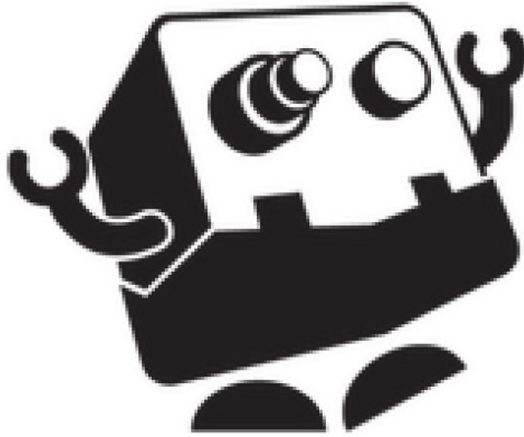




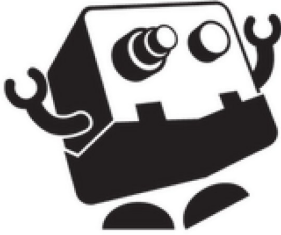
Ugly Print Troubleshooting

This guide will discuss some issues associated with "Ugly" looking 3D Prints

Written By: BoXZY

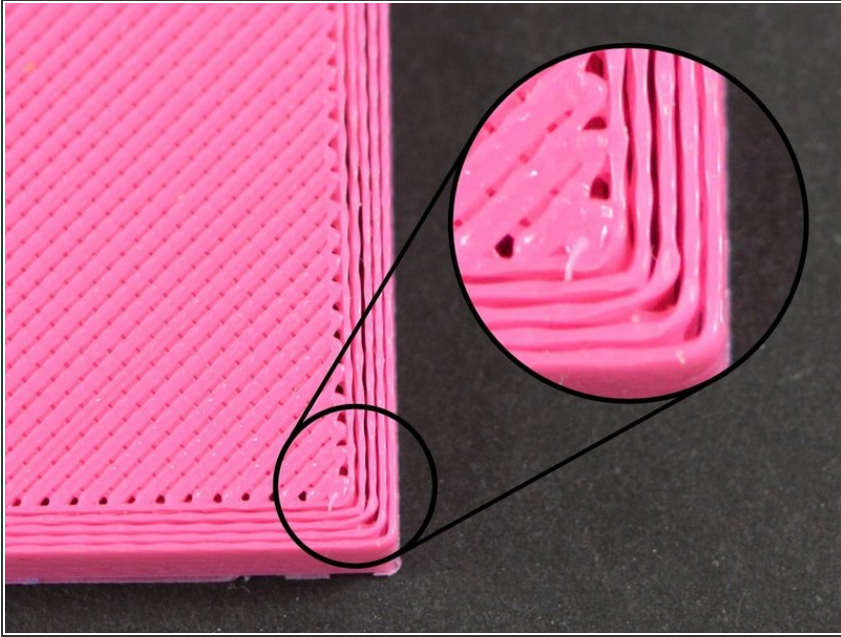


Step 1 — Examining the Print



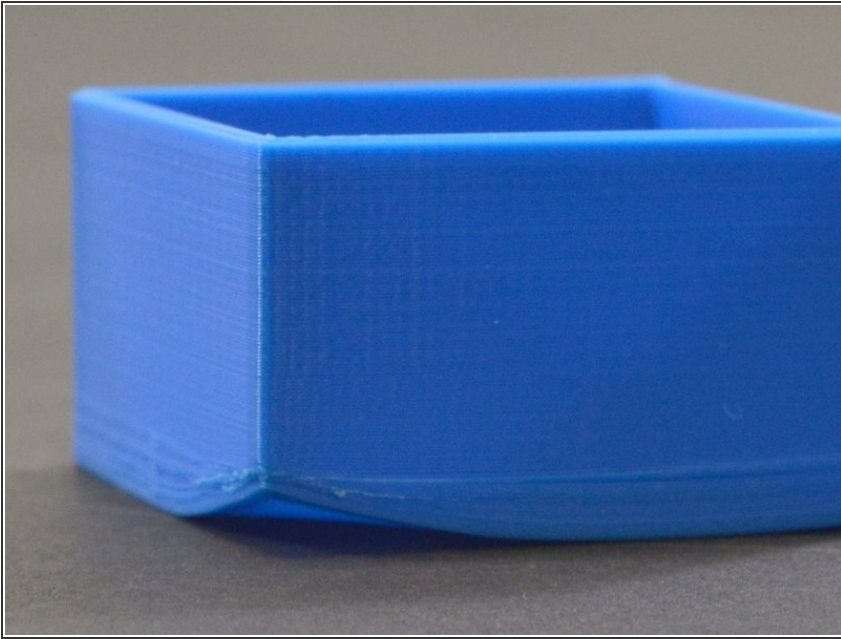
- When examining your print it is important that we know what we are looking at so please become familiar with the following
- The type of **Plastic** being extruded
- The **First Layer** that is directly applied to the printing surface
 - ⓘ There are also *Rafts* or *Brim*s/*Skirt*s that can be printed along with the First Layer
- The **Shells** that make up the Exterior Perimeter of each layer in the part
- The **Infill** that fills the void space between the Shells of each layer
- The **Support Material** generated to support overhanging elements of the 3D Part

