Troubleshooting printers

Intro

While printers normally function fine, there is a wide variety of things that can happen that interfere with day to day printing. Here are some along with how to identify them as well as some potential solutions.

Heat creep

What is it?
Heat creep specifically describes how heat travels (sneakily) up the hot end and melts filament too early, before the melt zone.This problem can manifest mid-print or after a print (during cooling) but usually occurs and is noticed in the middle of a print when temperatures are highest.

How do I know?
You can see evidence of heat creep when your 3D printed part looks only half-complete with a fuzzy top. You can also detect heat creep when you pull your filament out and there’s an air bubble.

What causes it?

- Hot end temperature is too high
- Cooling fan is broken, not working properly, or isn't cooling the hot end enough
- Too high retraction length
- Heat sink is dusty
- Printing speed is too low
- Filament is in the hot end for too long

Heat creep cont.

Solutions:

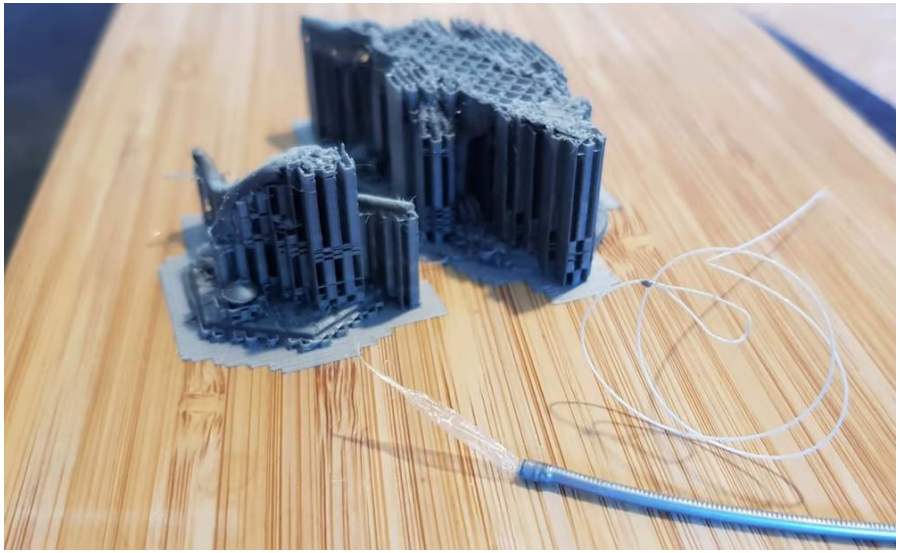
- Lower the hot end temperature:
  - Lowering the hot end temperature means less heat will be present in the hot end. This can help prevent heat creep from causing clogs, though it isn't a super permanent fix. And of course, you can only do so as long as you're within the temperature range required by your material.

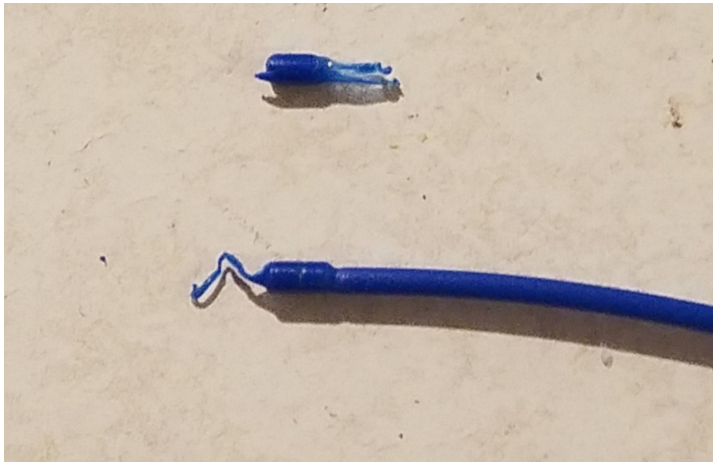
- Increase Fan speed:
  - First, you should set the fan to 100% speed and make sure that the heat creep stops occurring. After it does, lower the speed in increments of 10% if you experience warping, cracking, or other cooling-related print issues.

- Increase printing speed:
  - You should use the same temperature with a speed of 10-15 mm/s faster than what you normally use. Consider adjusting your printing speed in increments of 2-5 mm/s until you find the right mix between proper melting and preventing under-extrusion.

- Replace the hot end: (Last resort)
  - Though all-metal hot ends are often seen as upgrades over others, they could act as a heat creep catalyst if other conditions are resulting in an excess of heat. In this case, you might consider using a PTFE-lined hot end.
  - A PTFE-lined hot end, along with the removable PTFE tube, provides more insulation for the filament and preserves the sub-melting temperature until the filament hits the melt zone. This solution prevents heat from getting into the hot end but doesn't stop the problem.

Heat creep cont.





Under extruding

What is it?
Under-extrusion is the term given to the printer not supplying sufficient material for the print. Under-extrusion has many telltale signs — most significantly thin layers, unwanted gaps and even missing layers entirely

What causes it?
There are several possible causes. First, the diameter of the filament used does not match the diameter set in the slicing software. Secondly, the amount of material that is extruded is too low because of faulty slicer software settings. Alternatively, the flow of the material through the extruder is restricted by debris in the nozzle

How do I know?
Layers will look thin and wimpy, layers will also sometimes have a sputtering look to them.

Under extruding cont.

Solutions:

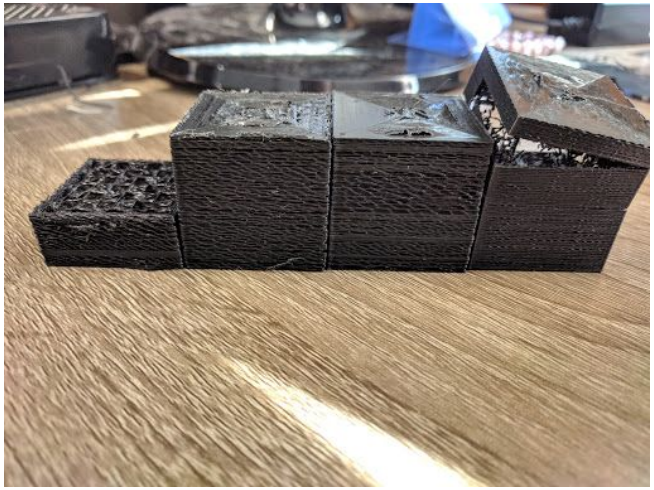
- Check the filament/nozzle diameter
  - Start with the simplest issue, have you set the correct filament diameter in the slicing software. If you're unsure about the diameter the value along with the recommended temperature is usually printed on the box.

