PRUSA | PrusaSlicer Setup

Here’s some tips/info on its initial setup:

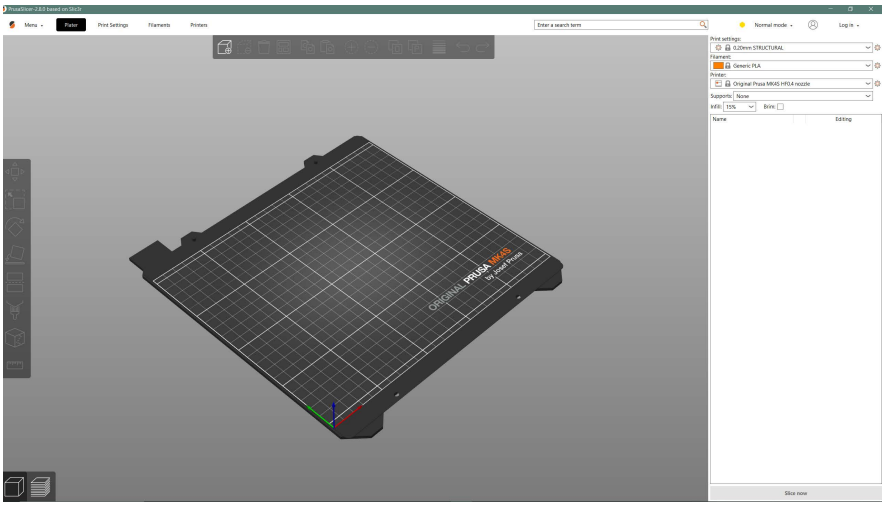


Figure 1 PrusaSlicer's main interface.

When setting up PrusaSlicer:

i. **Select "Prusa FFF" for Configuration sources** and then **“Original Prusa MK4S”** from the list of printers, and make sure **“HF0.4mm nozzle"** is selected. Deselect all other printers

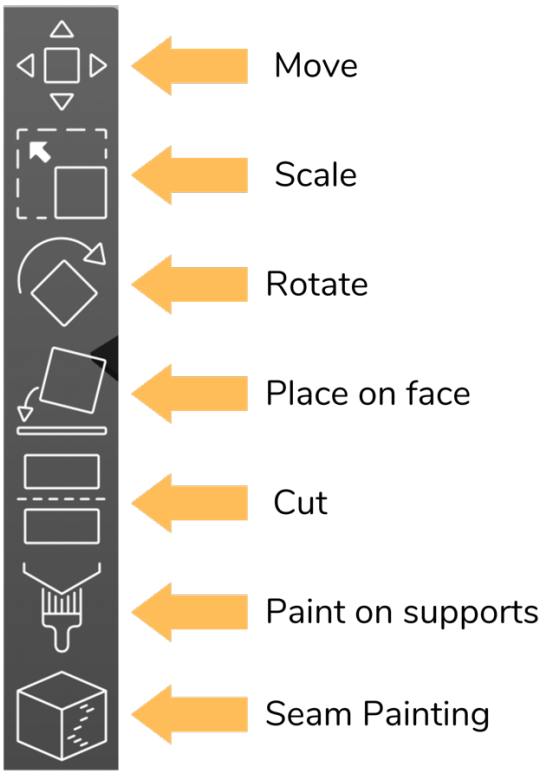
PRUSA | PrusaSlicer Tools Overview

Before we prep a print, here's a quick guide to all the functions available in PrusaSlicer's main "Plater" area:

The bar on the top contains the following tools:

1. **Add** – to add a file to the plate
2. **Delete** – to delete a file from the plate
3. **Delete all** – to delete all files from the plate
4. **Arrange** – arranges all files/objects neatly on the plate
5. **Copy**
6. **Paste**
7. **Add instance** – multiplies the selected object once
8. **Remove instance** – deletes any copy of an object
9. **Split into objects** – if multiple objects are in a file, it will separate them for easier moving
10. **Split into parts** – splits an object into parts, if available
11. **Search** – search PrusaSlicer for specific settings
12. **Variable layer height** – allows different sections of a file to print with different infill percentages
13. **Undo** – undo any changes made to a file
14. **Redo** – redo any changes made to a file

The sidebar contains the following functions:



Move – Allows movement of an object

Scale – changes the size of the object

Rotate – allows 360º movement of the object

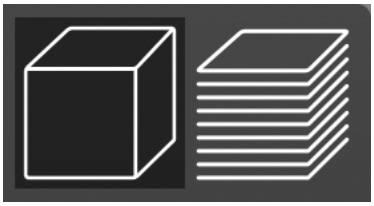
Place on face – sets object on its flattest edge possible

Cut – slices an object into 2 parts

Paint on supports – adds custom supports where needed

Seam painting – customizes where the layer seam is located

The two blocks at the bottom of the screen do the following:



1. 3D editor view – where the print file can be manipulated/moved/rotated/etc.
2. Preview – shows what the print file will look like when it's actually printed; usually includes all layers/supports/infill.

