

Simon Carlsson

Print not sticking to the bed (WARPING)

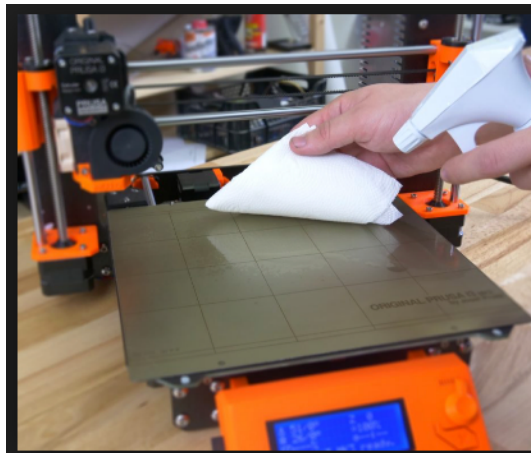


Sometimes the print may not adhere to the **PEI flexible sheet** on the Prusa I3 MK3. This can be due to many things but the most common ones are:

- **Bed not cleaned properly**

Yes, something as simple as forgetting to clean the bed can have a huge impact on the prints ability to adhere. Even the slightest bit of dust or oil can cause poor bed adhesion.

The **solution** is very simple, just wipe the platform thoroughly with **IPA** (isopropanol alcohol) before you start a print.



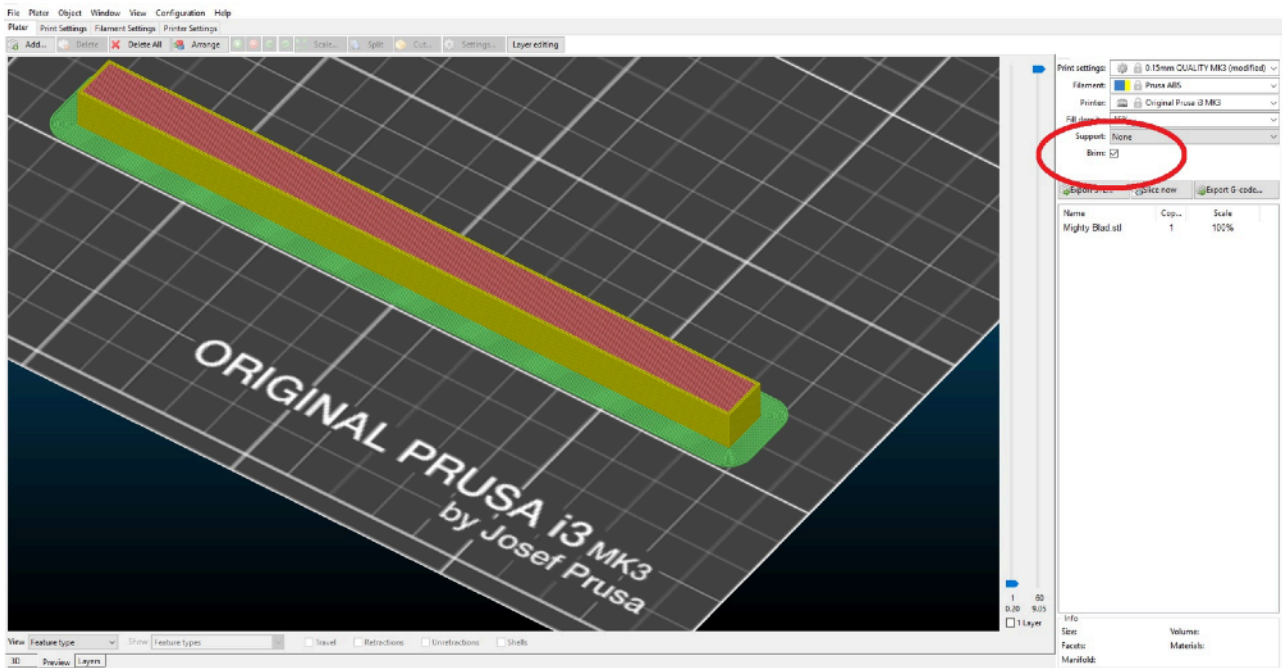
If it still doesn't work you can try cleaning the bed with **Acetone**. This is abrasive to the bed and should only be done in occasions where IPA does not work.