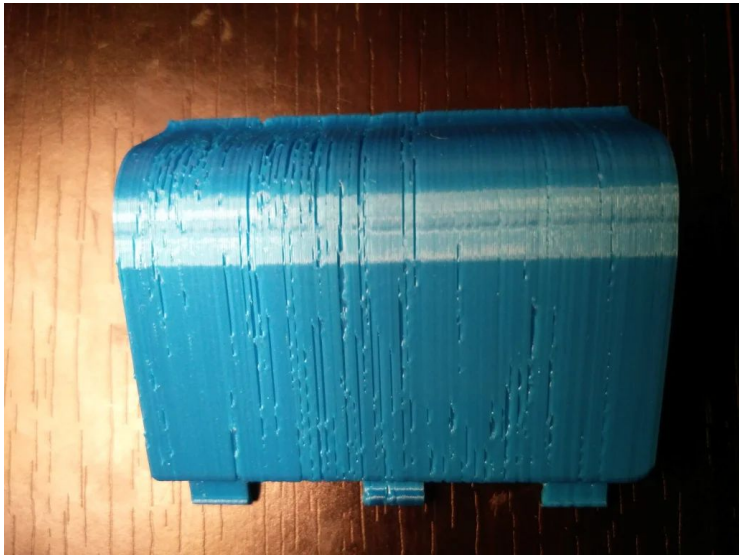
- Measure the filament
  - If you're still not getting the results you want and filament flow is the issue, then use a set of calipers to double check the filament diameter. You should be able to tweak the filament diameter settings accurately in the slicer software settings.

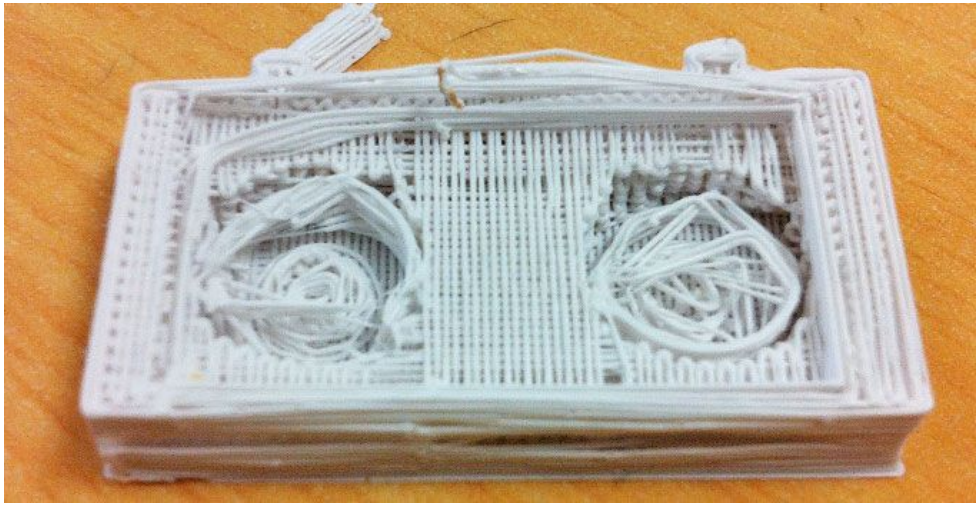
- Check the hot end for debris
  - After printing, most printers will lift the printhead away from the print base. Quickly check that the nozzle is clear from a build up of filament and dirt.

- Adjust the extrusion multiplier
  - If there is no mismatch between actual filament diameter and the software setting, then then the extrusion multiplier (also referred to as the flow rate or flow compensation) may be too low. Each slicer application will handle this slightly differently but the principle is to increase this value in steps of 5% until you see the problem is gone.

Under extruding cont.







Over extruding

What is it?
Over-extrusion means that the printer supplies more material than needed. This results in excess material on the outside of the model printed

What causes it?
Typically, the Extrusion multiplier or Flow setting in your slicing software is too high

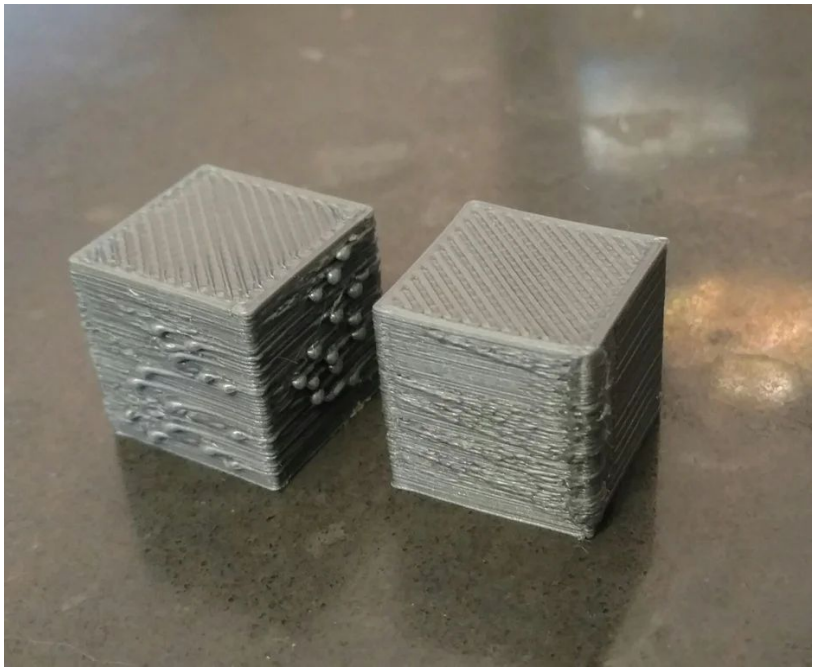
How do I know?
The print will be stringy and droopy

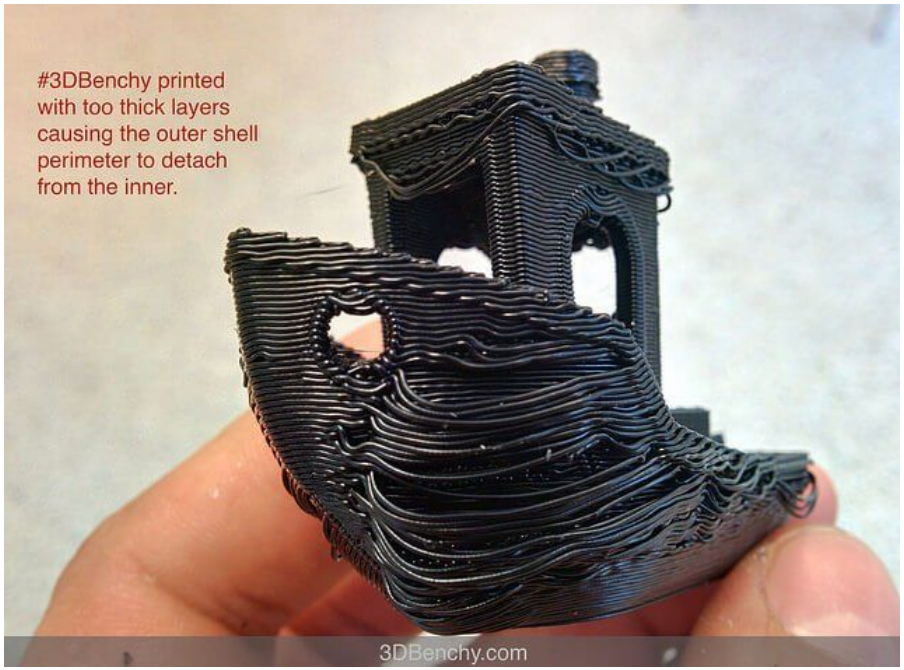
Over extruding cont.

Solutions:

- Check you have the right extrusion multiplier settings
  - Open your slicer software and check that you have the correct Extrusion multiplier selected
- Decrease your flow settings
  - If that all looks correct then decrease the Flow setting in your printer’s software.

Over extruding cont.





Loud beeping noise and a black screen

What is it?
When the printer is turned on, the menu screen will be black and a loud beeping sound will continuously yell at you.

