

# PRUSA MK4S 3D PRINTER

- TYPE: FDM printer
- BUILD VOLUME: 25 x 21 x 20 cm
- LAYER HEIGHT: up to 0.05 mm |
- MATERIALS: PLA, PETG



# MATERIALS

## PLA

Biopolymer.

- Small amount of warping.
- Easy to print.
- Sensitive to temperature.
- Least expensive.

0.05 CHF / gr

## PETG

Modification of PET..

- A bit of warping.
- Fairly easy to print.
- Not as sensitive to temperature.

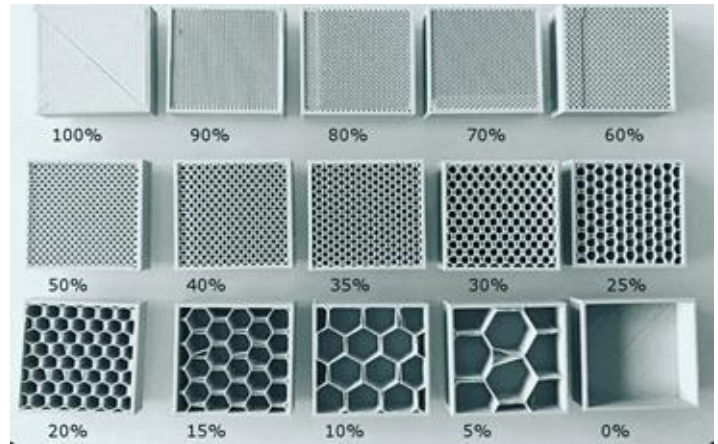
0.06 CHF / gr

# TERMINOLOGY

## INFILL

*Internal structure of a 3D printed part.*

- Parts are not solid.
- Infill %: most of the time 15% is enough.



## SUPPORTS

*Additional structures used to support overhanging geometries during printing.*

- Machine cannot print mid-air.
- Threshold value: 45 degrees.



## BRIM

*3D printed border that extends from the edges of the 3D printed part.*

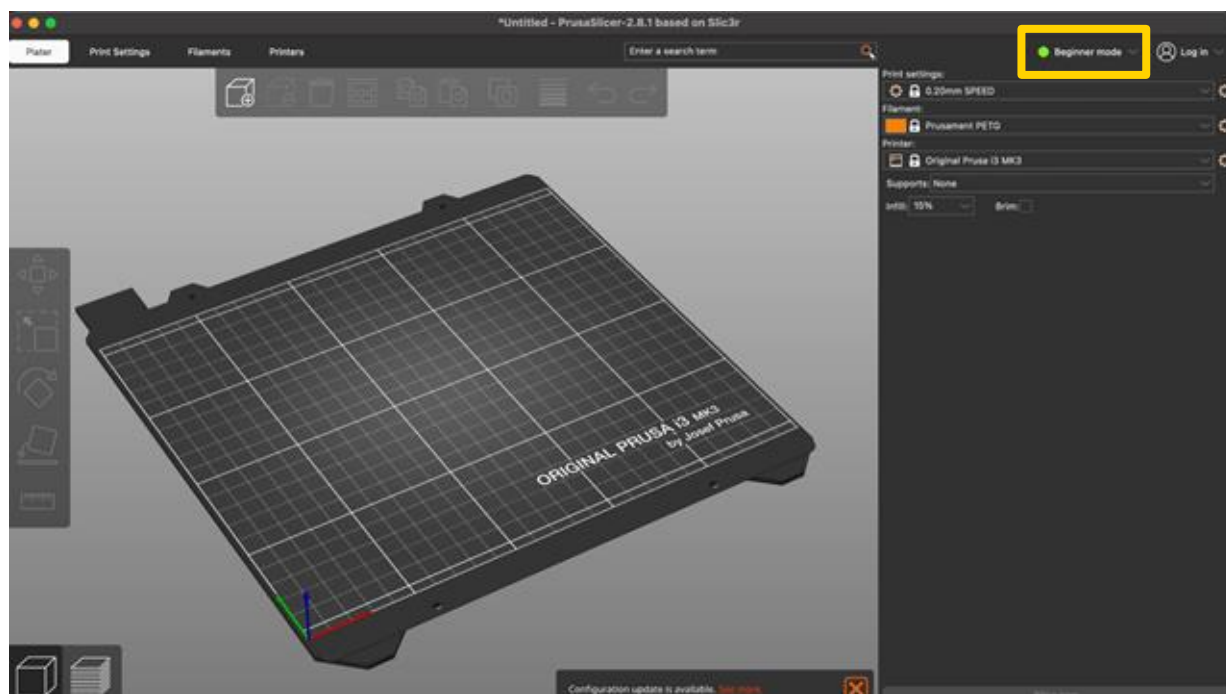
- Needed when: area of contact to the bed is small.
- Prevents lifting of the part from the print bed.





