

PRINTING GUIDE

PREPARED BY:
STANDARD PRINT CO.
www.standardprintco.com
info@standardprintco.com
Suite 5 / 1-5 Woodburn St
Redfern, 2016, NSW

PRINTING GUIDE

VERSION: 1.1 - 2020

PREPARED BY:
STANDARD PRINT CO.
www.standardprintco.com
info@standardprintco.com
Suite 5 / 1-5 Woodburn St
Redfern, 2016, NSW

Step 1: Slicer

Tip 1: Extruder Temperature



Nozzle: 190°C - 210°C



The vast majority of 3D printing problems encountered when printing in PLA are caused by incorrect extruder and bed temperatures. When printing in PLA start with the lowest temperature recommended by the manufacturer - only increasing temperature if extrusion problems become evident.

We recommend an **extruder temperature of 200°C**.

Tip 2: Bed Temperature



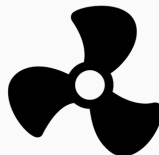
Bed: 50°C - 65°C



While PLA can be printed on an unheated bed, for best results we recommend a bed temp of between **50°C - 60°C**.

If poor adhesion is an issue, print the first two layers **ONLY** with the bed at 65°C (and 60°C for every subsequent layer). Do not maintain a bed temp >65°C for more than the first two layers as this can cause the print to warp (see: Elephants Foot) or in extreme cases for the print to dislodge from the bed completely.

Tip 3: Fan Control & Minimum Layer Time



100% Fanspeed



PLA prints best when rapidly cooled with the aid of your extruder's fan. This will ensure a clean, crisp layers with neat details.

For the best surface resolution possible, set the **extruder fan speed to 100%**. For smaller parts, manually setting a **minimum layer time of >15 seconds per layer** will ensure each layer is solid before another layer is printed on top - preventing sloppy layers from forming.

Tip 4: Retraction



Retraction Speed: 30-80mm/s
Retraction Distance: Bowden: 2 - 5mm
Direct Drive: 1 - 2mm



Increasing the retraction speed and distance will decrease that amount of material that oozes out of the nozzle between print moves. Turning on retraction settings in your slicer will tell your printer to actively rewind some filament at the end of a print move, causing a negative pressure to build up in the nozzle.

We recommend starting with a **retraction speed of between 30-80mm/s** and a retraction distance of Between **2-5 mm for a Bowden setup or 1-2mm for a direct drive printer**.

Tip 5: Build Plate Adhesion



Print with a Raft or Brim



While PLA is less prone to warping compared to other materials, even PLA can use a little extra help sticking to the bed sometimes.

If your prints require a little help sticking to the bed, try printing with a **Brim** - a single layer flat area around the base of the model to prevent warping. If the model is not completely flat or has little adhesion to the build plate, printing with a **Raft** may be a better option.

Tip 6: Print Slower



45-80mm/s Print Speed



With proper fan cooling, PLA can achieve blistering print speeds but if you are having problems with interlayer adhesion or print quality, try printing a little slower. Printing at speeds greater than 80mm/s and your printer may struggle to lay filament down fast enough. Layers may have problems sticking to one another, leading to weak prints and poor surface resolution.

We've found that printing at 55mm/s or slower yields great results. It represents for us the sweet spot between print speed and achieving great surface resolution.

Tip 7: Flow Control



Decrease Flow in 5% Increments



PLA can be sensitive to over extrusion. If you've tried all the steps outlined above and you're getting sloppy layers, the problem may be linked to over extrusion. This occurs when your printer extrudes more filament than intended, causing sloppy layers and in some cases, excessive material to build up on the hot end.

To remedy this issue, simply **reduce the flow of filament in 5%** increments to prevent over extrusion. Be careful not to reduce it too much as this can lead to poor part strength and thin, wispy layers.

Step 2: Environment

Tip 8: Drought Free Environment



Keep Printer Sheltered



For best results, it's important that your 3D printer is placed in an environment that is kept at a relatively constant temperature and free of draughts. Draughts and cold breezes can cause undue warping on your parts as they print, causing warping and print failure.

Keep your 3D printer in a sheltered environment. Corridors, thoroughfares, large open spaces and rooms with high foot traffic are not ideal places to setup of your 3D printer in.

Tip 9: Open Enclosure



Enclosure Door: Open



As rapid cooling of the print is key for PLA to produce great surface resolution (unlike other materials such as ABS, and HIPS), keeping the printer in an enclosure with the door closed can be detrimental to achieving great print quality.

In extreme cases, the enclosed space of the enclosure can cause heat to build up and cause feeding and printing issues. For best results, keep the door your your print enclosure open to allow for proper cooling.

Tip 10: Maintain Good Ventilation



Leave Print Room
When Printing



Good ventilation is also key to your health as the operator. While printing in PLA generates far fewer Volatile Organic Compounds (VOC's) than materials such as ABS and HIPS, it's still important to ensure good ventilation of the room your printer is located in if you plan on being present in the room for long periods of time. If proper ventilation cannot be maintained without causing draughts and breezes, consider working in another room while the printer is active. If you must be present in the room when the printer is in operation, you may consider using an air purifier such as a Zimpure to keep VOC's within acceptable levels.

Step 3: Filament Care

Tip 11: Keep Filament Dry



Keep Filament Dry



PLA is prone to absorbing moisture from the air when left out, resulting in weaker interlayer bonding on the molecular level. When printing wet filament, small bubbles can appear on the skin of your print marring an otherwise smooth surface. In severe cases, steam and small popping noises can be noticed exiting the nozzle when printing.

Wet filament can be rescued by **dehydrating it in an oven at 60°C for a minimum of 6 hours or by storing the filament in a container with ample amounts of desiccant for a minimum of 24 hours.**

Tip 12: Unload Filament



Unload Filament
When Not In Use



Whenever possible, when not printing with PLA (and all other filaments for that matter), unload the filament from the printer and store it in an airtight container in a cool dry place to keep it dry. Ensure the **desiccant that comes with the filament is also stored in the same container** to keep the filament in optimal printing condition. **Additional desiccant may be required** depending on the size of the container and amount of filament being stored. **Aim to keep the humidity level within the container at 10-15%.**



Still having printing issues?

We've got you covered.

**Check out our full PLA
troubleshooting guide [here](#).**