PRUSA | Preparing a Print

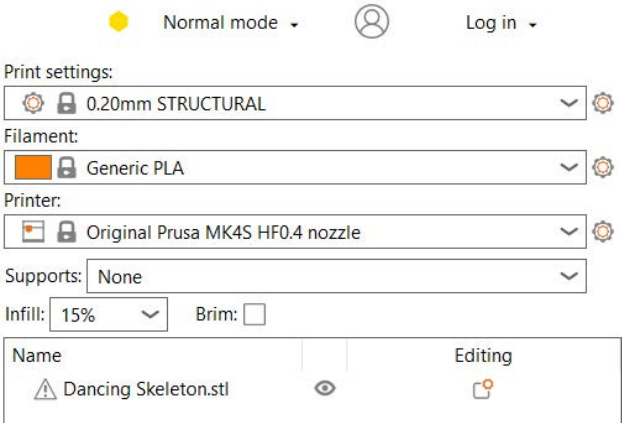
Huzzah! You're ready to prep a print!

To get started, **make sure you're in the "Plater" section of PrusaSlicer**. Note: please only use **.stl** files.

From there:

1. Click the **"Add"** button at the top of the workspace and select your .stl file. The print file will generate in the center of the plate.

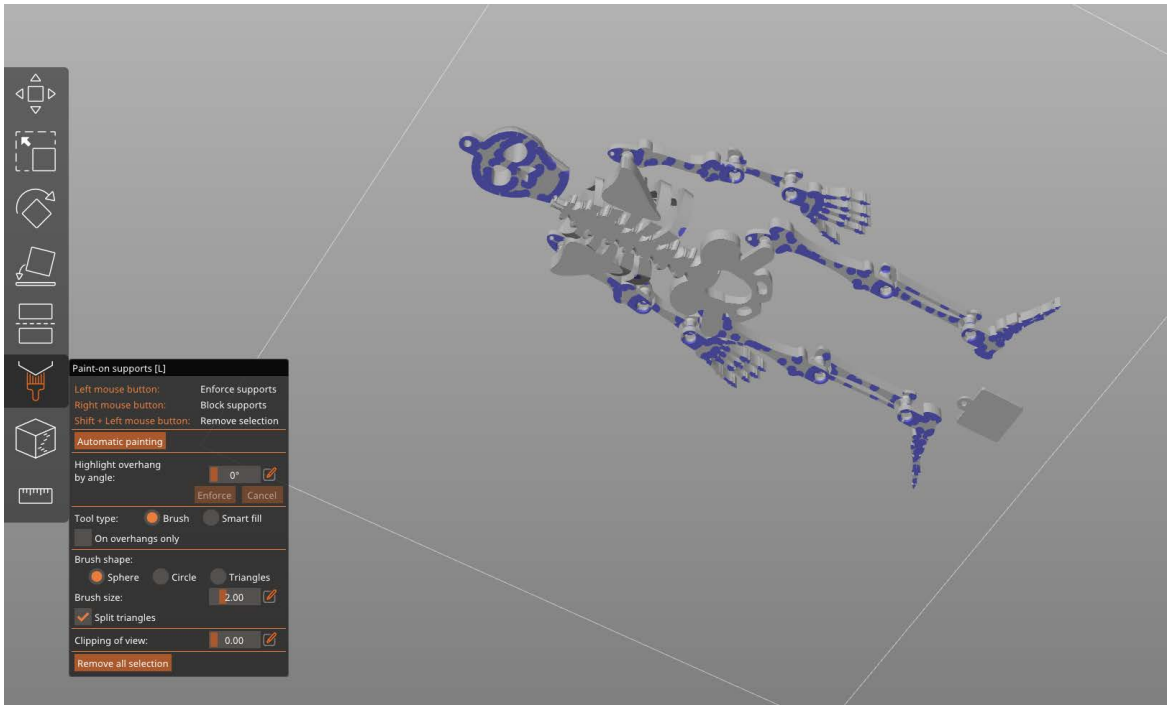
2. **Move/rotate/resize/multiply the print as desired** with the toolbar on the left side of the screen.
3. Once the file is set as desired, **check the right side of the screen to adjust the following settings:**
4.