What causes it?
Outdated, corrupt or improper firmware flashed onto the printer

How will I know?
There will be an obnoxiously loud continuous beeping sound and the screen will be black.

Loud beeping noise and black screen cont.

Solutions
-Format an SD card and load the .bin file below to it (can be found in the FRB drive additionally). Install the SD card into the printer and turn the printer on. Wait a few seconds and the printer will load the firmware on and refresh its system.

https://drive.google.com/file/d/188TpLAu2__Tdoh19C_zd3Y5HPfy-6gUk/view?usp=sharing

The nozzle is too close to the bed

What is it?
Inexplicably, despite loading the filament and the print head moving without a hitch, little to no filament is depositing on the print bed.

What causes it?
Quite simply, your nozzle may be too close to the print bed. If you've somehow tuned your print bed to mere microns from your nozzle opening, it's unlikely the melted filament has room to escape. At best your print will be missing its first layers, and have a higher chance of not sticking once the filament does extrude. At worst, you'll cause a backup of melted filament in your hot end, possibly leading to a blockage/clog.

How do I know?
During printing, especially during the first few layers either extremely thin/transparent or no filament will extrude from the nozzle

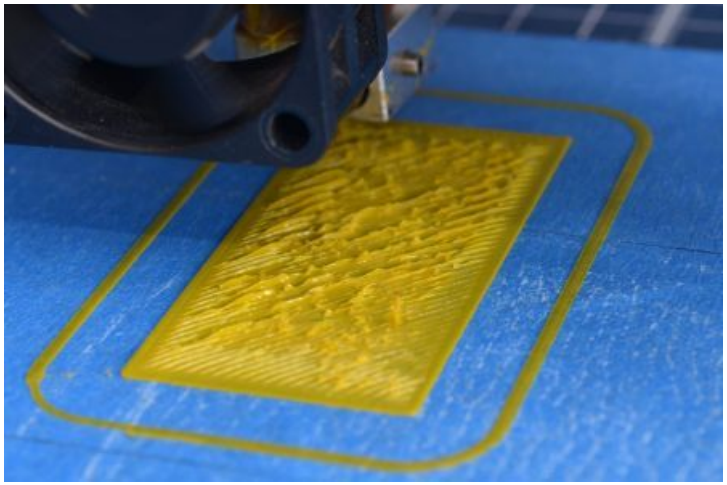
Nozzle is too close cont.

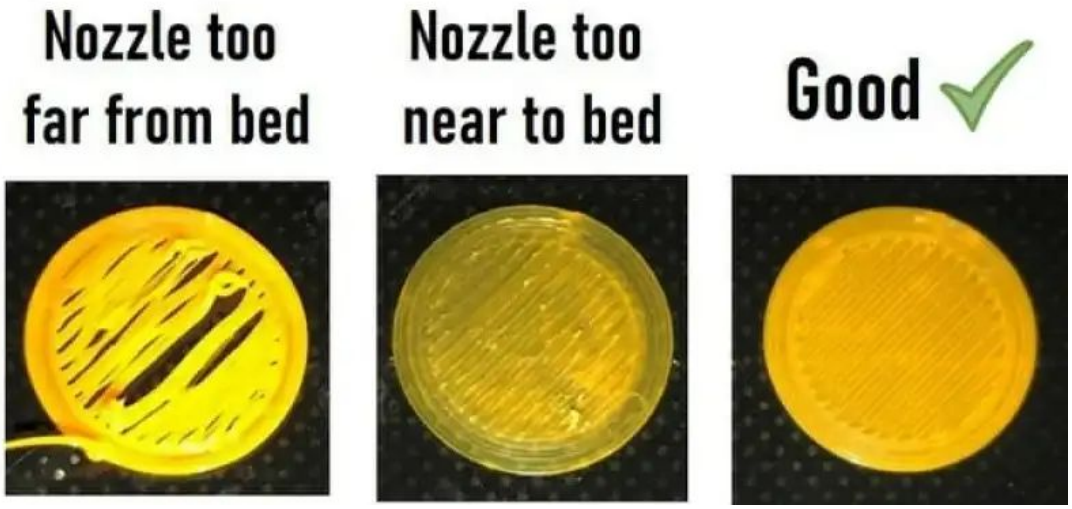
Solutions;

- Adjust the Z-axis
  - Just raising the height of the nozzle slightly can often help. Most 3D printers in their system settings will allow you to set a Z-axis offset. To raise your nozzle away from the print bed you'll need to increase the offset into the positive value. This also works for the reverse, with a negative offset helping to address your prints not sticking to your bed. Be careful though, too high of an offset and it won't stick to the platform

- Lower the print bed
  - Alternatively if your printer allows for it, you can achieve the same effect by lowering your print bed. This is the more troublesome fix though, as it requires you to re-calibrate and level the bed for even prints.

Nozzle is to close cont.





Warping

What is it?
At the base of the model, the print bends upwards until it's no longer level with the print platform. This can also result in horizontal cracks in upper parts and cause your print to come unstuck from the print bed.

What causes it?
Warping is common as it's caused by a natural characteristic of the plastic. As the ABS or PLA filament cools it starts to contract very slightly; the problem of warping arises if the plastic is cooled too quickly.

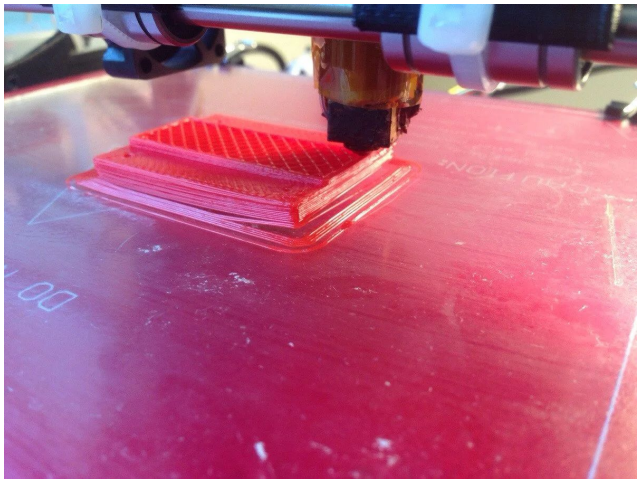
How will I know?
Corners at the base of the print will be lifting

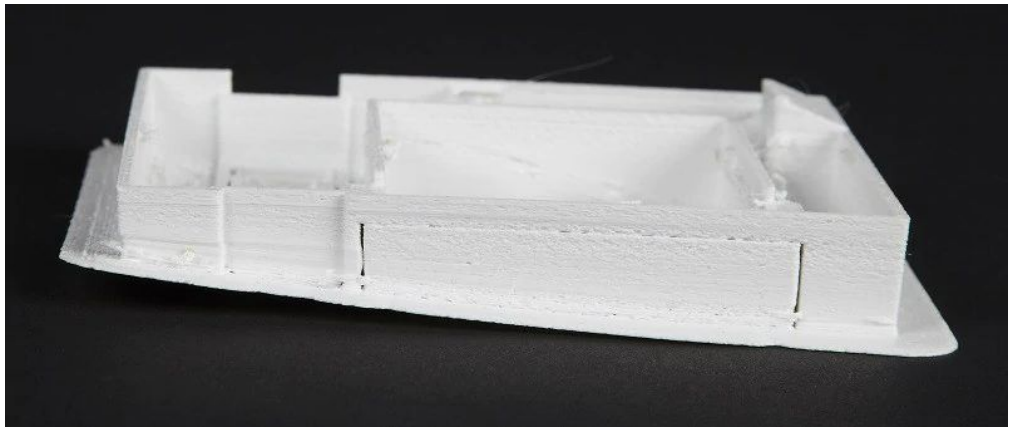
Warping cont.