## BASE VIEW

- Open PrusaSlicer and make sure the “beginner mode” is activated.
- Drag and drop your .stl file on the print bed.



## FUNCTIONS

### → 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.

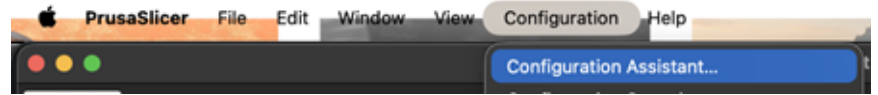




## SET UP

→ If this is your first time using Prusa slicer on our computer or on your laptop, you will have to configure the Wizard.

→ In Prusa Slicer > Configuration > Configuration Assistant



→ Select your printer of choice, using the left selection menu:

- Prusa Research > Original Prusa MK4s HF0.4 mm nozzle (FDM 3D printing)
- Prusa Research > Original Prusa XL 0.4mm nozzle (FDM 3D printing)
- Prusa Research > Original Prusa SL1S Speed (Only FHK #3), Original Prusa SL1

→ Select filaments:

- Filaments > PLA > Prusament PLA
- Filaments > PETG > Prusament PETG

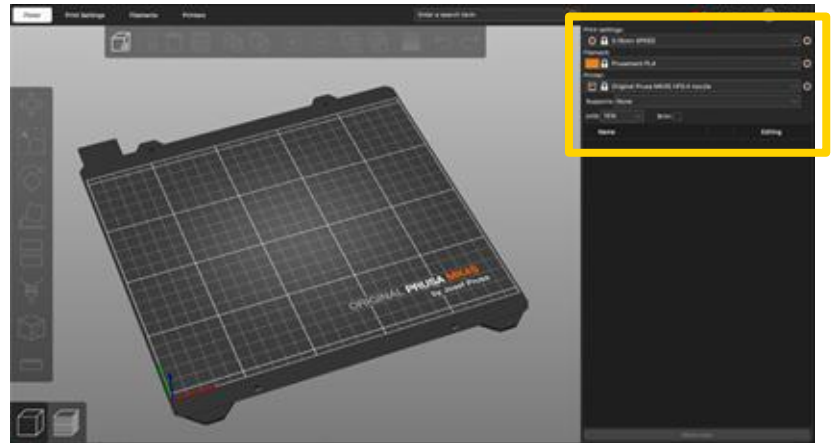
→ Select resins:

- SLA Materials > all the Prusament Resin Model and Prusament Resin

Though





Edit settings. You can find them on the right side of the software interface.



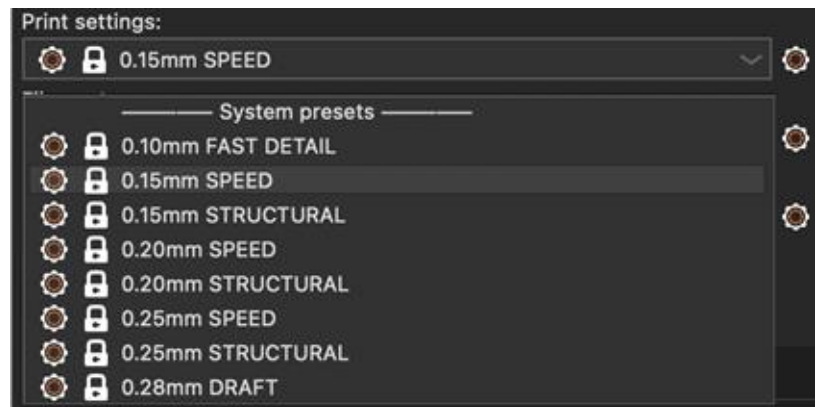
## PRINTER

Printer:

