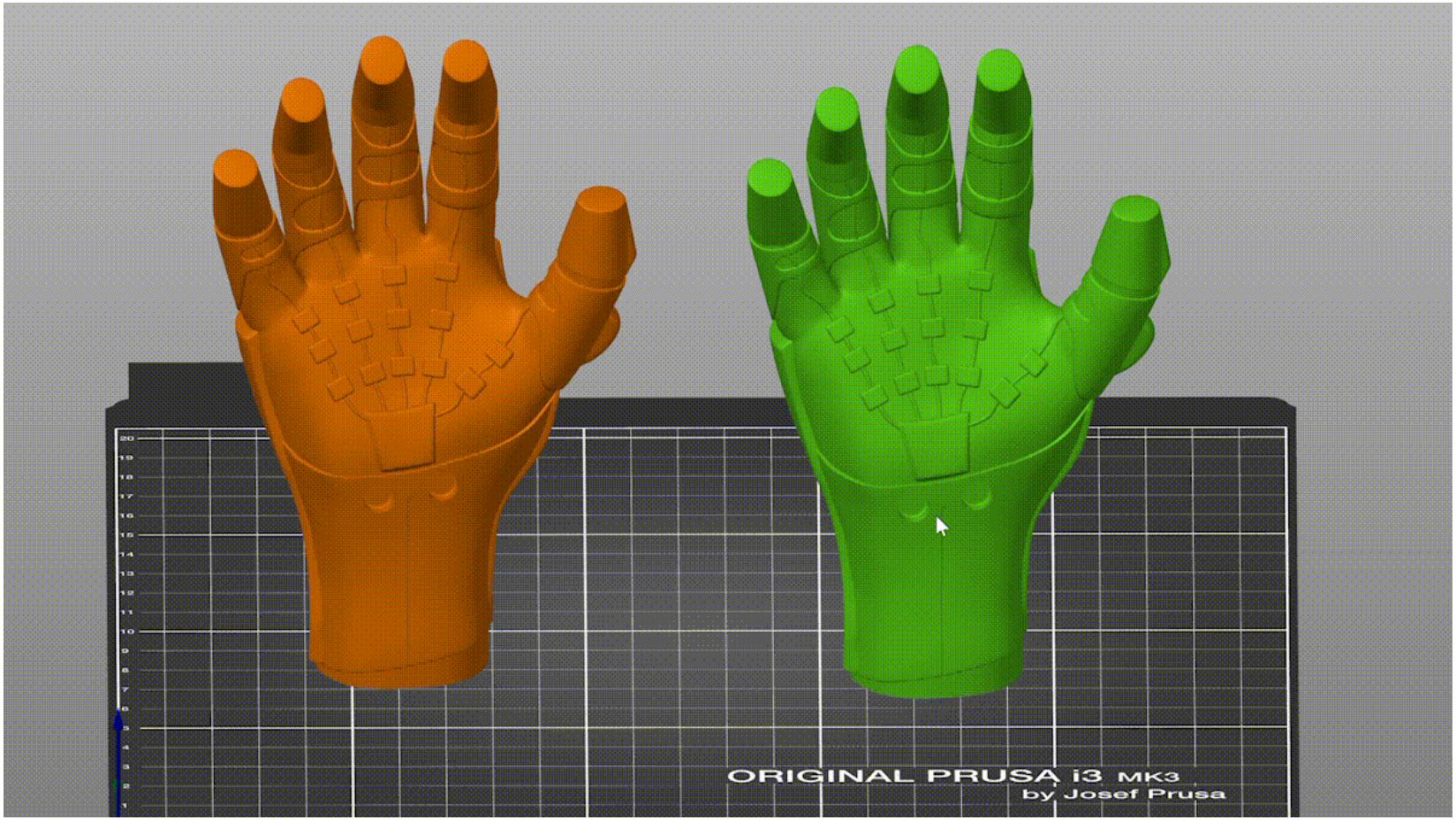
If we try to pull apart the layers from each other, the 3D printed object is much less durable than if we apply force at a *perpendicular* angle to the layer orientation. **The difference can be quite significant** - double the toughness of an injected plastic in one direction, half that toughness in the other.