Solutions:

- Add an/more adhesive to the print bed
  - If you still find your print lifting at the edges then apply a tiny amount of stick glue evenly on the bed to increase adhesion
- Level the print bed
  - Print platform calibration can be another cause, run through the calibration process to check that the bed is level and nozzle height is correct.
- Increase contact
  - Increasing the contact between the model and bed is an easy fix and most print software has the option to add rafts or platforms.
- Adjust advanced temperature settings
  - If all else fails then you'll need to take a look at your advanced print settings both on your printer and in your print software. Try increasing the print bed temperature by increments of 5 degrees. In the slicer software take a look at the fan cooling, this is usually set so the cooling fans switch to full power at a height of around 0.5mm, try extending this to 0.75 to give the base layers a little more time to cool naturally.

Warping cont.





Clogged nozzle

What is it?
You initiate a print job but whatever you try, nothing comes out of the nozzle. Extracting the filament and reinserting doesn't work.

What causes it?
A small piece of filament has been left behind in the nozzle after changing spools, often because the filament has snapped off at the end. When the new filament is loaded, the piece of old filament that is left in the nozzle doesn't allow the new filament to be pushed through.

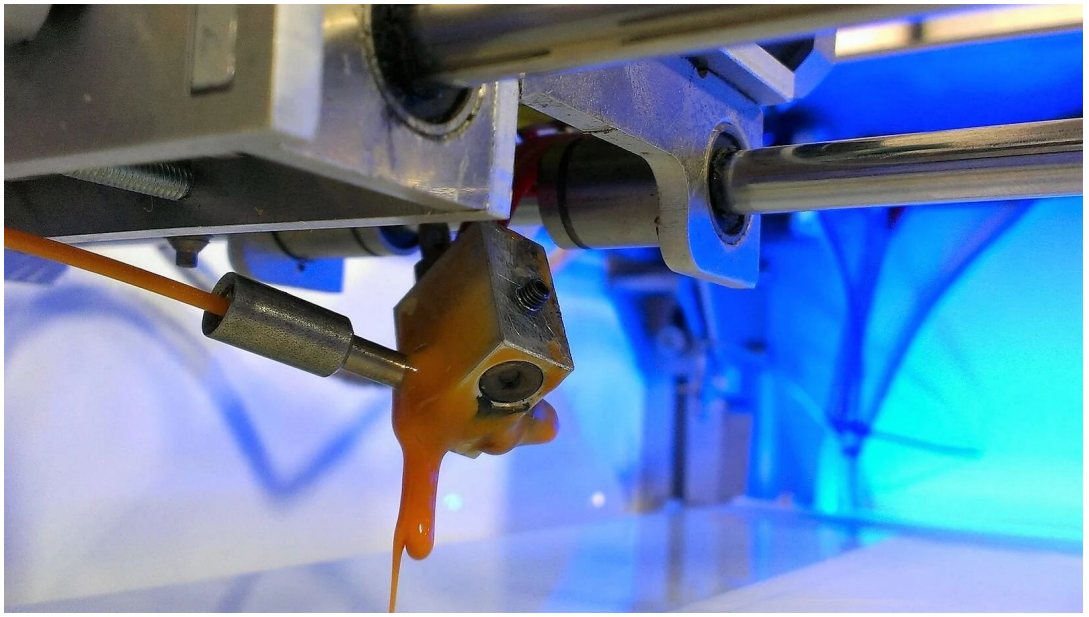
How do I know?
Filament will not extrude even if it manually fed through the nozzle.

Clogged nozzle cont.

Solutions:

- Unblock with a needle
  - If you're lucky then unblocking can be a quick and easy process. Start by removing the filament. Then using your printer's control panel select the "heat up nozzle" setting and increase to the melting point of the stuck filament. Once the nozzle reaches the correct temperature, use a small pin to clear the hole (being careful not to burn your fingers)
- Push the old filament through
  - If you find that the nozzle is still blocked then you may be able to push the filament through with another bit of filament. Start by removing the filament as before and then remove the feeder tube from the print head. Heat up the hot end and then using another piece of filament to push this through from the top to try to force the stuck filament in the nozzle out. Usually if the new filament hasn't succeeded in unblocking then the extra pressure you can exert by hand might just do the job. However don't push to hard as you'll risk bending the horizontal printer rods.
- Disassemble, clear and rebuild the hot end
  - In extreme cases when the nozzle remains blocked, you'll need to do a little surgery and dismantle the hot end. If you've never done this before then it's a good idea to make notes and take photographs so you know where everything fits when you reassemble. Start by removing the filament, then check your printer's manual to see exactly how to dismantle the hot end.
- Get aggressive with it
  - For nozzles that use PLA, a torch can be used to burn or melt out any excess or clogged filament
  - Nozzles that use ABS can be soaked in acetone to melt out excess material

Clogged nozzle cont.



Elephant's foot

What is it?
Elephant's foot is a 3D printing term that refers to the outward bulge of a model's base. Simply put, it is when the print bows or curves out at the bottom

What causes it?
This usually happens when the weight of the model is pressing down on its base before it cools back to solid

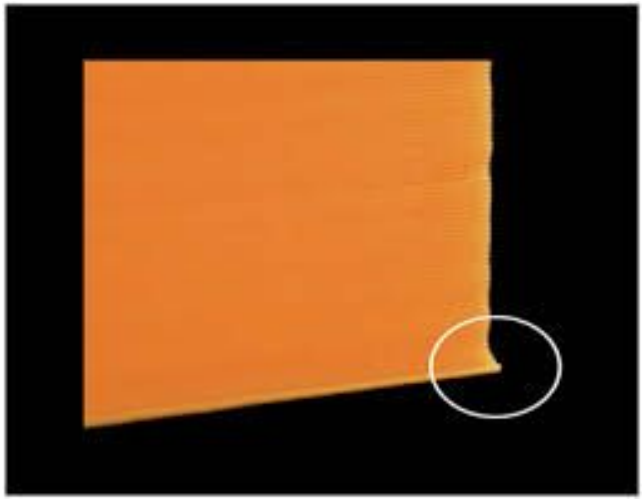
How do I know?
Prints will curve out towards the bottom of the print