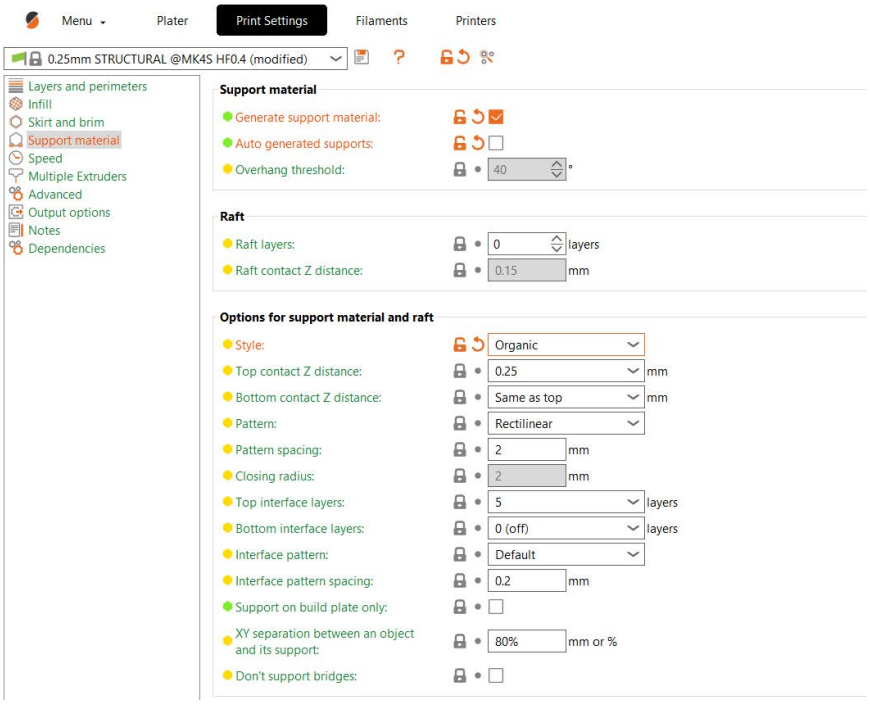
   a. **Print Settings** – Select your layer height profile. The smaller the number, the more detailed your print will be. However, smaller layer heights will use more material and take longer! .20 layer height is typical.
   b. **Filament –** Select the color filament type
   c. **Infill** – How dense you want your print to be. 10-20% is typical.
   d. **Brim** – this is an optional support setting that adds an extra set of lines printed on the first layer to help stabilize an object print should there be tall/thin structures.
   e. **Supports** – choose which type of supports you think will best work with the print from the drop-down list.
   f. **File repair –** If you see an exclamation point next to your file name, right click the exclamation point to run repairs.

More On Supports (E)

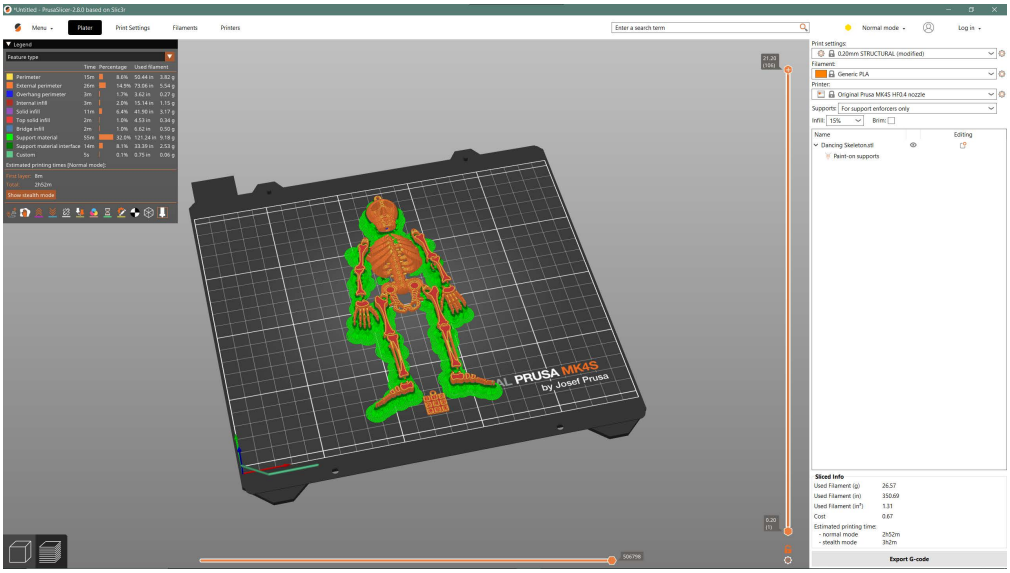
To use less material and to generate supports that are easier to remove, consider using "**for support enforcers only.**" You will generate these supports by using the paint-on supports feature. You can do this manually, but for the most reliable results, use **Automatic Painting**



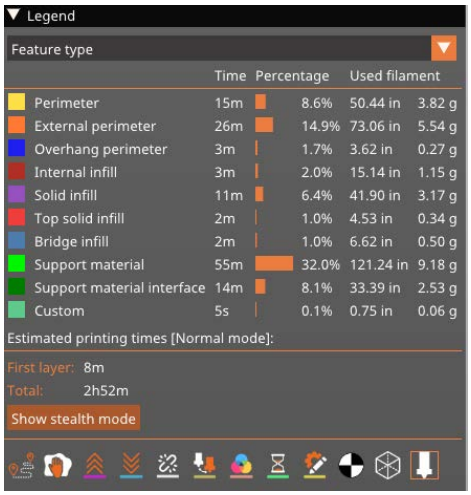
To generate organic supports, go to the **Print Settings** panel in the upper left, and under **Support Material,** change the Style to **Organic.**



5. Once all settings are chosen, click **“Slice now”**. The screen will change to Preview mode, showing you how all the supports and layers will print. **If changes are needed, click the “3D editor view” button** at the bottom of the screen to adjust the print.



6. **If the preview looks good, click “Export G-code”** and save the .gcode file to your computer.



Take note of how much filament is required for your print in grams, how long the full print will take, and how long you must stay for your first layer.

Add the file to the Prusa’s USB stick, and you’re set to print.

PRUSA | Starting a Print & Print Removal

To start a print:

1. Make sure your .gcode file is on an SD card and inserted into the printer.
2. On the printer, press the knob and scroll to “Print from SD”. Press to select.
3. Scroll to your .gcode file and press to start the print.