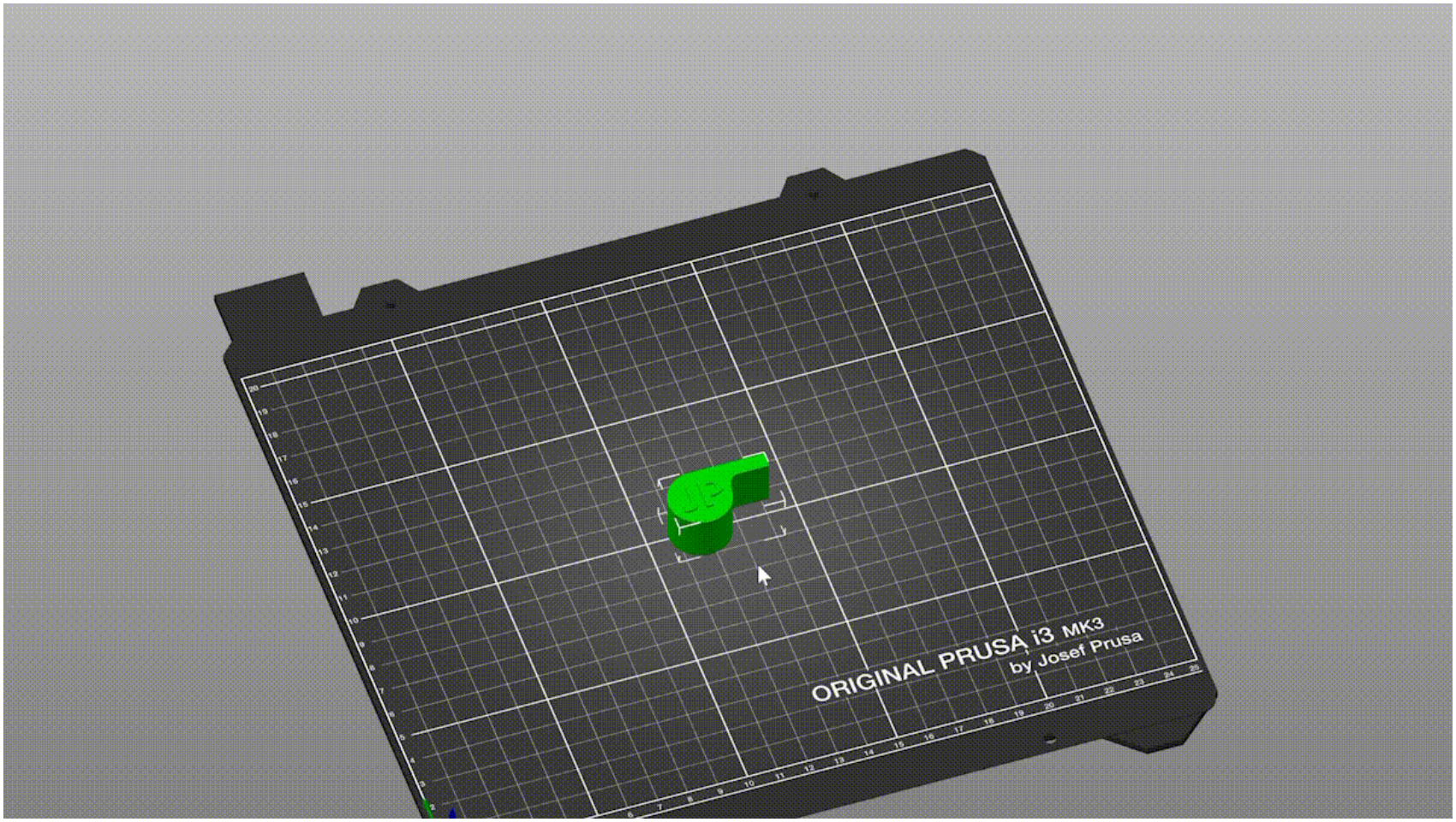
Tips & tricks: Mirror

Right-click a model and select **Mirror** to mirror it.