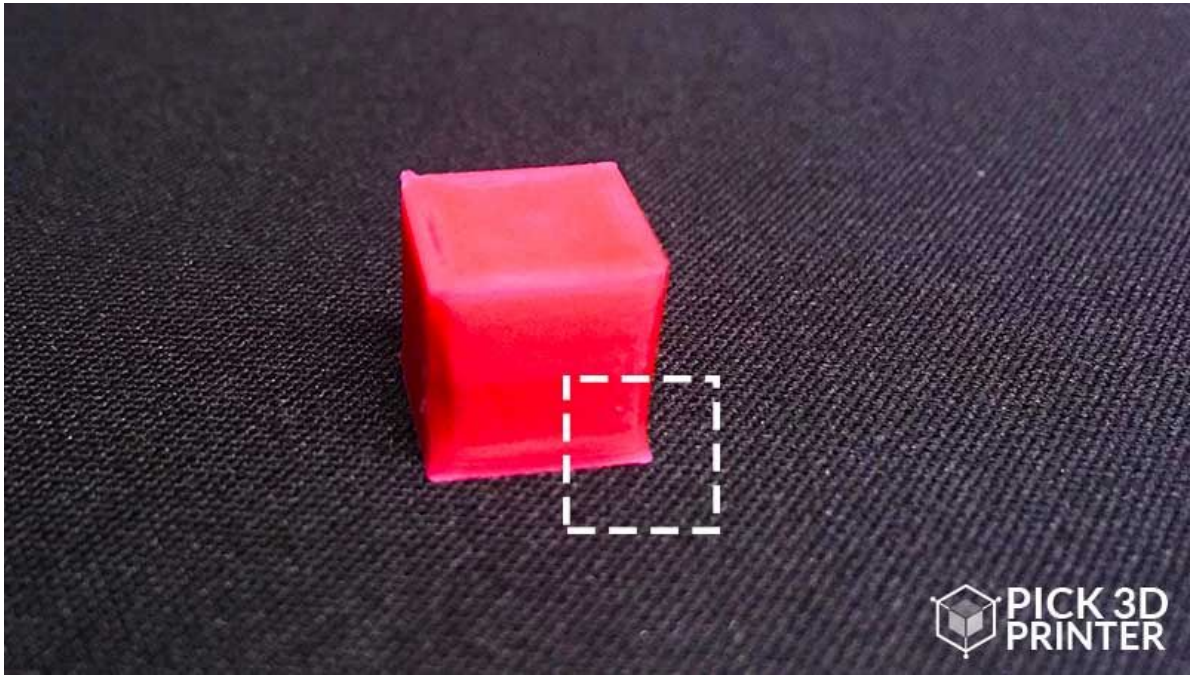
Elephant's foot cont.

Solutions

- Another 3D printing troubleshooting tip you can use is to ensure that the base layers get enough coolness so as to be able to support the top structure. You need to ensure that the cooling is just the right amount as too much cooling will cause the base layers to get warped. You might find this part tricky but the best way to go about this 3D printing troubleshooting process is to lower the print platform's temperature by five-degree intervals to about plus or minus twenty degrees of the temperature recommended. If you have your Bottom/Top thickness set to 0.6mm, you can start the fan at a height that's slightly lower than that.

Elephant's foot cont.





Stringing

What is it?
There are unsightly strings of plastic between parts of the model

What causes it?
When the print head moves over an open area (otherwise known as travel move), some filament has dripped from the nozzle

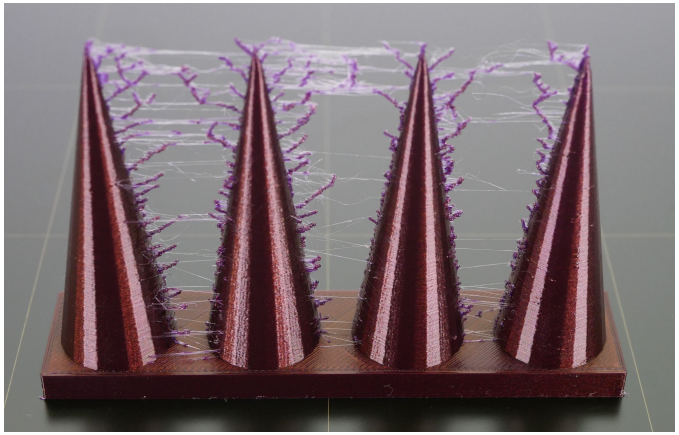
How do I know?
There are thin spider web looking strands on the print

Stringing cont.

Solutions:

- Enable retraction
  - Retraction is an important factor when it comes to quality of finish and can be enabled through most slicing software. Its function is pretty simple and works by retracting the filament back into the nozzle before the head moves. The idea is that it avoids molten filament from trailing behind the head creating thin strings in its wake. Most applications such as Cura offer a one click activation option. This uses a set of default parameters and for the most part is perfectly adequate, but for fine tuning there are customizable options that give greater control. Adjusting the minimum travel of the head before retraction is activated, for example.
- Minimum travel
  - Reducing the minimum travel is usually the quickest fix for stringing if the standard retraction isn't doing the job. Drop the value in 0.5mm increments until the stringing has stopped.
- Just cut them off
  - This isn't the most elegant of solutions but simply taking a scalpel to the strings is quite often the quickest and easiest solution.

Stringing cont.



Bad bowden coupler

What is it?
The coupler/fitting that holds the bowden tube in place is either broken or worn to the point of failure.

What causes it?
General use over time, improper use and application, general wear and tear.

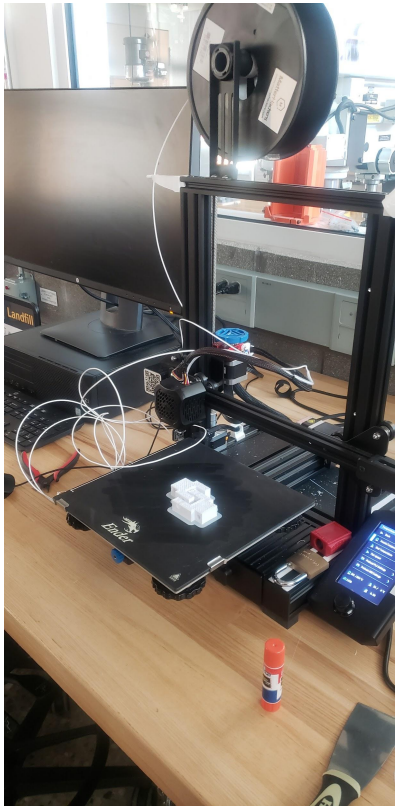
How do I know?
The coupler will not be able to hold onto the bowden tube or the tube has been pushed out of the fitting during a job.

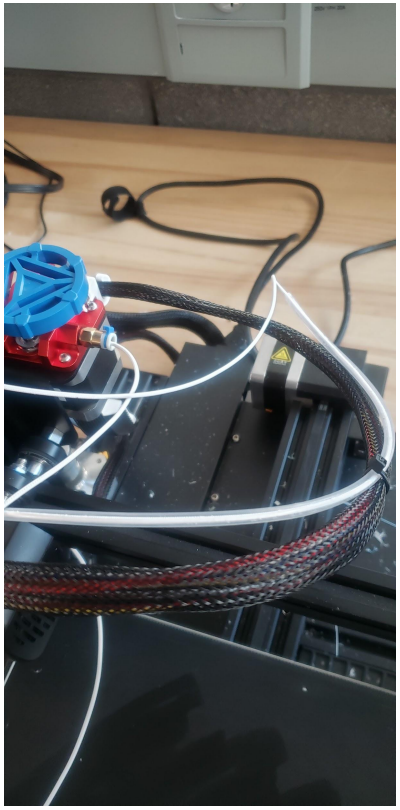
Bad bowden coupler cont.

