E-PLA

2021-11-13

E-PLA

How to print with E-PLA

Recommended printing temperature: 195-225°C nozzle temp and no or up to 60°C on the bed

Our E-PLA is our most easy to print filament and should print trouble free on any 3D-printer. Regular print issues that you can run into printing with PLA are: Stringing caused by improper retractions settings or too high temperatures. Bad bed adhesion caused by a dirty/greasy bed. Improper bed leveling. Bridges and overhangs that sag caused by insufficient part cooling or too high print temperatures.

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 E-PLA to stick to the bed

E-PLA 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.

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.