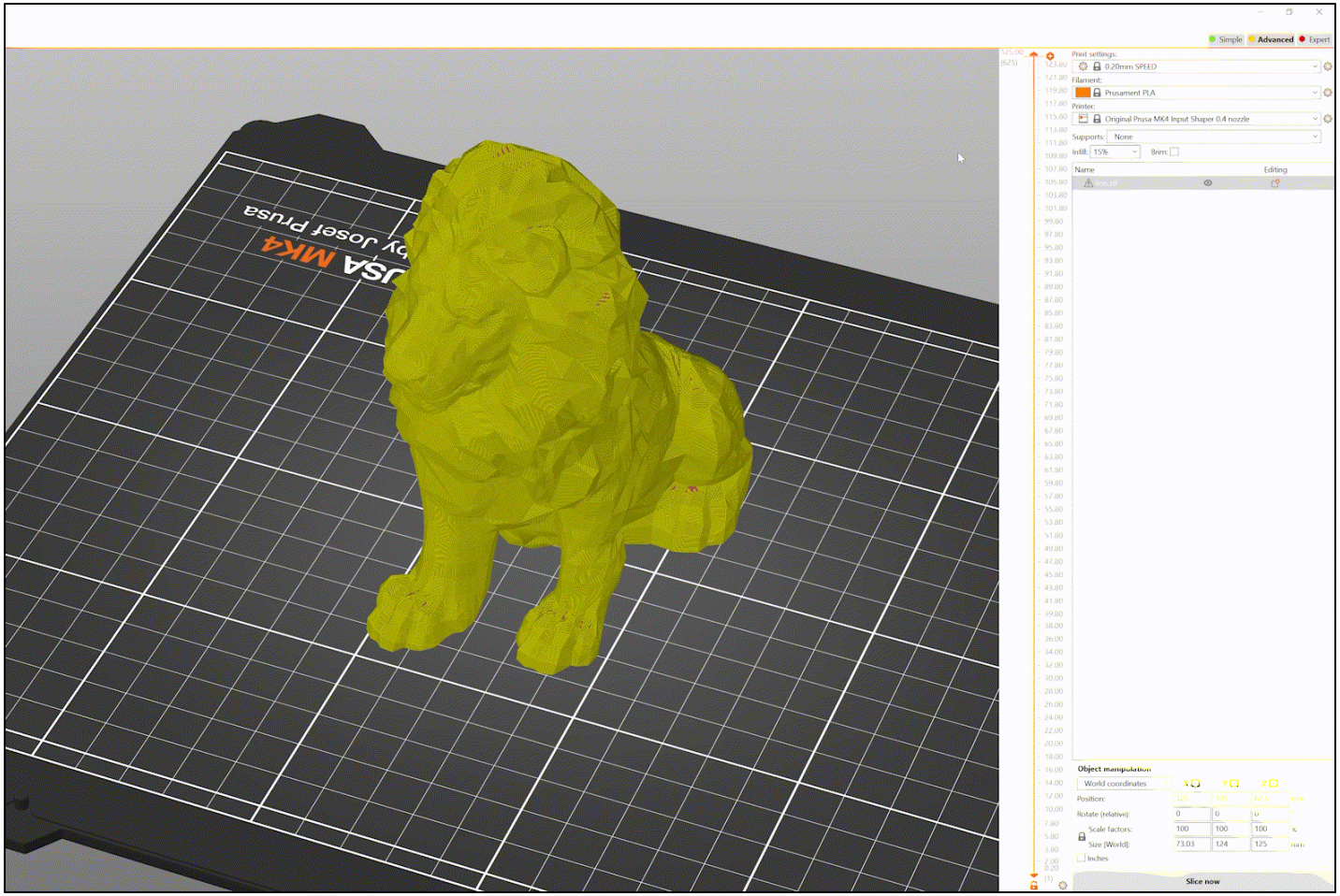
Tips & tricks: Scale to print volume

Right-click and select **Scale to print volume** to automatically scale your model up (or down) exactly to the maximum size your printer can manage.



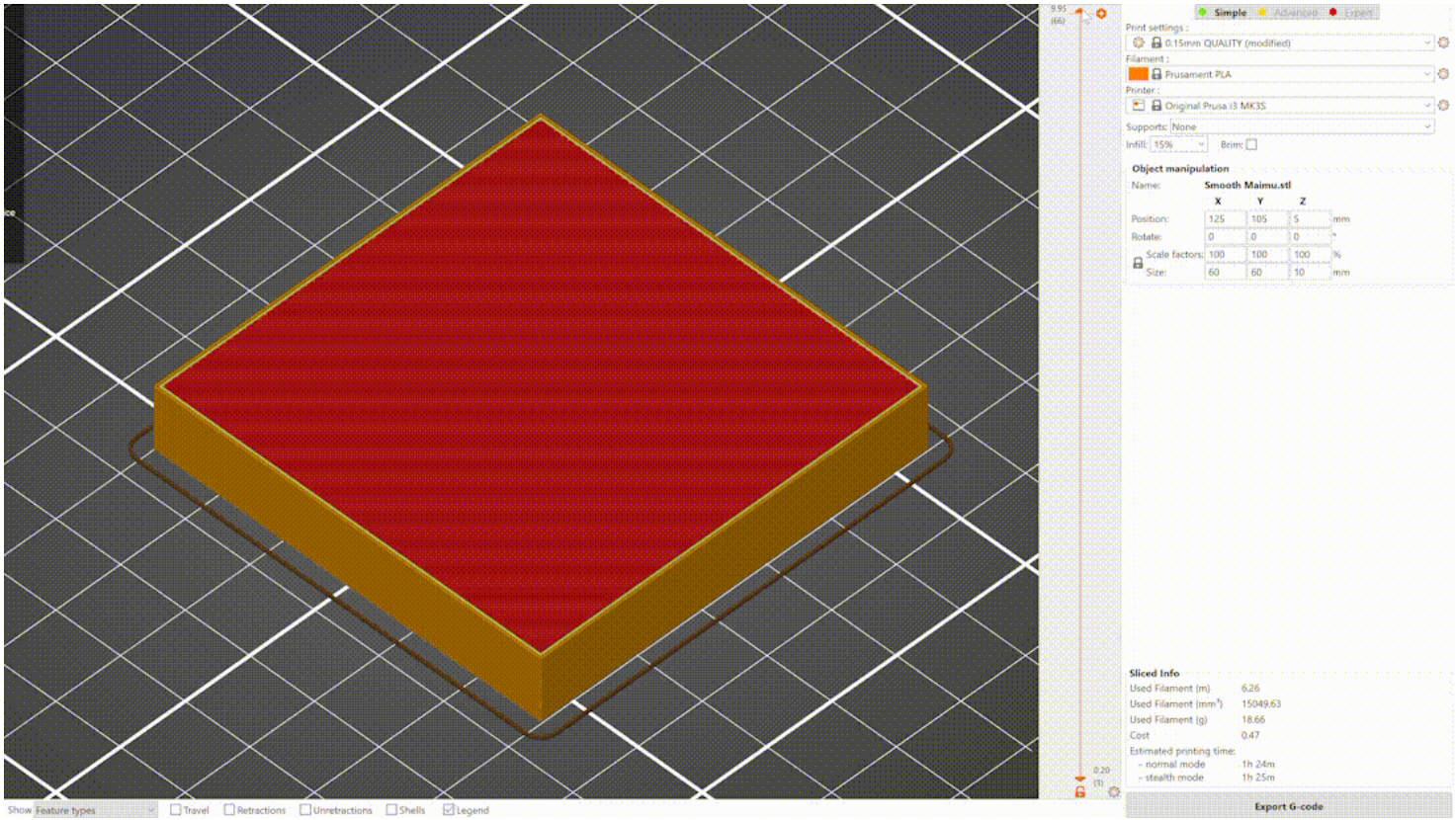
Tips & tricks: Automatic print pause

Colors can be changed between print layers. This method is most often used for highlighting letters or other surface details