WARNING!

KEEP THE ACETONE AWAY FROM ANY PLASTICS. IT WILL MELT THEM!

- **Contact area is too small**

If your model has very little contact area with the bed it will also have a very high chance of lifting from the bed. The solution to this is to use **BRIM** in the slicing software



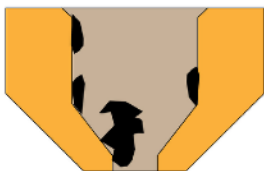
- **Wrong material**

Some materials have a tougher time sticking to the bed than others, this is especially true for ABS. Printing such materials is more suited for the Zortrax printers, which have special features to counteract this.

Filament not extruding properly (CLOG)

Arguably, the most annoying thing in 3D printing is what is called a **clog**. This happens when something is blocking the filament from extruding. This can be anything really, dust, debris or even filament that has come stuck in one way or another. The picture below to the left illustrates the cross-section of the nozzle with a **partial clog** and a **full clog**. The partial one would still work but would likely produce poor results and uneven surfaces (like the print to the right illustrates).

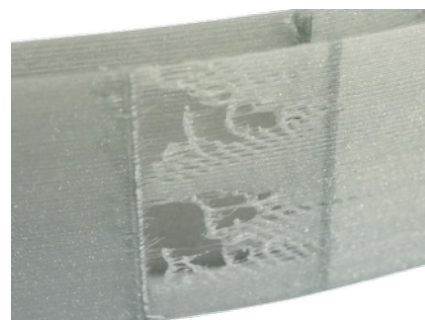
The full clog would **not print at all** and the printer would just move around without extruding any filament. Sometimes, a "clicking" noise may be heard from the extruder as it tries to push filament through the clogged nozzle.



Partial clog



Full clog



Thankfully, this is usually pretty easy to fix.

1. **Unload** the filament from the printer. Use the screen on the printer and select "Unload filament" and select the material that is currently loaded in the printer