  Original Prusa MK4S HF0.4 nozzle

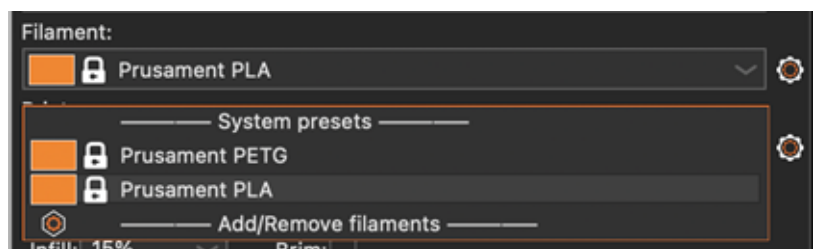
## LAYER HEIGHT

- Select the layer height to decide the print quality.
- Smaller layer heights take longer to print, but will result in better quality.



## MATERIAL

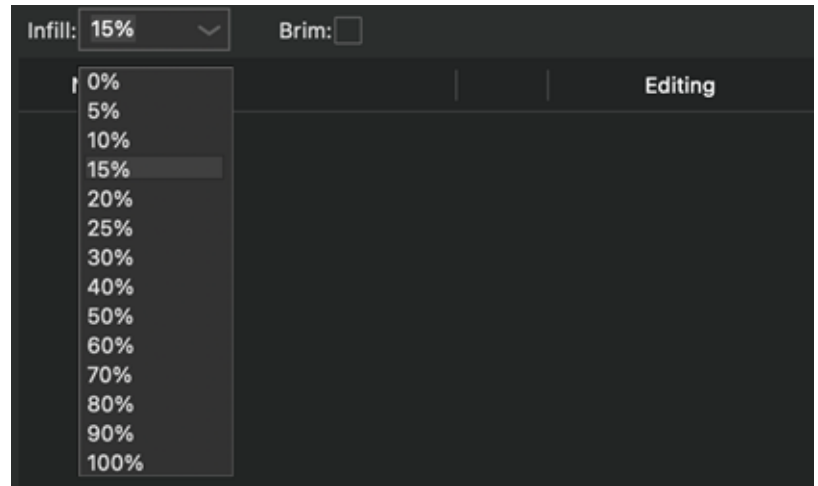
- Select the same material you will load in the machine.
- Please only use Prusament PLA or PETG.





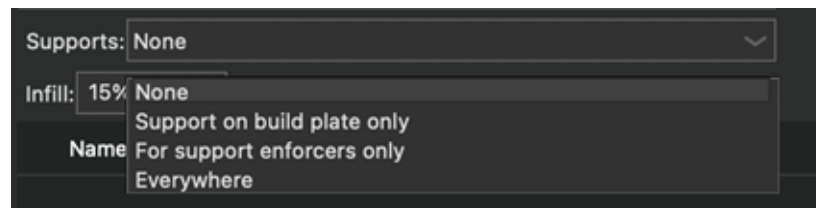
## INFILL

- Set your infill percentage.
- More infill means more weight: longer print times and higher costs.
- For most parts 15% is fine.



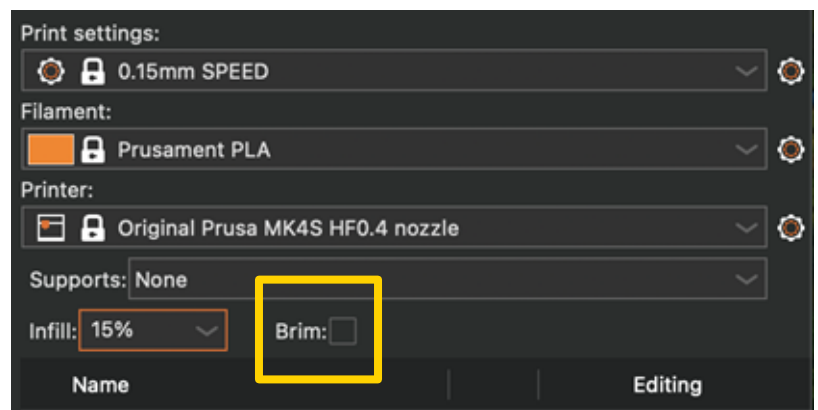
## SUPPORTS

- Supports are disabled by default. **Always double check that they are enabled, if you have steep overhangs.**
- If you are unsure, add supports “Everywhere”.



## BRIM

- Add a brim if your model has small base (like tall, thin parts).
- Add a brim if you want to reduce corner lifting or warping.