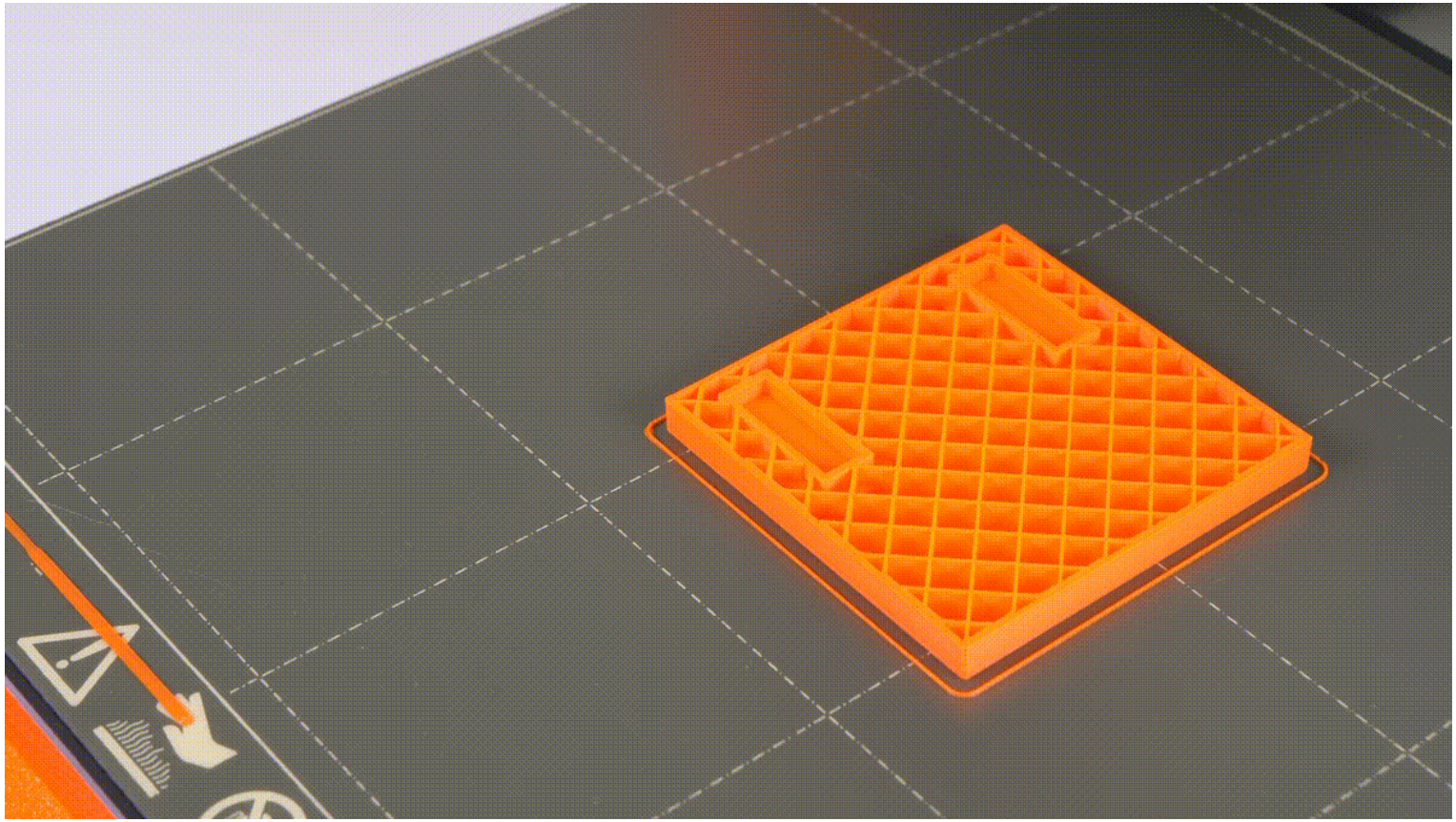
Tips & tricks: Automatic print pause

Note that you can also use the automatic pause for other purposes. Typically, you might want **to insert magnets, screws,** or other fasteners into your print