IMPORTANT! Make sure to monitor the print’s first layer for success. If there’s any peeling/filament not sticking to the plate, immediately stop the print. Clean off the plate and apply a LOT of fresh glue. Make sure the nozzle is clean, and brush it with a toothbrush if it isn’t. Restart the print.

Print Removal

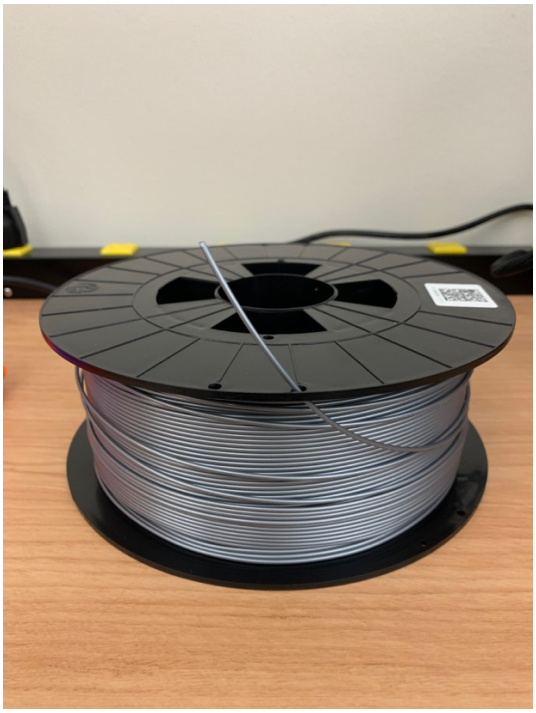
You do not have to wait for the plate to cool when a print is complete, but it’s recommended to do so for safety. Allow the plate to cool for a minute or two before touching. To remove a completed print:

1. Lift the magnetic plate off the printer.
2. If needed, gently bend the plate at its corners to loosen the print at its base.
3. Pull the print from the plate.
4. If the print is still stuck to the plate, you may need a tool like one of the blue handled knives or spatula scrapers to pry the print off the plate.
5. Place the magnetic plate back on the printer, making sure to line it up with the rest of the base. The two screws on the back of the plate are great alignment guides.
6. Add a new layer of glue before starting any new prints.

PRUSA | Filament/Plate Information

Our Prusa printers use 1.75mm PLA plastic filament for printing.

When changing/storing an opened roll, please be sure to keep the end of the roll tucked in one of the holes on the spool to prevent any tangling. **Tangled rolls can lead to failed prints and a damaged extruder/broken printer, so it is very important to store filament properly.**

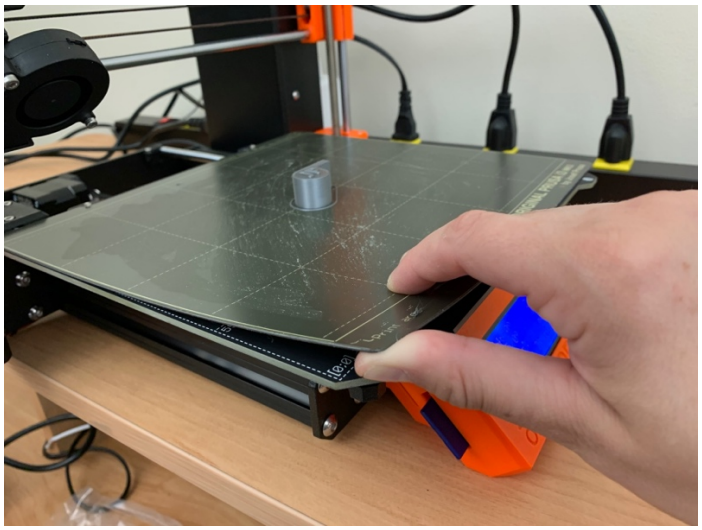


(continued on next page)

For filament to properly stick to the printer plate, **prep the plate with glue prior to starting a print.** This will prevent the print from detaching or popping off the plate while printing, which can cause filament to overflow into the nozzle and surrounding wires on the extruder (which is incredibly difficult to fix). Glue sticks like Elmer's craft glue work best.

Printer Plate

The Prusa printer comes with a magnetic PEI sheet. It is fully removeable.

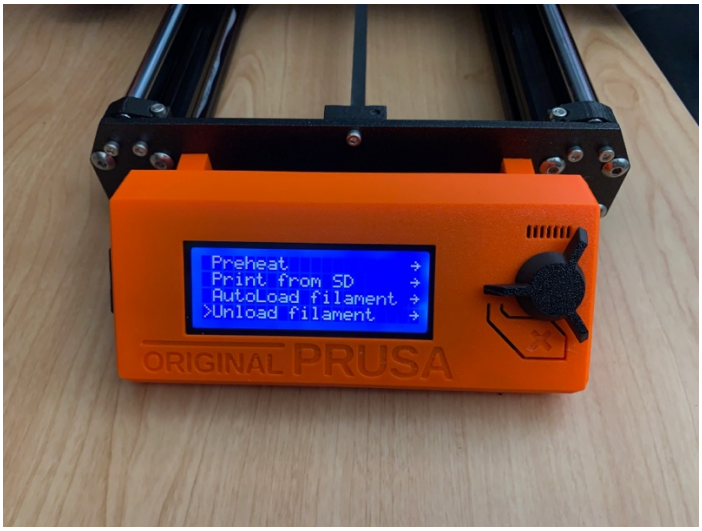


The printer plate can be cleaned with soap and water if there is too much glue buildup. Make sure to dry the plate completely before placing it back onto the printer. Do not put the plate glue side down on the printer.