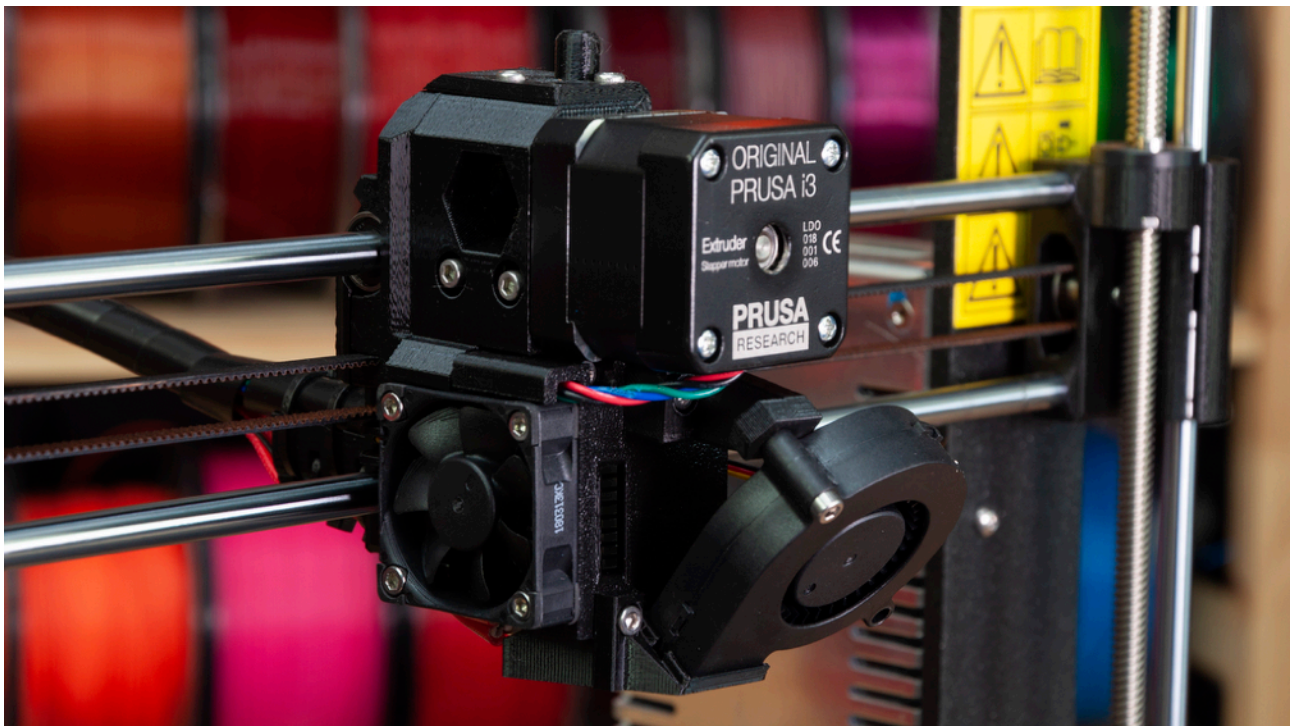
2. While the printhead is still hot, grab the 0.35 mm needle from the drawer underneath the printer and push it through the nozzle (**Careful the nozzle is very hot**). Sometimes it can help to use a pair of small pliers to hold the needle. Once the 0.35 mm needle moves freely, change to the 0.4 mm needle and repeat the process.
3. It should now be cleaned and ready to print again.



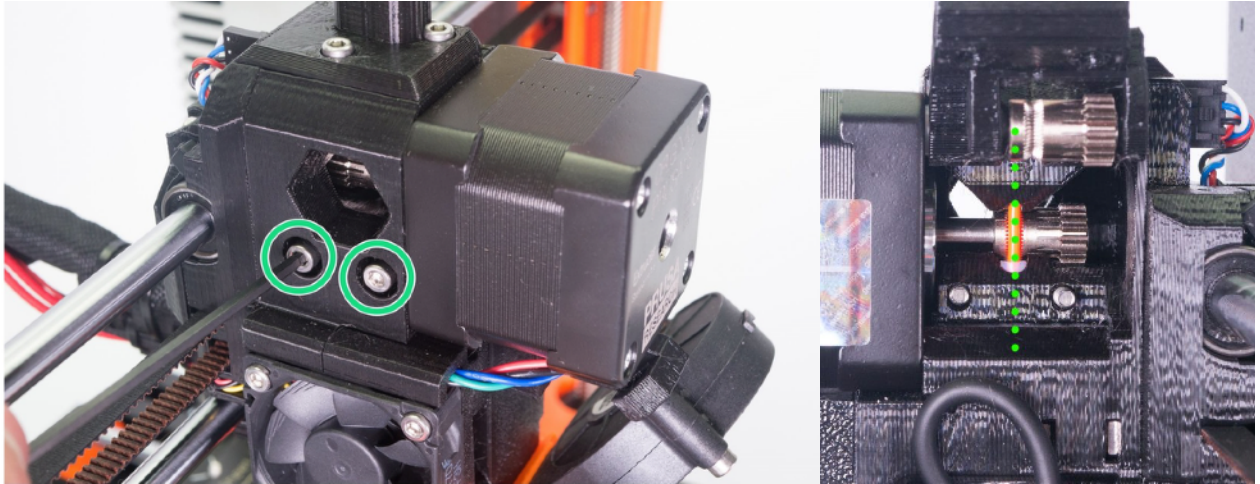
Filament won't unload properly

Sometimes, using the "Unload filament" function in the Prusa will just not work, the filament gets stuck and won't pull out. Usually, this is because the filament got stuck in the extruder gears due to some kink or blob on the end. When this happens there are two ways of solving it:

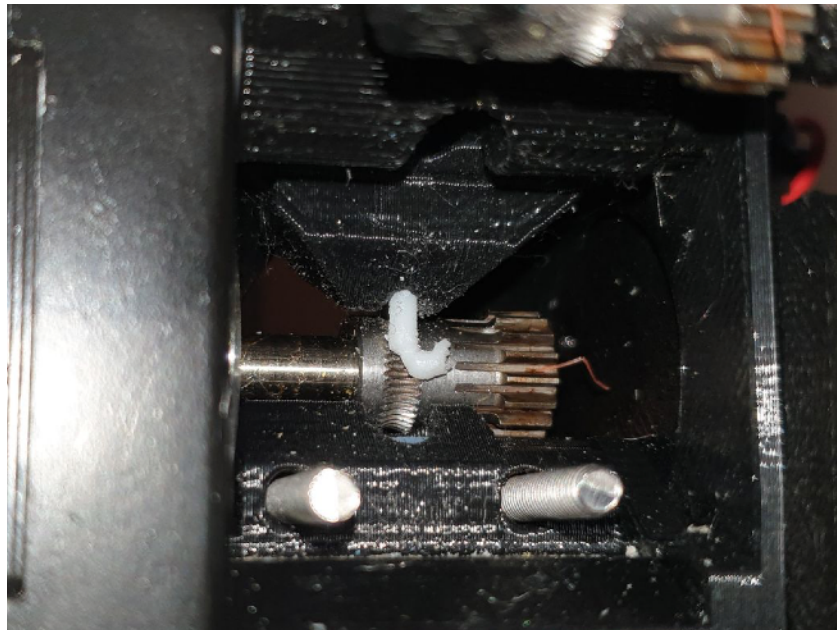
1. Reload filament. Yes, just by going into "load filament" and reloading the filament in the printer will re-form the end so that it may be unloaded again. Keep in mind to pull quite firmly on the filament as you click "unload filament" so that it doesn't get stuck again.



2. If the filament won't load again or you're just unable to unload it you have to resort to open the "idler door". This is done by unscrewing the two hex screws on the left side of the head. This will release the door and allow you to open it.

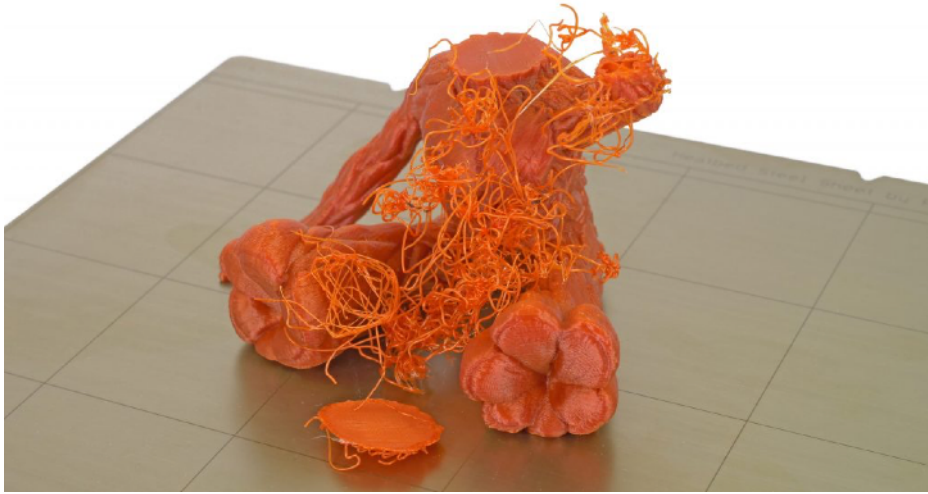


Now you should see the blob (similar to the image below). You should be able to grab a small pair of side cutters and cut the end off, which then allows you to pull the filament out.