Step 2 — Under Extruding



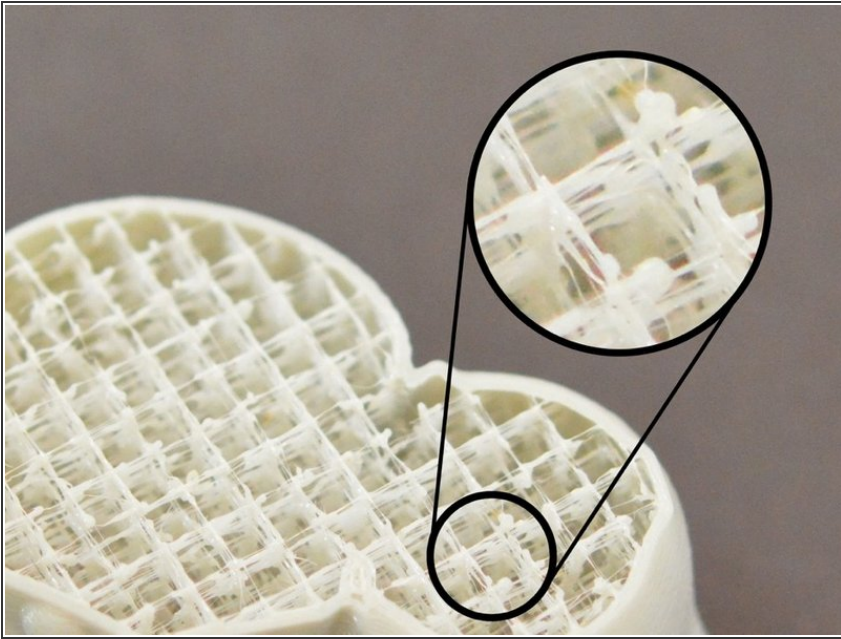
- *Under Extruding* is likely if the **Shells** of your 3D Part are not well adhered together or if there are obvious gaps between the 3D Part **Shells**
 - Examine and Check:
 - If the **Nozzle** is clogged
 - If the **Filament Drive** is grinding or deforming the Filament
 - If the **Speed** is too *Fast*
 - If the extruding **Temperature** is too *Low*