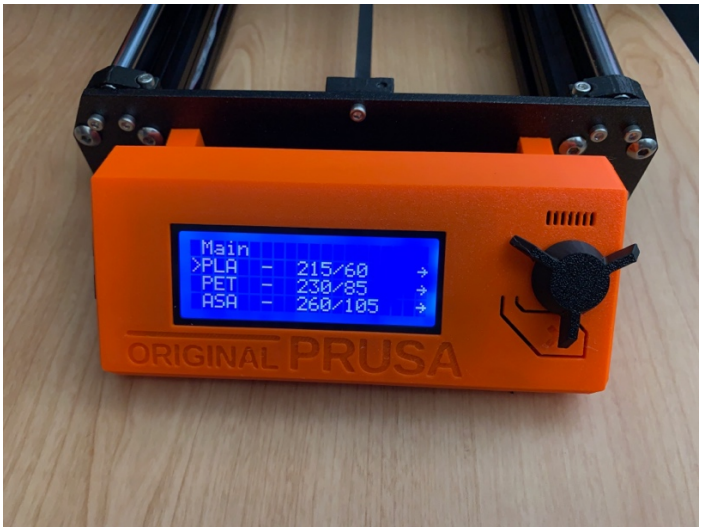
Changing Filament

To change from one roll of filament to another:

1. Press the black knob and scroll to “Unload Filament”. Click to select.

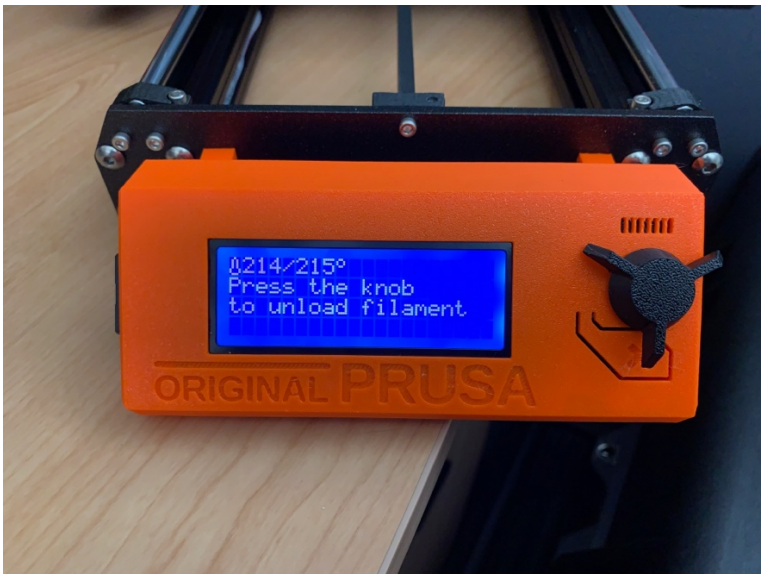


2. Select PLA.



3. Wait for the printer to heat up fully. The plate will also be very hot, so use caution.

4. When prompted, press the black knob to unload the filament. It will push the old filament out from the top of the extruder.

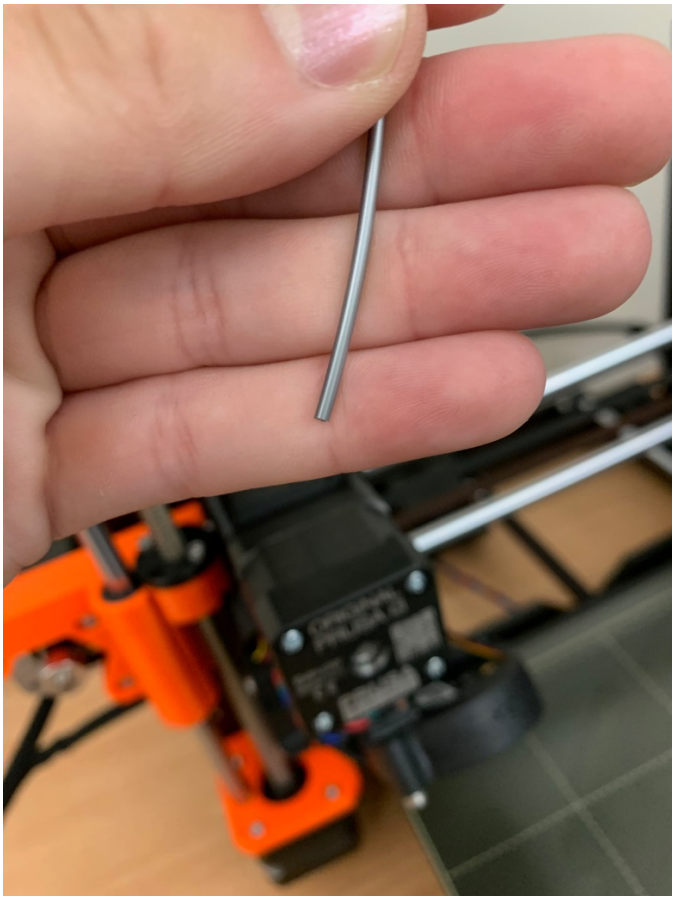


5. Pull the filament out gently.

(continued on next page)