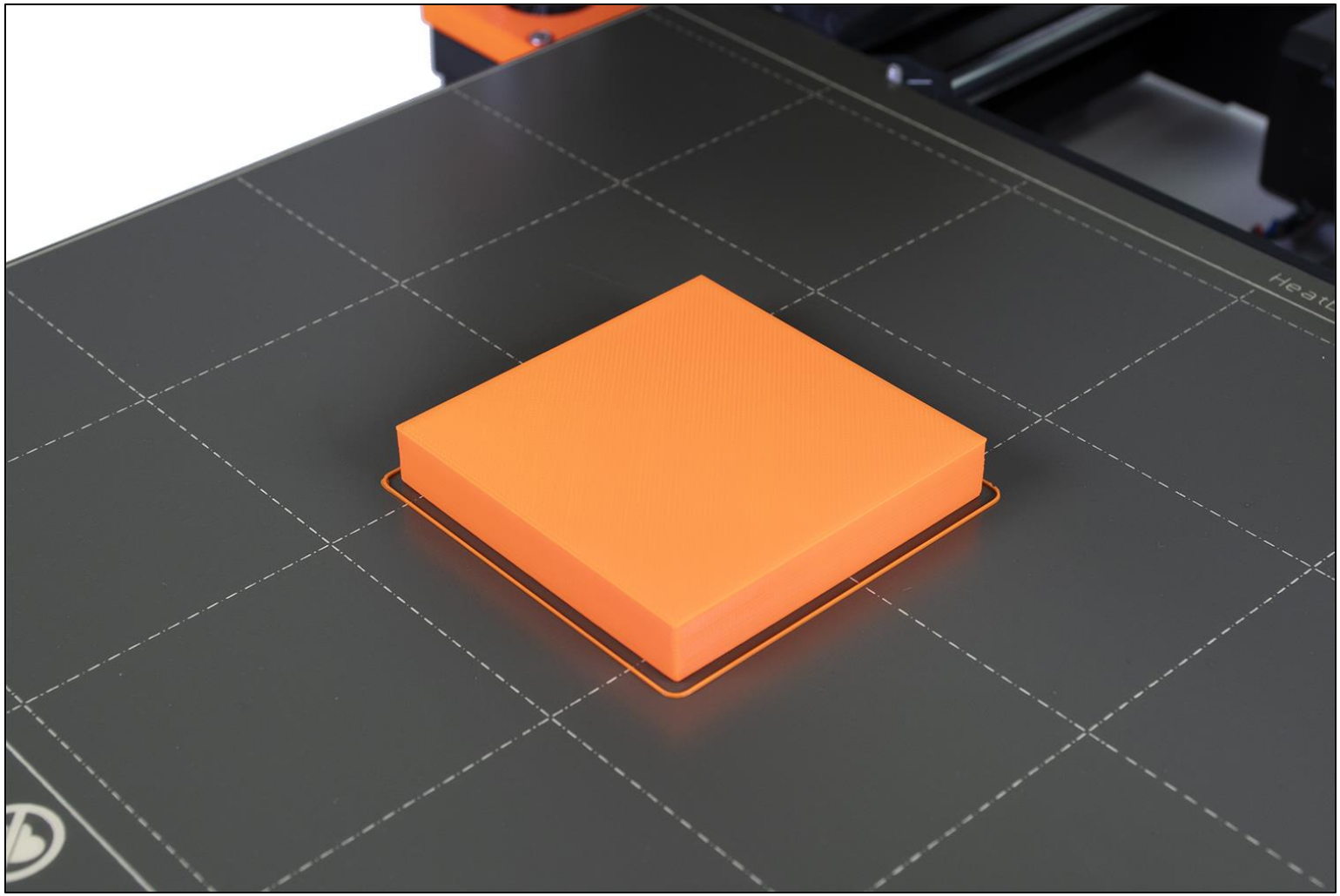
Tips & tricks: Automatic print pause

Insert the fasteners according to your plan (neodymium magnets in this case)…



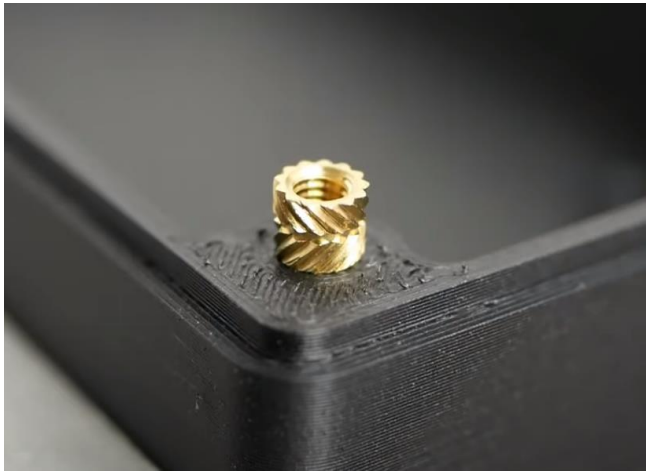
Tips & tricks: Automatic print pause

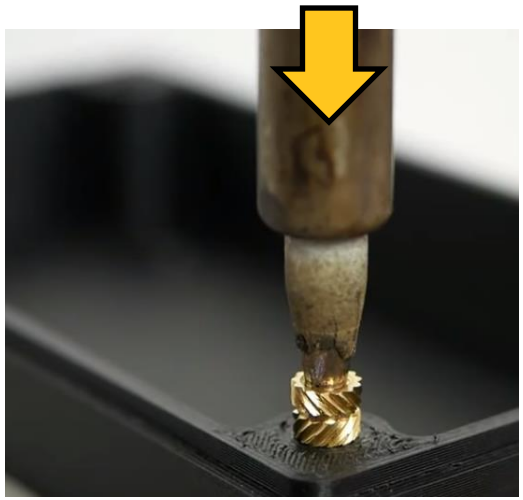
...and let the print continue. You now have **magnets hidden** inside your print. 🙂