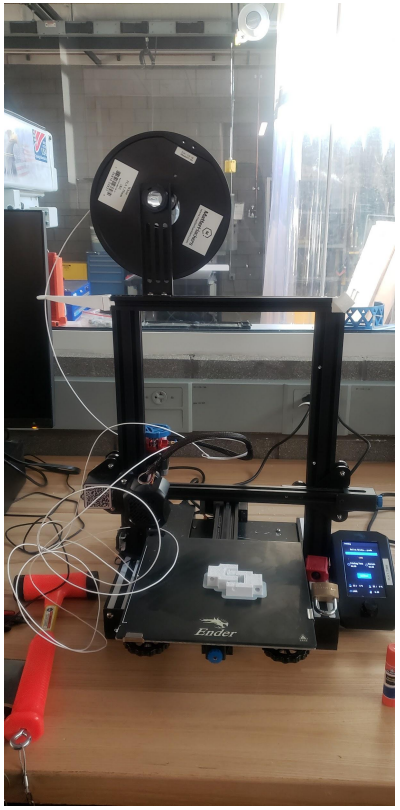
Solutions:

- Remove and replace
  - Remove the coupler on the side that failed. Find/purchase a spare and replace. Install the bowden tube as normal.

Bad bowden coupler cont.







Raft silhouetting (SLA resin)

What is it?

“Raft silhouetting” describes a print defect in which the early layers adhere to the build platform, creating a silhouette of the raft or initial layers. After these initial layers that create the raft silhouette, the remainder of the part does not print.

What causes it?

Contamination, dust, or debris on the printer’s optical surfaces can diffuse or weaken laser light, causing a print failure where only the early layers of the print adhere to the build platform.

How will I know?

- Only the first few layers adhere to the build platform.

- The rest of the part is missing from the build platform, and may be present as cured resin in the tank bottom or absent entirely.

Raft silhouetting cont.

Solutions:

- Check the resin tank
  - Any damage or debris in or on the resin tank can block the laser at that location, preventing the resin directly above it from curing. Inspect the inside and underside of the tank for dirt, dust, and/or fingerprints. The bottom of the tank must be clean for the best print results.
    i. Inspect the resin tank for debris with the tank tool.
    ii. Remove any failed prints and cured resin using the tank tool.
    iii. Alternatively, remove any failed prints and cured resin by printing a cleaning mesh.
    iv. Filter the resin to remove any contamination or cured resin.
    v. Inspect the flexible film at the bottom of the resin tank for scratches or wear. Excessive wear means the resin tank needs to be replaced.
    vi. Inspect the underside of the flexible film for dirt, dust, or fingerprints. Clean it with Formlabs recommended steps and supplies.
- Check the optical window
  - Any contamination, dust, or debris on the printer's optical surfaces can cause the laser light to diffuse or weaken, resulting in a print failure. Contamination is most likely to occur on an exposed optical surface in the printer.
    i. With the printer plugged in and powered on, tap the wrench icon on the touchscreen to enter the Settings menu, then tap Maintenance, then tap LPU Replacement. The LPU housing moves from its storage position on the right side of the printer into its maintenance position.
    ii. When prompted by the touchscreen, disconnect the power cable before continuing with maintenance.
    iii. Inspect the optical window for contamination.
    iv. Follow the instructions for cleaning the glass optical window.
    v. Form 3/Form 3B
    vi. Form 3L/Form 3BL
    vii. If streaks are difficult to remove, contact Formlabs Support or your authorized reseller.

Raft silhouetting cont.

No adherence (SLA resin)

What is it?

“Non-adherence” describes a print defect in which prints are either partially or completely detached from the build platform. This occurs when the initial layers of a print fail to adhere to the build platform.

What causes it?

Printing without a raft, when the part has no suitable flat surface to grip the build platform. The first layer of the part on the build platform is too small to withstand peel forces. Debris/clouding/damage in the resin tank weakening the printer’s laser. There could be contaminated optical surfaces weakening the printer’s laser. There is excessive space between the build platform and the elastic layer or the flexible film in the resin tank

How will I know?

- The print is missing from the build platform and there is a flat area of cured resin on the bottom of the resin tank.
- The print’s raft partially separates from the build platform.
- The print has formed partially, but has fallen into the tank during the print, leaving nothing on the build platform.

No adherence cont.

Solutions

- Check the model's supports in PreForm
  - Printing with a raft helps parts adhere to the build platform. Generate supports in PreForm when setting up your model. Check if the print detached at the point of contact between the model and the supports.
- Check the model's orientation in PreForm
  - If the surface area of the initial layers is too small, the suction from the peel process may pull the part off the build platform. Check the model in PreForm for orientation issues:
    1. Orient the model so that the surface area in contact with the build platform is not too small proportional to your part.
       - If using a raft, orient your part on a tilt, which is the recommended best practice.
       - If printing directly on the build platform, make sure that the part has a large, flat side in direct contact with the platform.
- Check the resin tank
  - Any damage or debris in or on the resin tank can block the laser at that location, preventing the resin directly above it from curing. Inspect the inside and underside of the tank for dirt, dust, and/or fingerprints. The bottom of the tank must be clean for the best print results.

Non adherence cont.

Delamination

What is it?
"Delamination" describes a print defect in which the cured layers of a print separate from one another. Delamination can occur on any type of 3D printer, though the causes are unique to each print process.

What causes it?
Any of the following factors, or a combination of them, can cause delamination in prints:

- Model orientation, layout, or support issues
- A print that has been paused for an hour or longer
- An older resin tank
- Loose build platform
- Contaminated optical surfaces

How will I know?

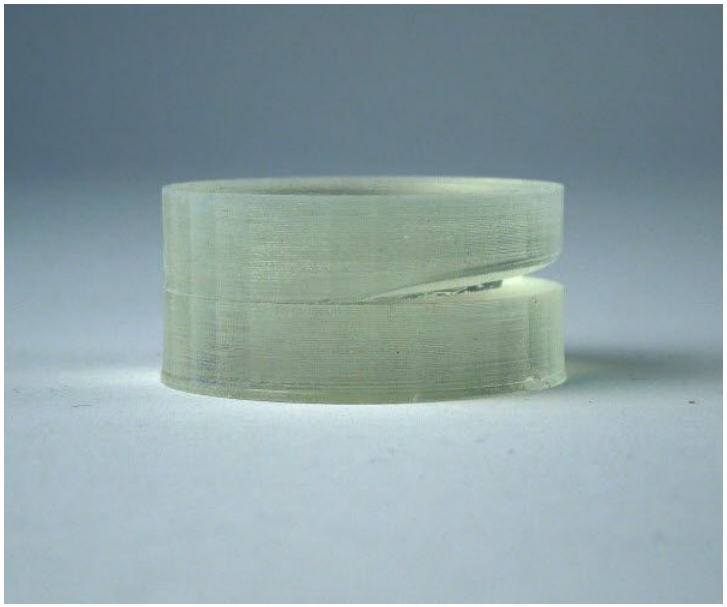
- Peeling or separation between printed layers
- Pieces of cured parts floating in the resin tank

Delamination cont.