To add new filament:

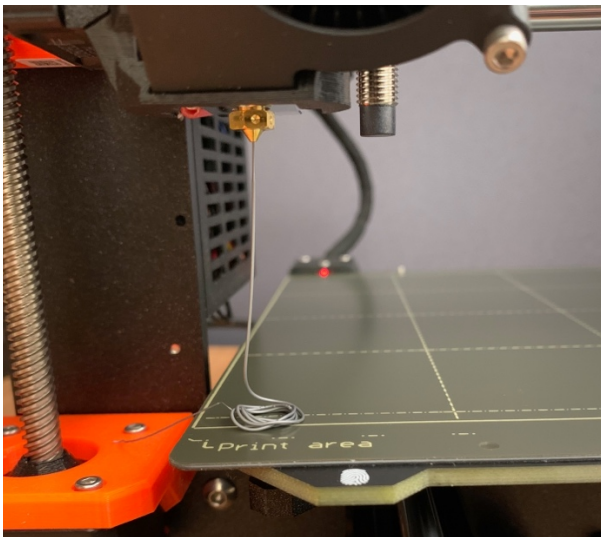
1. Prepare the new roll of filament by cutting off about 1” from the start of the roll. This removes any stringy/clumped bits that can cause clogs.



2. In the printer menu, select “Autoload Filament”.

3. Gently feed new filament into the top of the extruder. You should feel the extruder pulling down the filament. Do not push filament into the extruder.

4. Allow filament to extrude.

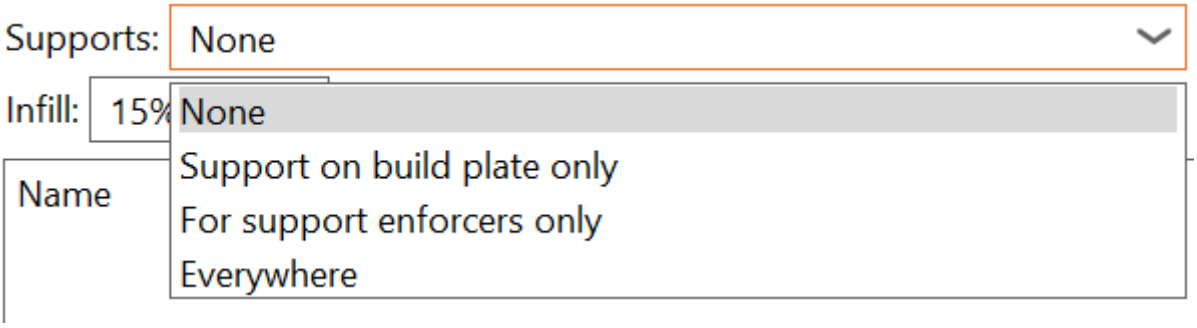


5. When prompted, select “Yes” to confirm filament change was successful.

That’s it!

3D Print Supports

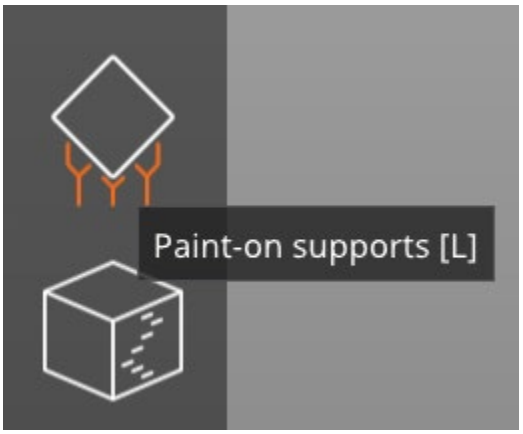
There are 4 different support options listed in Prusa Slicer:



None will print without any supports.

