Part 7 - Failing prints and fixing 3D printers

Where to get help

- **Official Support Guides**
  Bambu Lab provides detailed troubleshooting resources:
  https://bambulab.com/en/support/804403369274351616

- **YouTube Tutorials**
  Many creators offer step-by-step guides for common issues and printer setups.

- **Manufacturer Support**
  You can contact the official support team—preparing a short video of the issue often helps them diagnose the problem more quickly.

- **Community & Friends & Maker spaces**
  Don't hesitate to ask friends or community members who have experience with 3D printing.

How to prevent prints to fail?

Printer or Filament Issues – Quick Checks

- **Verify the build plate**
  Make sure the print plate is correctly attached and properly seated on the build platform.

- **Check the surface**
  Ensure the build plate is clean and free of any leftover filament, debris, or previous print residue.

- **Confirm filament availability**
  Check that you have enough filament to complete the entire print.

- **Monitor long prints**
  For prints longer than an hour, check periodically (e.g., every 30 minutes) to ensure the filament is feeding correctly and not tangled or snagged.

- **Dry TPU before printing**
  TPU absorbs moisture easily, so drying it beforehand is recommended for better print quality and more reliable extrusion.

Model Issues

- **Parts are too thin or fragile**
  In design tools like Tinkercad, a part may look solid and strong on the screen, but in reality it might be too thin to print reliably. Thin or filigree features can break easily after printing.

- **Model requires supports**
  Some models need support structures to print correctly. Without proper support, overhangs and bridges may sag or produce poor-quality surfaces.

- **Excessive or poorly placed supports**
  In some cases, the model does require supports, but the slicer may generate too many or place them in difficult areas. This can damage the model during removal or make post-processing very time-consuming.

- **Unwanted holes or small gaps**
  The model may contain tiny gaps, holes, or non-manifold geometry that can cause slicing problems or result in weak printed parts.

Slicer Issues

- you want to print TPU but you made the setup for PLA

- you have several complex models in the slicer and sliced/printed wrong one