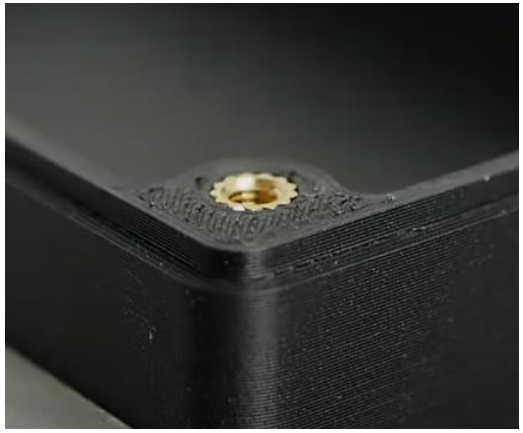
Tips & tricks: Threaded Inserts

Threaded brass inserts can be inserted using a **soldering iron**.



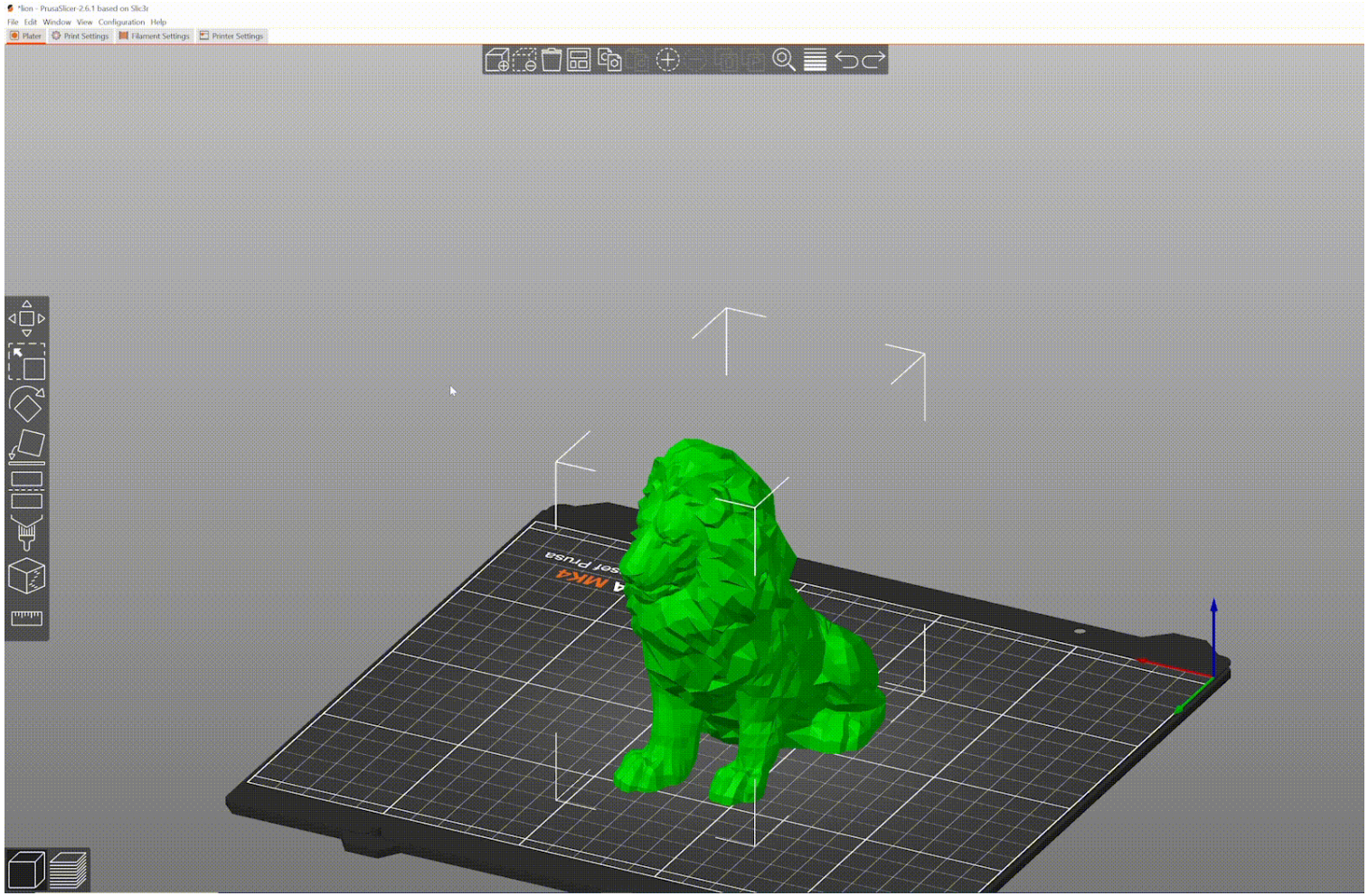


Remember to design **the holes in your model slightly smaller** than the inserts you plan to install. This will account for the plastic that melts when installing the inserts with a soldering iron



