

Rigid X

2021-11-13

RIGID X NOZZLE X

How to print with Rigid X

Recommended printing temperature: 245-265°C nozzle temp and 70°C on the bed.

Rigid X is very hygroscopic, so to get the best result both visual and strength wise, it's important to keep dry. When not in use, store it in our re-sealable bag with the included desiccant. If needed, dry it for 5-6 hours at 65°C before use. Signs of moisture are: Poor layer adhesion, weak and fragile prints. Excessive stringing and oozing.

When you get the settings right, Rigid X prints easily. The most common issues are: Stringing and oozing due to higher print temps and material properties. Smearing and residue build up on the nozzle caused by the nozzle being too close to the bed.

To prevent excess smearing and residue buildup on the nozzle, turn on "z hop while retracted" in your slicer software.

General tips on 3D-printing temperatures

Finding the right printing temperature for any filament could be a compromise between visual looks and strength. With lowered nozzle temp you often get rid of problems like oozing, stringing and are able to handle cleaner bridging and steeper overhangs. However lowered temps will affect layer adhesion, providing weaker parts. Every printer is different so to get the best result experiment with nozzle temperature until you're satisfied with the result. A good advice is to start in the middle of the recommended temperature span and work your way down if needed for greater visual looks or the other way around for increased strength. With some patience and tuning you can achieve both. It's always a good idea to print a temp-tower whenever you test a new filament. You can find numerous of them over at "Thingiverse", this one for example [link](#).

How to get Rigid X to stick to the bed

Rigid X sticks well to most bed surfaces. If you need increased adhesion or to prevent warping when printing large objects, use glue stick, hairspray or Magigoo. A PEI-sheet, smooth or textured is a really good investment to help your prints to stick. Be careful printing directly on glass, always use some sort of barrier like glue stick or similar in between. If not, you can have a hard time getting your prints off and the glass might break.

General tips on bed adhesion/leveling

The first rule of 3D-printing – always get a perfect first layer, 😊

Not too close, not too far, just perfect. If the printed plastic strings overlap each other and plastic oozes out to the sides, the nozzle is too close to the bed. The opposite would be that the plastic has a hard time sticking to the bed and that you'll see gaps between the printed lines. A perfect first layer is smooth to the touch, without any gaps. Filaments like PLA often like to be a bit squished to the bed for a good first layer. PETG on the other hand needs more clearance to prevent residue buildup on the nozzle.

If you have trouble with warping, check your surroundings for drafts or low temperatures. At least 20°C ambient temperature is recommended. If needed, try an enclosure for better results. If you plan on printing an especially complicated model with a lot of details on the first layer, lower your first layer printing speed to 15-20mm/s to ensure a good first layer.

For extra bed adhesion, if needed, bump up the first layer hotend and bed temperature 5-10°C and use a brim.

Improve visual quality of your 3D-prints

The second rule of 3D-printing – lowered print speeds will produce better looking prints. Regardless of what printer you use, you'll most likely get better results and improved visual quality if you lower your print speeds. This along with decreased layer heights of course. For good visual results we recommend a layer height of no more than 0,15mm and a printing speed that doesn't exceed 50mm/s.

To further improve visual quality, tune your retractions speeds and distances to get rid of unwanted strings or blobs. If you still have problems, try lowering your nozzle temp in 5°C increments.

Check that your part cooling fan is optimized and working correctly. Cooling is needed to handle overhangs and bridging and can make a world of difference to improve visual part quality.

For a nicer surface finish on your top layer, you can enable "ironing" in your slicer software, it could make major improvements on especially PLA-prints. (Works only on flat surfaces).

Increase the mechanical strength of your 3D-prints

Rule number three in 3D-printing - Maximal strength is achieved with as high layer height and temperature as possible, and as little layer cooling as possible.

Are you willing to sacrifice a bit of the visual quality for maximum strength of your part? Then you should instead use as high layer height and printing temperature as possible and as little cooling fan as possible. The higher layer height you have, the longer the extruded string will remain warm and the bigger the contact surface between the layers become when the string is

extruded. The higher the temperature, the harder the bonding between the layers. And finally, less layer cooling flattens the cooling curve of each layer which gives the material more time to bond firmly to the layer below. This also means that bigger nozzles produce stronger prints, and is also the reason why an elevated print chamber temperature is useful sometimes.

If you want both strong and visually good prints you need to find a compromise that will balance both strength and overhangs/bridging/detail resolution. Then slicer settings like minimum layer time with dynamic cooling fan, bridge fan override etc. comes into play, which is part of the somewhat higher school of 3D-printing, which it is impossible to give general advice on since it differs a lot from case to case.