



# PRINTING THE PLASTIC PARTS FOR THE ORIGINAL PRUSA I3 MK3S+

We have decided to prepare a short guide explaining how to print the printer parts with ease.

## PRINTING THE PLASTIC PARTS FOR YOUR PRINTER

Almost all\* of Prusa 3D printed parts are from PETG, but if you are more used to printing with ASA/PC, you can absolutely use it for printing the parts instead of PETG.

\*for Fan-shroud part, please use ASA/PC

To print all the parts properly, please follow this guide:

1. Clean the print surface, remove any dirt or grease.
2. Download G-codes or .stl files for the upgrade from [prusa3d.com/prusa-i3-printable-parts](https://prusa3d.com/prusa-i3-printable-parts). These G-codes are for PETG only! If you want to prepare your own G-codes, use PETG or similar material settings with at least 20% grid infill.
3. Start the print, wait and observe the first few layers.
4. If you suspect that the first layer does not stick properly, tweak the Live adjust Z value. You can do it from the menu while printing. Press the **Knob, select Live adjust Z and then lower the value slowly until the plastic sticks nicely**. If object is already detached from the print surface, clean up and start the print again.
5. Once the print is finished, wait until the heatbed cools down to ambient temperature. **Do not remove the printed objects right away**. Otherwise, you might damage the print surface.
6. Once the parts are ready, you can just follow the manual for your printer upgrade available at [help.prusa3d.com](https://help.prusa3d.com).

## TROUBLESHOOTING

If you struggle with printing plastic parts, here are some troubleshooting tips to help you out.

- Make sure the print surface is clean. If the Windex didn't work, try to clean the surface **with dish soap water** (dish soap water can actually **dissolve sugar residue**).
- For smooth PEI try to use the bundled glue stick, which increases adhesion for PETG and creates a separation layer.
- Generate your own G-codes using **PrusaSlicer**. Select PETG or similar material with at least 20% grid infill. Place only 2-3 objects around the center of the heated bed for each individual G-code.
- If none of that made any difference, try to start the print and slower the print speed to 70-80%. You can do that by **rotating the knob in the counterclockwise direction**.

If you have any issues with printing the plastic parts, do not hesitate to contact our support team through Live chat or e-mail [info@prusa3d.com](mailto:info@prusa3d.com).