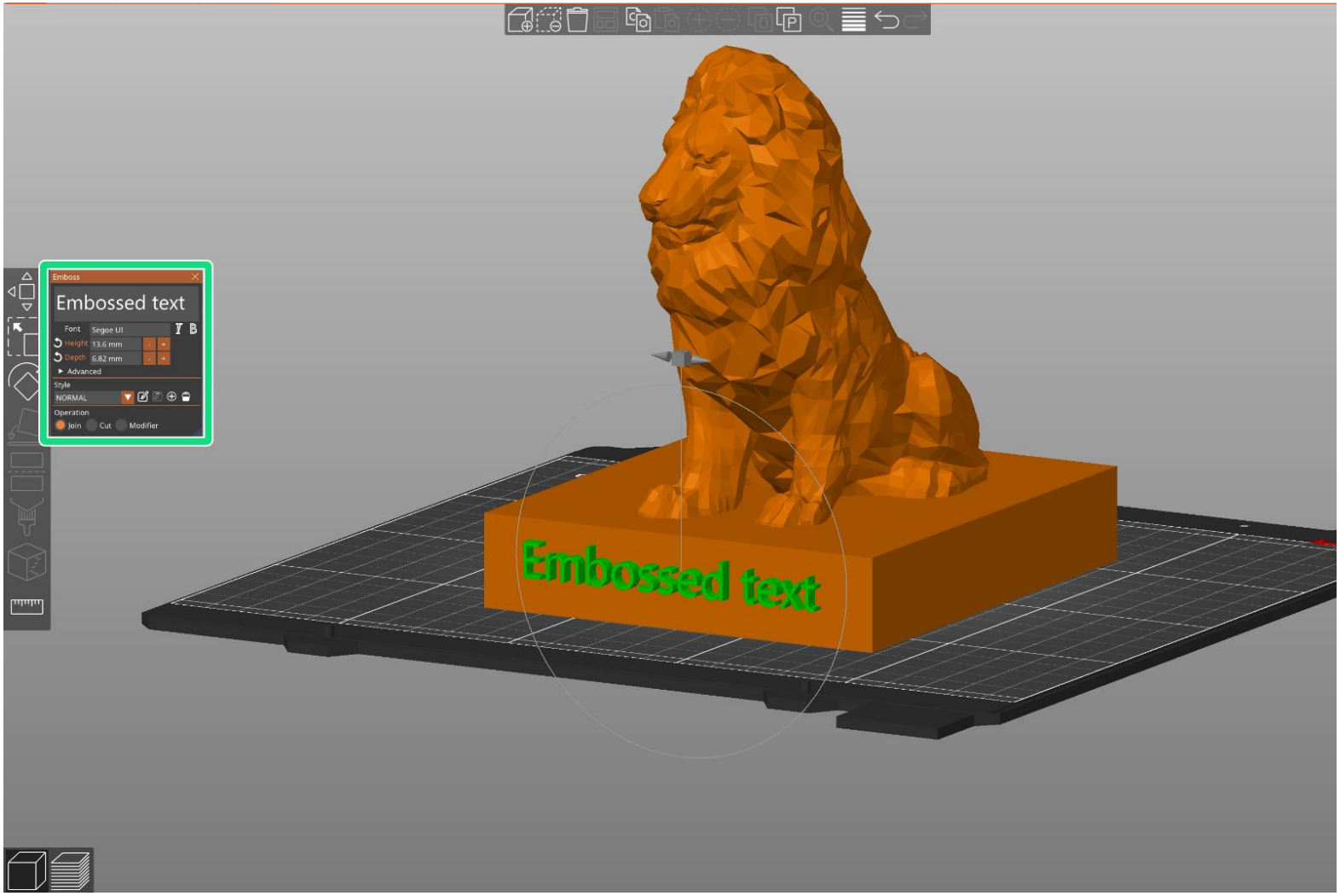
Tips & tricks: Add part

You can add basic geometric shapes as real printable objects via the Add part option.