Step 3 — Warping



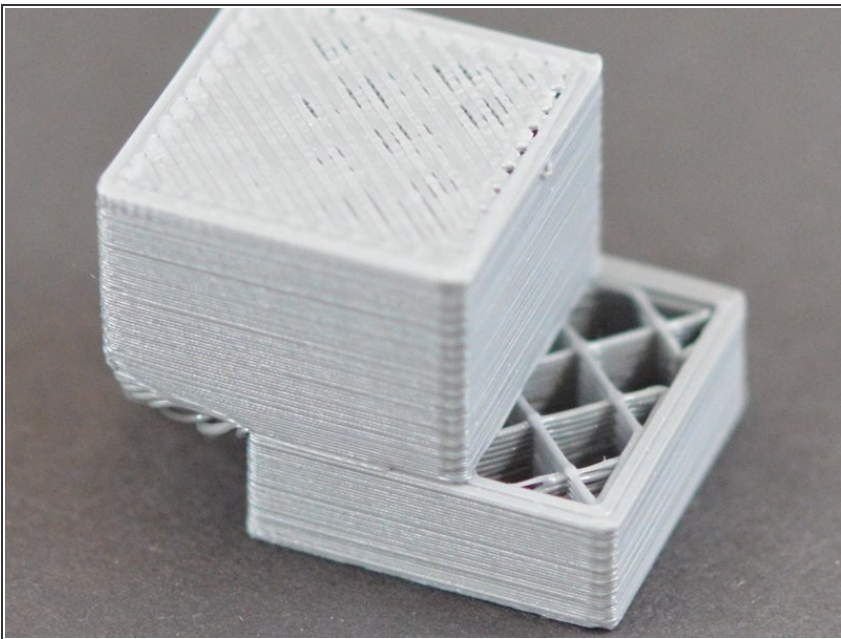
- **Warping** can happen through the first few layers of a print
 - First try printing with a **Brim/Skirt** or a **Raft**, this will provide more surface area on the first layer of print and will adhere to the platform easier
 - Try **Disabling** the **cooling fans** in the *Slicer Configuration Settings*
 - Use a **Heated Enclosure** or an Enclosure to keep heat in. This will keep a consistent and concentrated heat throughout the print
 - Install a **Heated Bed** to keep the first few layers of the print warm and adhered properly

Step 4 — Weak Infill



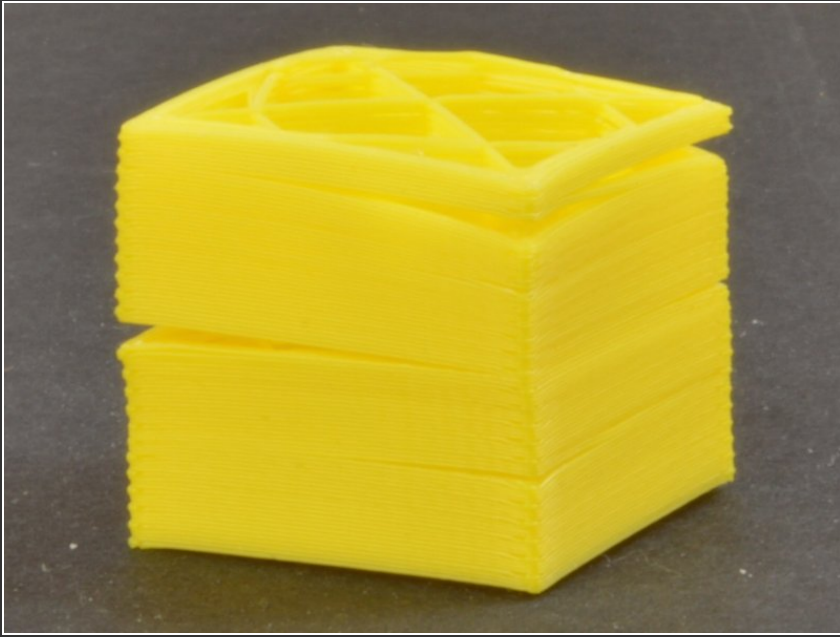
- **Infill** is the pattern that fills the internal space of your 3D Part. The 3D Part can be Infilled with various patterns
 - Try a **New Infill Pattern**
 - *Slow* the **Print Speed** down
 - *Change* the **Infill Extrusion Density** percentage