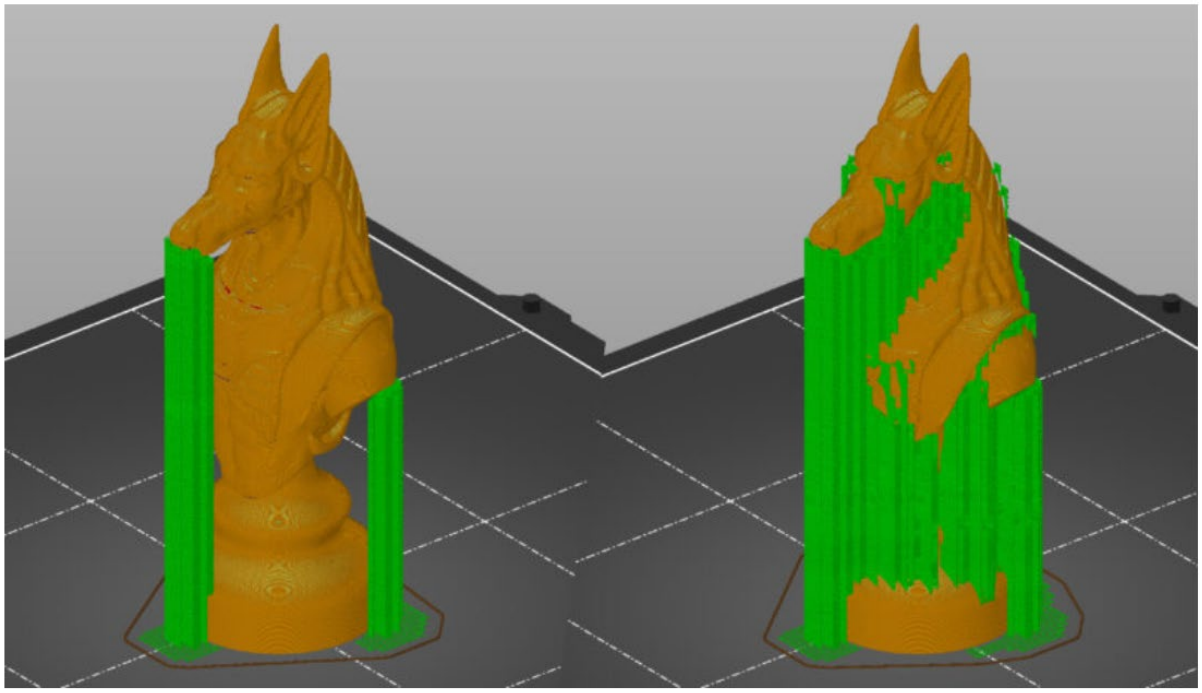
Support on build plate only will grow from the build plate to the object, but will not print object to object supports.

For support enforcers only is designed to only work with **paint on supports**, which can be found in the left toolbar.



If you use this setting and do not paint on supports, the slicer will not generate this error to warn you that the print needs supports:

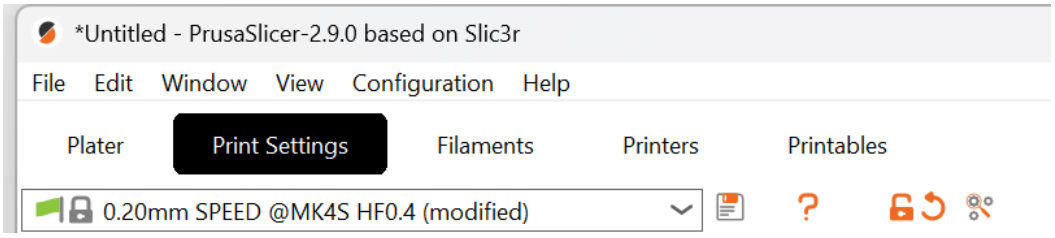
Everywhere will generate supports that touch the build plate to the object as well as object to object supports.

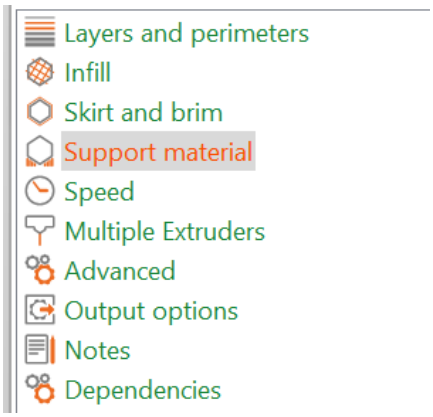


Supports from build plate only (left), supports everywhere (right)