Retighten the idler door to the point where it just grabs a piece of filament and won't allow you to pull it back out. They should not be screwed all the way in!

Spaghetti monster!

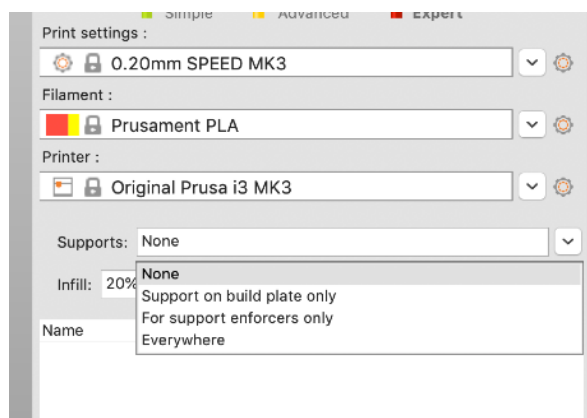


If you come to the printer and it looks like the image above, you have created spaghetti... Congratulations you're now a Prusa Chef@!
All jokes aside this terrible failure can be due to a number of things but the most common are:

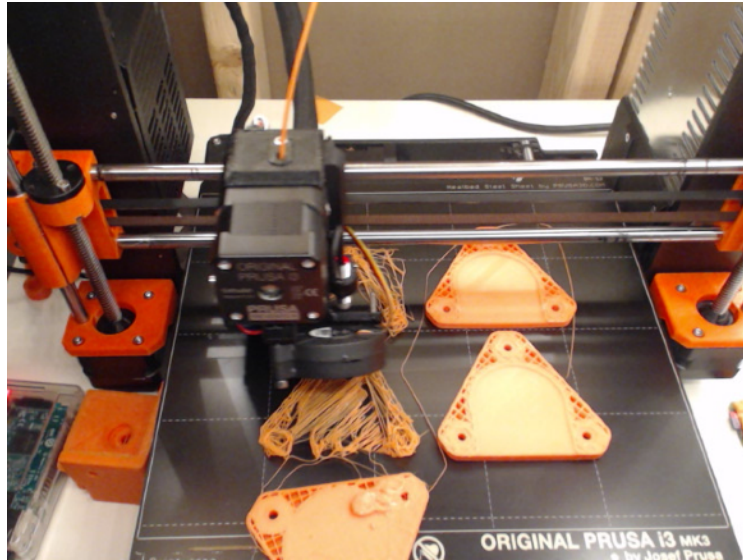
1. Forgotten to enable support. Support is sacrificial material that the slicing software (eg. Prusa Slicer) puts in to help the printer print large overhangs and unsupported structures. Like the image shows below, this wolf creature needs support and it can be generated in several ways.



Automatic support can be generated in Prusa Slicer either from just the build plate or from all surfaces. If you're unsure, select "Everywhere" and it will put in support wherever it thinks necessary. You can preview the supports by clicking "Slice now" at the bottom.



2. Partial clog. Like mentioned before, a partial clog can cause intermittent printing errors. If the print settings were correct and the print still turned out like the first image, this might be due to a partial clog and the nozzle should be cleaned.
3. Print dislodged from the build surface. If you're printing a small model or just several of them you might experience that the print completely dislodged from the build platform. This is usually very apparent, since the bottom of the part will be somewhere else on the build platform, covered in spaghetti. The solution to this is to improve the bed adhesion which was discussed in the "Warping topic" above.



The blob



If the printer looks like the picture above, you have created a blob. This can happen if the print lifts from the build platform and gets stuck to the warm nozzle. This can be fixed, however it's very easy to damage the printer while fixing it. So it is always recommended to consult help before attempting the repair.