Step 5 — Layer Shifting



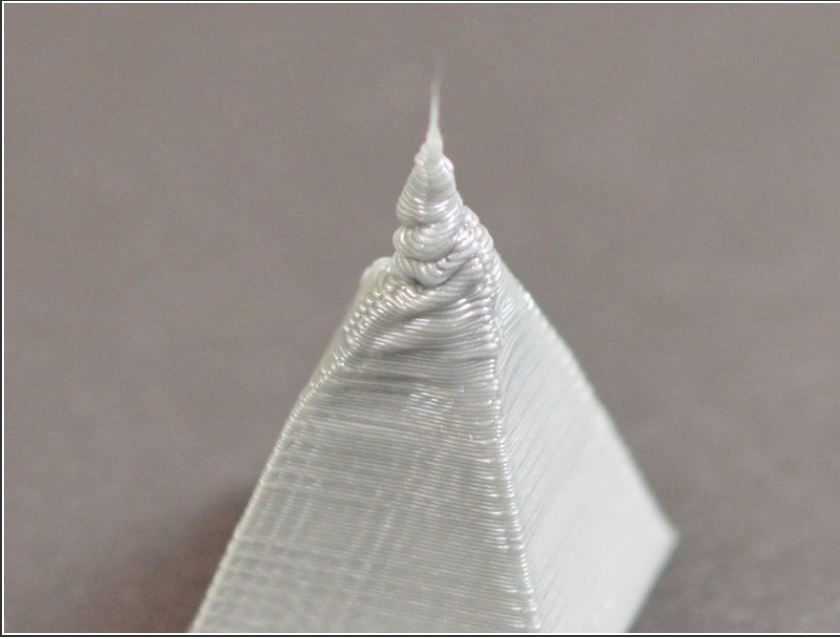
- **Layer Shifting** is often a result of something physically acting upon BoXZY
 - A heavy hit on BoXZY could cause the stepper motors to skip steps and offset an axis in such a manner
 - Loose Belts/Pulleys or loose set screws may also cause the motors to skip steps
- A communication breakage, USB disconnect/reconnect situation could also cause the stepper motors to lose their positioning in such a manner

Step 6 — Layer Separation



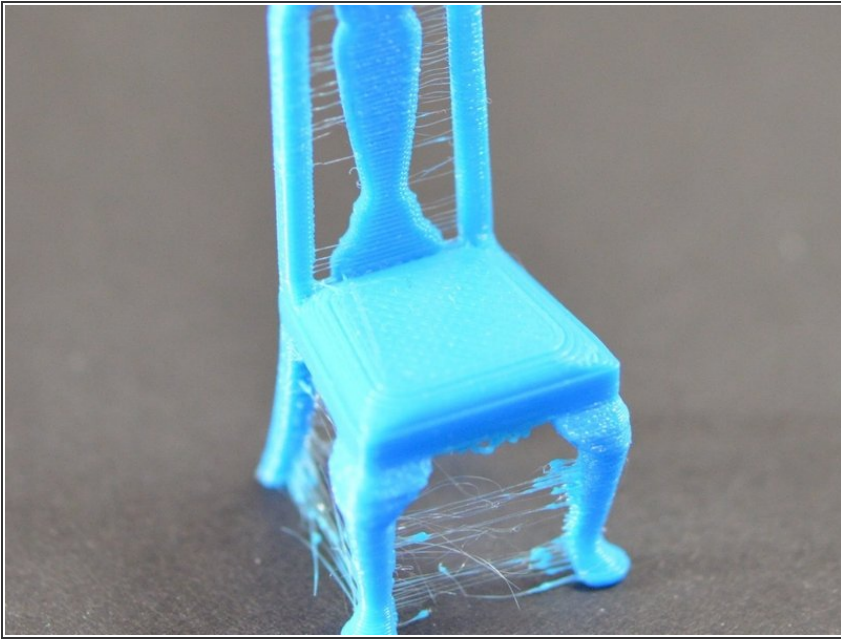
- Layer Separation is the a lack of adhesion between the layers of your 3D Printed Part
- ① A lack of adhesion is typically a temperature related issue
 - This could mean that the Extruder is *not hot enough* to allow the plastic to adhere to what is beneath it
 - This could also mean that the Fan is *cooling the plastic too aggressively* before the plastic can adhere to what is beneath it

Step 7 — Overheating



- **Overheating** is when the plastic has become so hot that it is unable to cool and solidify in a predictable and repeatable manner
 - It could be caused by a lack of cooling from the 3D Printing Attachment Fan
 - It could be caused by an increase in the extruding temperature
 - This could also be caused by printing too fast while printing small segments
 - It is less likely however it is still possible that this could be caused by drastic changes in room temperature

Step 8 — Stringy Prints



- **Strings** will arise on a print when the nozzle moves between areas where it is printing because plastic seeps out of the nozzle as it moves
- Try *lowering* your **Printing Temperature** incrementally until it prints and does not have strings
- *Edit* your **Retraction Distance** and **Speed** by accessing the Configuration menu for either Slic3r or CuraEngine
- *Edit* your **Travel Feed Rate** (speed between printing locations) in **Printer Settings > Printer**