Solutions

- Check the model in PreForm
  - Lateral forces on the part during the print can disrupt the molecular bonding between layers and separate them. This separation occurs for two reasons:
  - The part has a significant change in surface area between subsequent layers. Layers with large surface areas resist print forces better than layers with small surface areas, so a larger layer may separate from a smaller layer during the print. Reorient the part in PreForm to reduce the difference in surface area between neighboring layers.
  - Sections of the part are free-hanging or lack adequate supports. In PreForm, look for areas shaded in red, and add supports to these areas.
- Check for pauses in printing
  - Extended pauses during the print can lead to delamination after the print resumes.
  - Resolve any errors displayed on the touchscreen that pause the print.
  - Enable notifications on Dashboard to receive text or email alerts for print interruptions as soon as they occur.
- Check the resin tank
  - Any damage or debris in or on the resin tank can block the laser at that location, preventing the resin directly above it from curing. Inspect the inside and underside of the tank for dirt, dust, and/or fingerprints. The bottom of the tank must be clean for the best print results.
    i. Inspect the resin tank for debris with the tank tool.
    ii. Remove any failed prints and cured resin using the tank tool.
    iii. Alternatively, remove any failed prints and cured resin by printing a cleaning mesh.
    iv. Filter the resin to remove any contamination or cured resin.
    v. Inspect the flexible film at the bottom of the resin tank for scratches or wear. Excessive wear means the resin tank needs to be replaced.
    vi. Inspect the underside of the flexible film for dirt, dust, or fingerprints. Clean it with Formlabs recommended steps and supplies.
- Check the optical window
  - Any contamination, dust, or debris on the printer’s optical surfaces can cause the laser light to diffuse or weaken, resulting in a print failure. Contamination is most likely to occur on an exposed optical surface in the printer.

Delamination cont.



Cupping/blowout

What is it?

“Cupping” is a concern unique to upside-down stereolithography (SLA) printing. A print failure caused by unresolved cupping is often referred to as a “blowout.” This occurs when a hollow or convex portion of a part acts as a suction cup and traps air while printing. As the build platform pulls away from the tank during the peel process, the empty space within the cup increases. This reduces the air pressure within the cup to create a pressure differential that pushes inward on the surrounding cup walls. If the walls are too thin and unable to withstand the pressure, then they buckle inward to equalize the pressure. Depending on the location of the rupture, the part can potentially continue to print without further errors. The risk of a blowout is dependent on the resin type, size of the cup, shape of the model, and thickness of the cup wall. Small cups, or cups with thicker walls, may print successfully. Formlabs recommends adding a drainage hole and orienting the model in PreForm to minimize cupping.

What causes it?

Any of the following factors, or a combination of them, can cause cupping blowout in prints:

- No drainage holes in hollow or convex parts
- Model orientation issues
- Model printed without resolving cups highlighted in PreForm

How will I know?

- A “hole” in the wall of your part
- A large, ragged “rupture” in your part
- Poor surface finish on cup surfaces

Cupping/blowout cont.

Solutions

- Enable Show Cups and inspect the model with the Slicer tool, or add a drainage hole to the cup in your preferred 3D design package before returning the model to PreForm.

Cupping/blowout cont.

Underdeveloped features

What is it?

An "undeveloped feature" is a print defect in which a void of missing material forms within a print and widens as the print builds away from the build platform. These voids can look like a crater and frequently exhibit sharp edges and rough surfaces. In some cases, it can look like the missing material exploded" outwards from the build platform. This type of defect is a print failure unique to stereolithography (SLA). If a piece of the print sticks to the bottom of the resin tank, that piece partially blocks the laser from reaching the next layer of the print. The piece stuck to the tank expands with each additional cured layer and prevents other features from forming.

What causes it?

Any of the following factors, or a combination of them, can cause prints to have undeveloped features:

- Debris or damage in resin tanks
- Clouding in Form 2 standard resin tanks
- Model orientation, layout, or support issues
- Contaminated optical surfaces

How will I know?

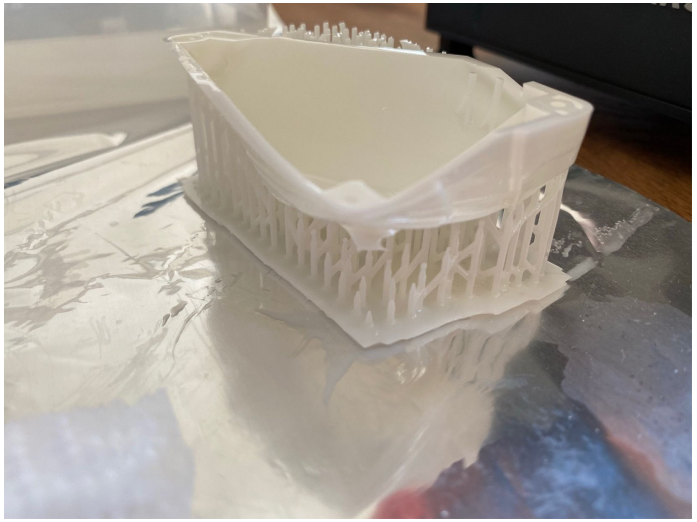
- A void within a print that expands outwards from the earliest layer to the most recent layer, usually with sharp edges and rough surfaces like a crater
- A layer of cured resin on the bottom of the resin tank

Underdeveloped features cont.

Solutions

- Check the model in PreForm
  - Inspect the printed part to determine where the undeveloped feature started. In PreForm, look for the area on the part where the outward-expanding explosion is closest to the build platform.
  - Examine individual layers of the model with the vertical Slicer tool on the right side of PreForm. Pay close attention to the following areas as the possible origin for the undeveloped feature:
  - Look for any areas on the model highlighted in red. These highlighted areas indicate printability concerns and may need additional support.
  - Even if there is no red shading, consider adding extra supports to any areas that failed during printing.
  - In particular, look for local minima and islands that need additional support.
- Check the resin tank
  - Any damage or debris in or on the resin tank can block the laser at that location, preventing the resin directly above it from curing. Inspect the inside and underside of the tank for dirt, dust, and/or fingerprints. The bottom of the tank must be clean for the best print results.
    - Inspect the resin tank for debris with the tank tool.
    - Remove any failed prints and cured resin using the tank tool.
    - Alternatively, remove any failed prints and cured resin by printing a cleaning mesh.
    - Filter the resin to remove any contamination or cured resin.
    - Inspect the flexible film at the bottom of the resin tank for scratches or wear. Excessive wear means the resin tank needs to be replaced.
    - Inspect the underside of the flexible film for dirt, dust, or fingerprints. Clean it with Formlabs recommended steps and supplies.
- Check the optical window
  - Any contamination, dust, or debris on the printer's optical surfaces can cause the laser light to diffuse or weaken, resulting in a print failure. Contamination is most likely to occur on an exposed optical surface in the printer.
    - With the printer plugged in and powered on, tap the wrench icon on the touchscreen to enter the Settings menu, then tap Maintenance, then tap LPU Replacement. The LPU housing moves from its storage position on the right side of the printer into its maintenance position.
    - When prompted by the touchscreen, disconnect the power cable before continuing with maintenance.
    - Inspect the optical window for contamination.
    - Follow the instructions for cleaning the glass optical window.
    - If streaks are difficult to remove, contact Formlabs Support or your authorized reseller.

Underdeveloped features cont.



Pinholes or cuts

What are they?
A “pinhole” or “cut” describes a print defect in which a small region of the part does not form. This void can take the form of a small hole, a tunnel that runs through the part, or a slit that cuts through the part. If other sections of the part would have been supported directly by the missing material, those sections may also fail to cure to the part.

What causes them?

- Debris or scratches on the bottom surface of the resin tank
- Debris or scratches on the optical window or another optical surface

If, while curing a layer, the laser is blocked before it reaches the resin, that portion of the part will not form properly. If the obstruction is on a stationary surface, a hole may develop directly above that point. If the blockage is on a moving surface, a linear cut may develop instead, following the path of the blockage.