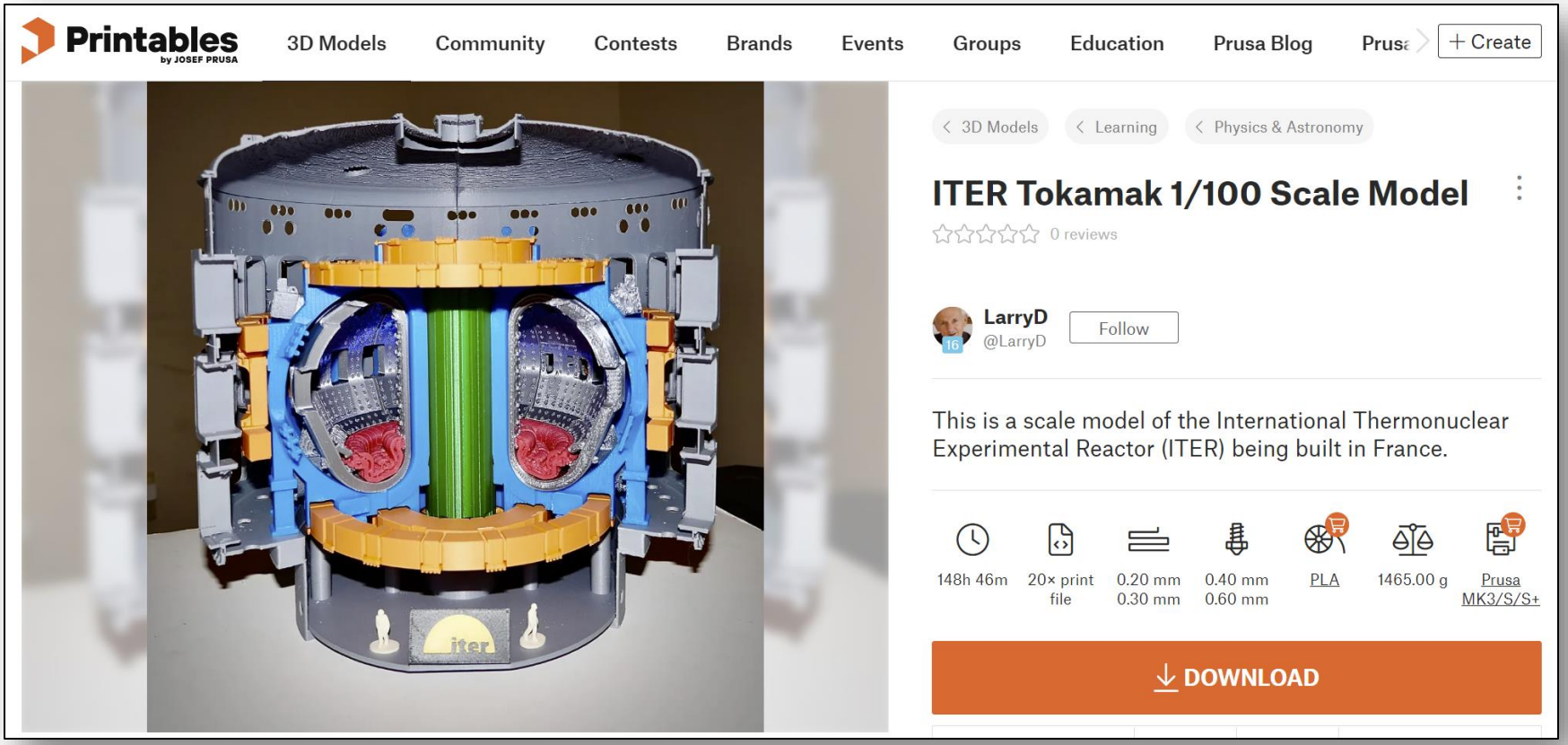


Tips & tricks: Add text

You can also **add text** via one of the Add part options.



printables.com



https://www.printables.com/

Best online resource for downloading free 3D models for printing

For braves only…

ITER NEWSLINE -

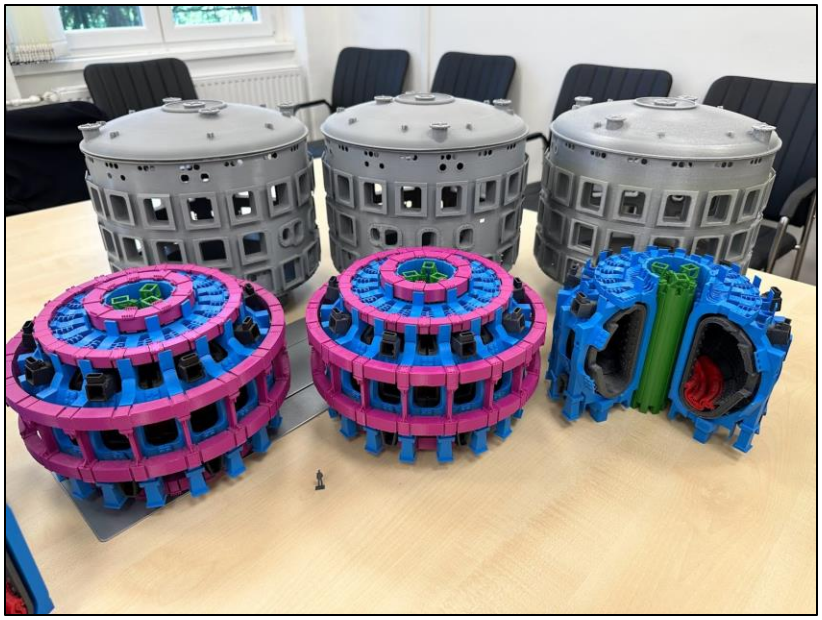
29 JUL, 2020 Print | Read the latest published articles

Education

MAKE YOUR OWN TOKAMAK WITH 3D PRINTING!

Laban Coblentz, Head of Communication

https://www.iter.org/newsline/-/3477



printables.com

Luke Skywalker - Star Wars Pot Plant

42 0 49

Admiral Ackbar - Star Wars Pot Plant

43 0 61

Greedo - Star Wars Pot Plant

45 4 83

Sandcrawler - Star Wars Pot Plant

66 0 117

Chewbacca & Han Solo - Star Wars Pot Plant

100 0 152

Mythosaur Skull (The Mandalorian) - Star Wars...

525 5 593

TIE Interceptor - Star Wars Pot Plant

94 0 139

The Razor Crest - Star Wars Pot Plant

73 0 103

TIE Fighter - Star Wars Pot Plant

116 0 165

C3PO - Star Wars Pot Plant

491 5 723

Endor Bunker - Star Wars Pot Plant

X-Wing Fighter - Star Wars Pot Plant

Tatooine (Luke Skywalker's home) - Star Wars Pot Plant

Millennium Falcon - Star Wars Pot Plant

Darth Vader Pot Plant