## SLICE

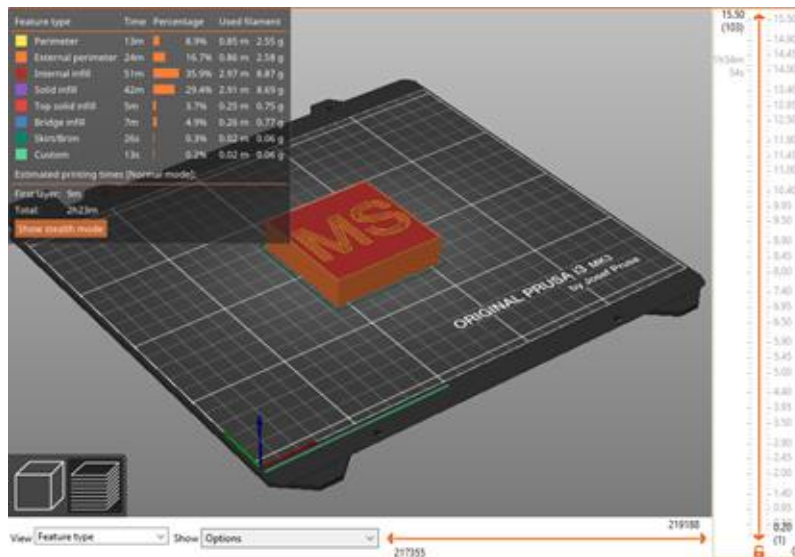
- Use the Slice now button to prepare your part for printing.





## PREVIEW

- Use the slider on the right to go through the different layers.
- Check that at no point something is appearing out of nothing.



## PRINTING INFO

- On the lower right corner you can see some information about your print.
- **IMPORTANT:** Make sure that there is enough filament left for your print. You can see the weight of your print in the first line.

### Sliced Info

Used Filament (g) (including spool)	24.33 (254.33)
Used Filament (m)	8.16
Used Filament (mm <sup>3</sup> )	19619.24
Cost	0.68
Estimated printing time:	
- normal mode	2h23m
- stealth mode	2h24m

## EXPORT

- Use the Export G-code button to export the code.
- Save it on the printer's USB stick.

Export G-code

# AT THE MACHINE

## STEP 1

Turn the machine on, using one of the terminals.



## STEP 2

Make sure the correct filament is loaded and that it is enough for your print.

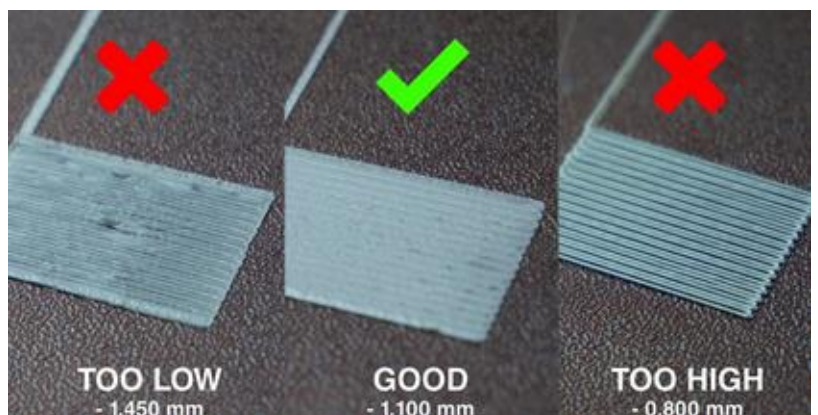
## STEP 3

- Press “Print” from the main menu.
- Select your part (usually the first in the list).



## STEP 4

**Important:** stay until the first layer is done. If you see something going wrong, please stop the print and notify a Makerspace Instructor or staff if present. See image for reference →



# CHANGING THE FILAMENT

## STEP 1

→ Press “Filament” from the main menu.



## STEP 2

→ Press “Change Filament”.



## STEP 3

→ Wait for the machine to reach the correct temperature.  
→ When the filament is successfully released, press “Yes”.



# CHANGING THE FILAMENT

## STEP 4

→ Select the type of the filament you will load next.



## STEP 5

→ Insert the new filament into the extruder, then press "Continue."



## STEP 6

→ If the loading was successful, press "Yes."  
→ If not, select "Purge more" and repeat until the filament loads correctly.

