Common Problem to Print PETG Filaments and Solutions

1) How to take care PETG Filaments?
   - Store filament in airtight container with desiccant to maintain optimal printing conditions.
   - PETG absorbs moisture, weakening interlayer bonding.
   - To rescue wet filament, dehydrate in a 50-60°C dryer for 2-3 hours or store with desiccant for 24 hours.

2) How to Solve Bed adhesion issue in PETG Filament?
   - Check bed level according to printer manual. Uneven beds can cause poor part adhesion.
   - Use brim option for increased first-layer surface area, improving adhesion, and minimizing corner lifting during printing.
   - Molten filament contracts upon cooling, stressing edges. Higher heat bed temp (60-80°C) reduces contraction, aiding adhesion. The first two layers at 80°C enhance results.IT varies model by model as well.
   - Set fan speed to 0% for initial layers to allow better filament flow and enhance adhesion.
   - Keep the print bed clean for better adhesion. Wipe glass with ammonia-based cleaner, sand PEI, and use a scraper for stubborn debris. For tough spots, print a PETG sheet the size of the bed, cool, and peel off with debris.
   - Upgrade to a Zi flex build surface for better adhesion on non-coated beds like aluminum or glass, addressing issues with smooth surfaces.

3) How to solve nozzle part collision issue in PETG Filaments?
   - Adjust Z offset for PETG to prevent molten material accumulation. Increase by 0.02mm over PLA/ABS settings; if issue persists, raise in 0.02mm increments up to 0.06mm.
   - Prevent warping in 3D prints by slowing cooling rate. Use an enclosure or print in a draft-free, warm area. Higher temperatures help, especially with materials like PETG.
   - Combat collisions in 3D prints by strategically placing support material below problematic areas. Use slicer's manual support generation for precise placement.

4) How to Prevent Stringing issues in PETG Filaments?
   - Prevent PETG stringing by adjusting Z offset in slicer. Increase by 0.02mm over PLA/ABS settings; if issue persists, raise in 0.02mm increments up to 0.06mm.
   - Prevent PETG oozing by starting at the lowest recommended temperature. Increase only if extrusion issues occur (grinding, slipping, or no extrusion).
   - For better surface resolution, increase fan speed to 100% for rapid cooling, sacrificing interlayer adhesion.
   - Prevent filament oozing during travel moves by increasing travel speed. Use a value 20% lower than the printer's maximum allowable speed for a balance of reliability and reduced oozing.

- Reduce nozzle oozing by adjusting retraction settings. Start with recommended values and adjust: 10mm/s for speed, 0.2mm for distance. For Bowden, use 2-5mm; for Direct, 1-2mm.

5) How to solve zits and blob issues in PETG Filaments?
- Avoid PETG "Zits" by adjusting Z offset. Increase by 0.02mm over PLA/ABS settings; if needed, raise in 0.02mm increments up to 0.06mm.
- Address PETG stringing/blobbing by combating over extrusion. Gradually reduce filament flow in 5% increments, ensuring proper extrusion. Print a calibration cube to validate optimal flow without compromising quality.

6) How to solve holes in print?
- Increase Z Offset-Prevent PETG holes by adjusting Z offset. Increase by 0.02mm over PLA/ABS settings; if needed, raise in 0.02mm increments up to 0.06mm. Ensure proper offset, especially with faster print speeds.
- Decrease Print Speed-Prevent PETG holes by adjusting Z offset. 55mm/s is the upper limit; start at 55mm/s and decrease in 5mm/s increments. Optimal results are often achieved at 35mm/s.

7) How to Prevent elephant foot in print?
- Fix elephant's foot by ensuring a level print bed. It's caused by uneven filament deposition. Before slicer adjustments, confirm optimal printer condition and bed level, often the initial solution for this issue.
- Address elephant's foot by adjusting Z offset. Increase by 0.02mm initially, and in 0.02mm increments up to 0.06mm if needed.
- Prevent elephant's foot by gradually lowering the heat bed temperature in 5°C increments. Ensure proper adhesion while avoiding temperatures below 20°C of the filament's recommended print bed temperature. If issues persist, explore other potential causes.

8) How to solve your warping problem around edges?
- Use heated Printed Bed-Modern printers have a heated bed just below the material's glass transition temperature, typically around 80°C for PETG. Standard Print Co. PETG has a glass transition temperature of 80°C.
- Print with Brim- Use the brim option for better first-layer adhesion, minimizing corner lifting and ensuring the print stays securely on the bed.
- Eliminate draughts & increase temp-Prevent warping in 3D prints by slowing cooling rate. Use an enclosure or print in a warm, draft-free area. Higher temperatures benefit most materials, especially PETG, which benefits from enclosure to retain heat.
- Heat up the bed before- Prevent warping by ensuring even heat bed heating. Let the bed stabilize at printing temperature for at least 10 minutes before starting a print.
- Temporary adhesive helps sometimes as well.
- Increase temperature- Print PETG with a heat bed temperature of 70-75°C to reduce base contraction. For improved adhesion, consider printing the first two layers slightly hotter at 80°C.

9) How to solve poor print from the start?

- Dry Filament- Keep PETG filament dry for optimal strength. Store in a sealed container with desiccant, away from UV light. Dehydrate wet filament in a 60°C oven for 6 hours 50-60C in dryer for 2-3 hrs. or store with desiccant for 24 hours.
- Decrease Fan Speed-Optimize print strength by reducing fan speed to 0%, though surface resolution may be affected. Begin at 0% and increase in 10% increments to find the desired balance between surface quality and print durability.
- Reorient Print-Boost part strength in FDM 3D prints by reorienting the part during slicing. Layer orientation can significantly impact overall strength.

10) How to remove print stuck from the bed?
- Wait for the part to cool off.
- If your 3D printer lacks a scraper, get one from a hardware store. If your print is stuck, cool the bed, then use a paint scraper to carefully lift a corner. Remember to relevel the bed afterward.
- To prevent excessive adhesion, clean your print bed periodically. Use water for glue residue or a cloth with isopropyl alcohol for hairspray buildup between prints, ensuring a clean and smooth surface.
- Switch to a Zi flex or similar composite build surface for improved adhesion on aluminum or glass beds. Coated surfaces provide a lasting solution, allowing prints to stick when hot and release when cool, addressing adhesion challenges during printing.

11) How to do best optimization to your 3d print and 3d printers?
- Wait for the part to cool off.
- Use a print scraper or re heat the bed to remove the model.
- Use the release agent if it's required.
- Clean up your printer frequently.
- Do Print bed upgrade periodically.