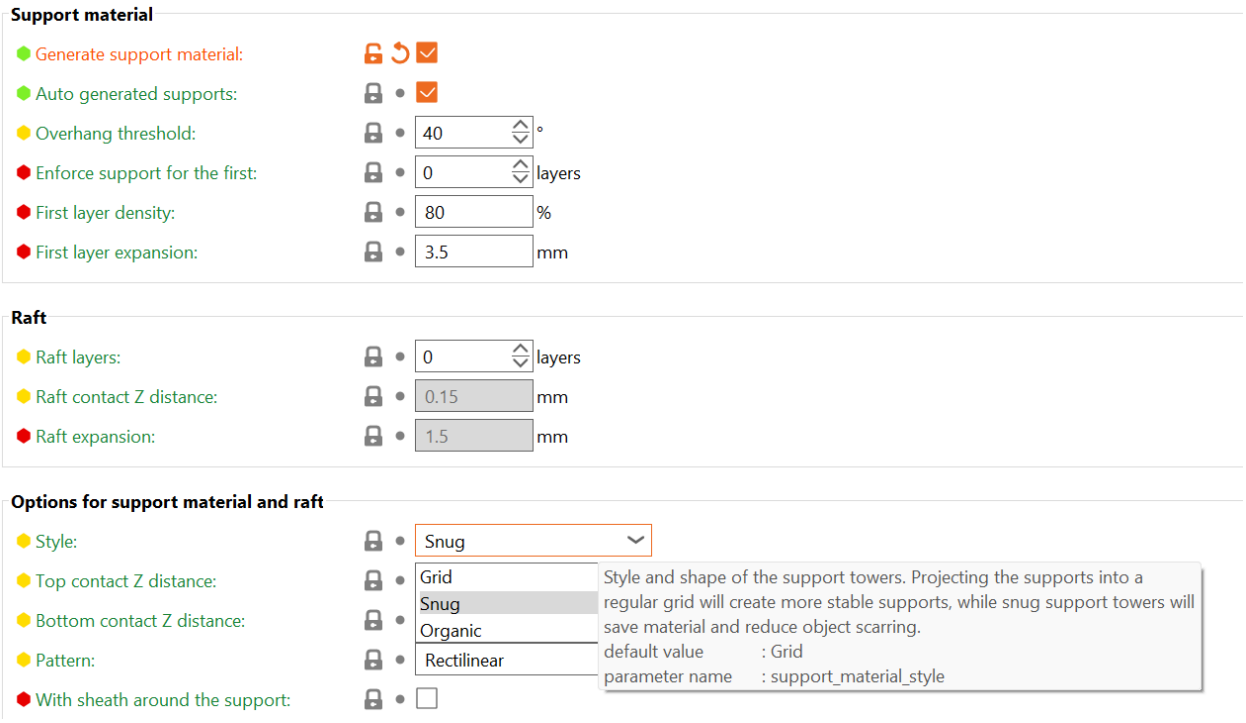
Style of Supports

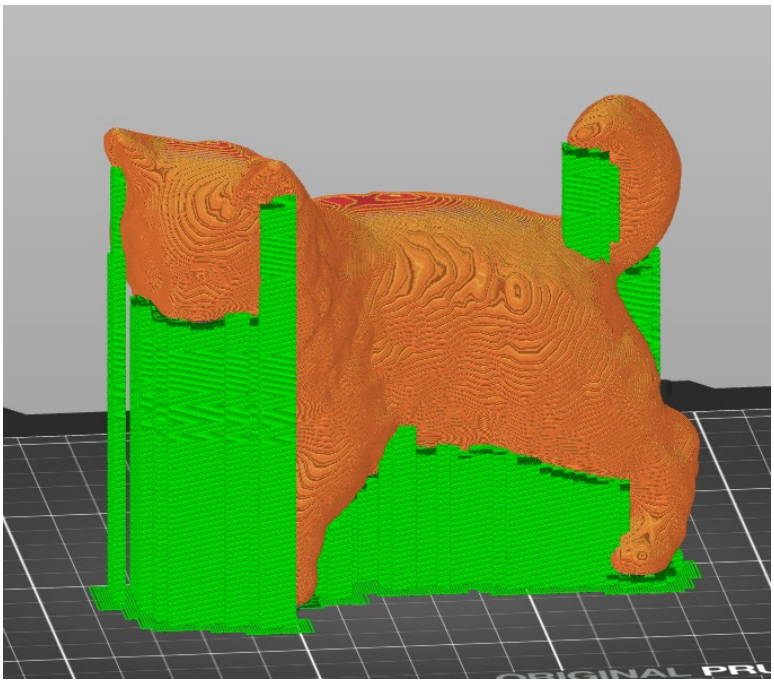
There are 3 styles of supports that Prusaslicer can generate: snug, grid, and organic. Our preferred supports are organic, which look like trees growing into the object. These are often a little easier to remove. To access support settings, choose Print Settings at the top toolbar, and then Support Material.



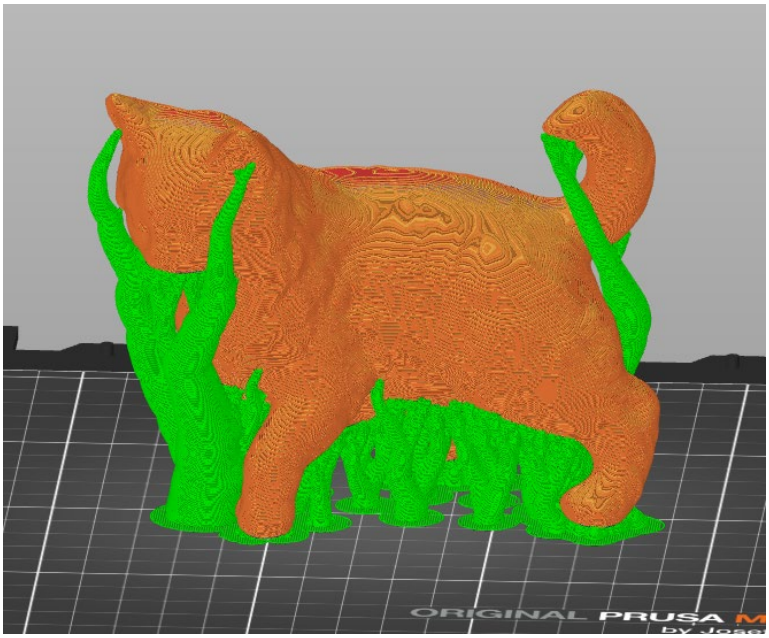


From here, make sure “Generate support material” is selected, and scroll down to “Options for support material and raft” to select style of support.





Example of an object with grid supports



Example of an object with organic supports