



PLA

PLA filament is one of the most widely used material in 3D printing. It is made from polylactic acid, a plant-based polymer typically derived from renewable resources such as corn starch or sugarcane. Due to its reliable performance and user-friendly nature, PLA is a popular choice for both beginners and experienced 3D printing users.

Key Advantages of PLA Filament

Easy to Print:

PLA has a relatively low printing temperature and minimal warping, making it easy to print on most FDM 3D printers. Its stable printing behavior makes it especially suitable for beginners who are learning 3D printing.

Good Strength:

PLA offers good tensile strength and rigidity, making it suitable for many functional parts, prototypes, and lightweight components.

Smooth Surface Finish:

PLA produces prints with clean details and a smooth surface finish. This makes it ideal for models, prototypes, and applications where appearance and surface quality are important.

Eco-Friendly Material:

PLA is made from renewable plant-based resources such as corn or sugarcane, making it a more environmentally responsible alternative compared to many petroleum-based filaments.



Proper Bed Levelling and First Layer Setup

Proper bed adhesion and correct bed levelling are essential for successful 3D printing. The most important step is ensuring that the first layer is printed correctly, with no gaps or excessive overlap. This can be achieved by adjusting the distance between the nozzle and the build plate so that it is neither too close nor too far from the surface.

Different filaments require slightly different first-layer settings. PLA typically needs a small amount of “squish” against the bed to ensure good adhesion and a smooth first layer.

If you notice warping or corners lifting, check the printing environment for drafts or low ambient temperatures. In some cases, using a printer enclosure can help maintain stable conditions during printing.

For models with fine details or small contact areas, reducing the first layer print speed to around 15–20 mm/s can help improve adhesion and ensure a consistent first layer.

If additional bed adhesion is required, you can try:

- Increasing the nozzle and bed temperature slightly (about 5–10°C)
- Using a brim or skirt to increase the contact area with the bed

By focusing on proper levelling, correct first-layer settings, and stable printing conditions, you can significantly improve the reliability and quality of your 3D prints.



Ensuring Strong Bed Adhesion for PLA Prints

Most common build surfaces work well with PLA, but proper setup is important to ensure good first-layer adhesion. Here are some tips to help PLA stick well to the print bed:

Use a Clean and Level Bed:

A clean and properly levelled build plate is essential. Dirt, grease, or an uneven bed can prevent the filament from sticking correctly. Clean the bed regularly with isopropyl alcohol.

Use a Heated Bed:

While PLA can print without a heated bed, using one improves adhesion. A recommended bed temperature for INFILL PLA is around 35–45°C, which helps the filament adhere better during the first layers.

Apply a Bed Adhesive:

If you experience adhesion issues, using a light layer of adhesive such as a **glue stick**, **hairspray**, or **specialized 3D printing adhesive** can improve bed grip and reduce the chances of prints lifting.

Use a PEI Build Surface:

PEI sheets are widely used for PLA printing. They provide a reliable surface that helps prints stick during printing and release easily once the bed cools.



Choosing the Right Nozzle Temperature for PLA

Selecting the right printing temperature is important for achieving a balance between print quality and part strength. Lower nozzle temperatures can help reduce issues such as stringing and oozing, while also improving bridging and overhang performance. However, temperatures that are too low may lead to weaker layer bonding and reduced part strength.

The ideal printing temperature can vary depending on the **3D printer, filament quality, and the type of model being printed**. A good approach is to begin with the **middle of the recommended temperature range** for the filament and then make small adjustments based on the print results.

For best results, it is helpful to experiment with different temperatures and settings. Printing a **temperature tower** is an effective way to compare how different temperatures affect surface finish, layer adhesion, and overall print quality. With careful tuning and testing. You can find the numerous temperature tower on MakerWorld, such as this [one](#).

Improving Surface Finish and Print Appearance

Achieving high-quality prints often requires optimizing your print settings. One of the most effective ways to improve surface finish is by **reducing print speed**. Slower printing allows the printer to place material more accurately, resulting in cleaner details and smoother surfaces. For better visual results, a **layer height of around 0.10–0.15 mm** and a **print speed below 50 mm/s** are commonly recommended.

Adjust Retraction Settings:

Fine-tuning retraction distance and speed can help reduce common issues such as stringing or small blobs between parts of the print. If stringing continues, try lowering the **nozzle temperature in small steps (about 5°C)** until the issue improves.



Use Proper Cooling:

Ensure the **part cooling fan** is working correctly and properly configured in your slicer. Good cooling improves surface quality and helps achieve better results when printing **overhangs and bridges**.

Enable Ironing for Smoother Top Surfaces:

Many slicer programs include an **“ironing” feature**, which lightly passes the nozzle over the top layer to smooth it out. Enabling this setting can significantly improve the finish of the top surfaces when printing with PLA.

By optimizing speed, temperature, cooling, and slicer settings, you can significantly enhance the overall appearance and surface quality of your 3D prints.

Preparing Your Printer for PLA Printing

Prepare Your 3D Printer:

Ensure your 3D printer is properly assembled and calibrated before starting. Clean the print bed with isopropyl alcohol to remove dust, grease, or residue to improve bed adhesion.

Use the Recommended Settings:

PLA prints at relatively low temperatures. A typical extruder temperature range is **190–230 °C**, while the recommended bed temperature is **35–55 °C**. These settings may vary slightly depending on your printer and model.

Load the Filament Properly:

Insert the PLA filament into the extruder and make sure it feeds smoothly through the drive gears and hotend without resistance.



Start the Print and Monitor:

Once printing begins, observe the first few layers carefully. Proper first-layer adhesion is important for a successful print.

Troubleshooting:

If you notice issues such as stringing, poor layer adhesion, or uneven layers, adjust the printing temperature, print speed, or bed temperature accordingly.

Additional Tips for Printing with PLA:

- **Store Filament Correctly:** Keep PLA in a sealed container < 20% RH to prevent moisture absorption.
- **Maintain the Print Bed:** Clean the build surface regularly with isopropyl alcohol to maintain good adhesion and easy print removal.
- **Ensure Good Cooling:** PLA generally prints best in an open chamber with part cooling fans enabled for better surface quality and detail.