How will I know?

- A void that passes through the entire print, including the raft (if one is used). The void may take the form of a small hole in one portion of the part or a linear slit that extends across the part.
- A hole or cut may lead to one or more undeveloped features,depending on the geometry and orientation of the part.

Pinholes or cuts cont.

Solutions

- Check the resin tank
  - Any damage or debris in or on the resin tank can block the laser at that location, preventing the resin directly above it from curing. Inspect the inside and underside of the tank for dirt, dust, and/or fingerprints. The bottom of the tank must be clean for the best print results. Any debris, clouding, or damage to the resin tank can diffuse the laser light and limit the printer's ability to fully cure the resin.
    - Inspect the resin tank for debris with the tank tool.
    - Remove any failed prints and cured resin using the tank tool.
    - Alternatively, remove any failed prints and cured resin by printing a cleaning mesh.
    - Filter the resin to remove any contamination or cured resin.
    - Inspect the flexible film at the bottom of the resin tank for scratches or wear. Excessive wear means the resin tank needs to be replaced.
    - Inspect the underside of the flexible film for dirt, dust, or fingerprints. Clean it with Formlabs recommended steps and supplies.
- Check the optical window
  - Any contamination, debris, or scratches on an optical surface can also block the laser at that location, preventing the resin directly above it from curing. Contamination is most likely to occur on an exposed optical surface in the printer. The optical surface most likely to be contaminated is the glass optical window, which sits atop the Light Processing Unit (LPU).
    1. With the printer plugged in and powered on, tap the wrench icon on the touchscreen to enter the Settings menu, then tap Maintenance, then tap LPU Replacement. The LPU housing moves from its storage position on the right side of the printer into its maintenance position.
    2. When prompted by the touchscreen, disconnect the power cable before continuing with maintenance.
    3. Inspect the optical window for contamination.
    4. Follow the instructions for cleaning the glass optical window.
    5. If streaks are difficult to remove, contact Formlabs Support or your

Pinholes or cuts cont.

Ragging

What is it?

“Ragging” describes a print defect in which cured or partially cured resin forms thin, shelf-like structures that hang horizontally from a print. These structures, also called “flaps,” can detach from the part during printing.

Once these flaps are floating in the resin tank, they can block the laser’s path and cause print failures in subsequent layers. Ragging occurs when diffused laser light spreads the cure area beyond the appropriate limits for each layer outline.

What causes it?

- Expired resin
- Debris/clouding/damage in the resin tank
- Contaminated optical surfaces
- Restricted resin flow due to improper model orientation or overly dense supports

How will I know?

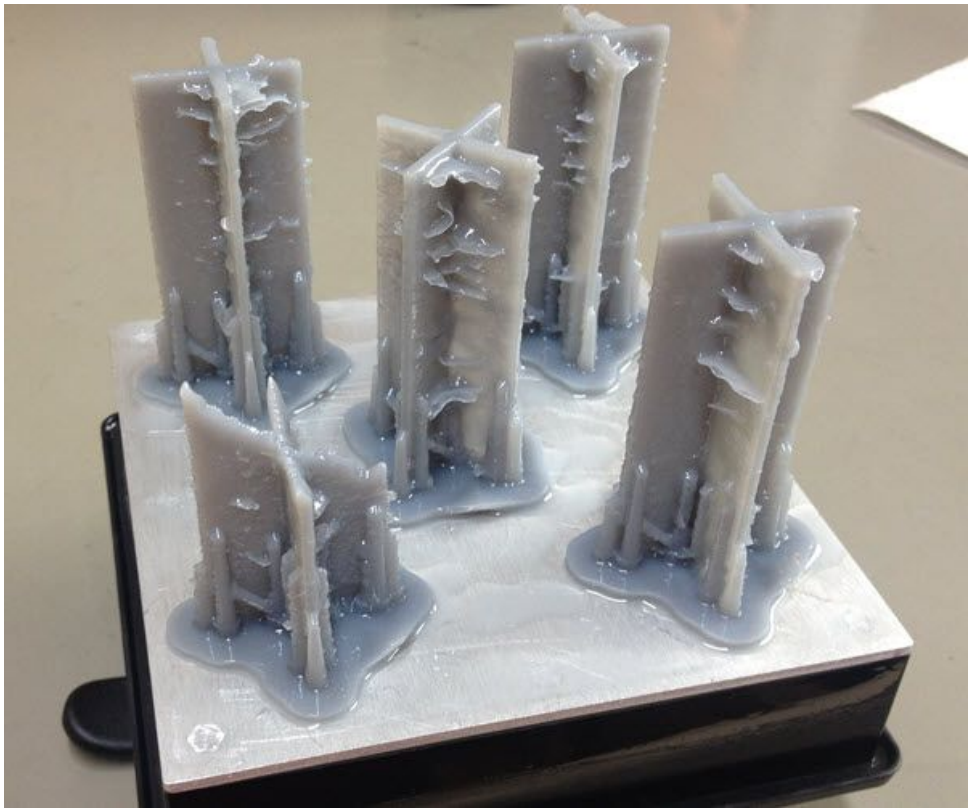
- Flaps of cured or partially cured resin on the print that degrade the overall cosmetic aspects of the print
- Thin flakes of cured or partially cured resin from the print floating in the resin tank, which may or may not cause a print failure

Ragging cont.

Solutions

- Check the resin's expiration date
  - Parts printed with expired resin may have poor surface finish and exhibit rashing and ragging. Formlabs resins last for 12 to 24 months in the resin cartridge, depending on the resin type. Learn more about resin lifetime and storage to determine if your resin has expired.
- Check the resin tank
  - Any damage or debris in or on the resin tank can block the laser at that location, preventing the resin directly above it from curing. Inspect the inside and underside of the tank for dirt, dust, and/or fingerprints. The bottom of the tank must be clean for the best print results.
    - Inspect the resin tank for debris with the tank tool.
    - Remove any failed prints and cured resin using the tank tool.
    - Alternatively, remove any failed prints and cured resin by printing a cleaning mesh.
    - Filter the resin to remove any contamination or cured resin.
    - Inspect the flexible film at the bottom of the resin tank for scratches or wear. Excessive wear means the resin tank needs to be replaced.
    - Inspect the underside of the flexible film for dirt, dust, or fingerprints. Clean it with Formlabs recommended steps and supplies
- Check the optical window
  - Any contamination, dust, or debris on the printer's optical surfaces can cause the laser light to diffuse or weaken, resulting in a print failure. Contamination is most likely to occur on an exposed optical surface in the printer. The optical surface most likely to be contaminated is the glass optical window, which sits atop the Light Processing Unit (LPU).
    1. With the printer plugged in and powered on, tap the wrench icon on the touchscreen to enter the Settings menu, then tap Maintenance, then tap LPU Replacement. The LPU housing moves from its storage position on the right side of the printer into its maintenance position.
    2. When prompted by the touchscreen, disconnect the power cable before continuing with maintenance.
    3. Inspect the optical window for contamination.
    4. Follow the instructions for cleaning the glass optical window.
- Check for restricted resin flow

Ragging cont.



Rashing

What is it?
"Rashing" describes a print defect in which a rough surface texture forms on one or both sides of an otherwise successful print.

What causes it?
- Expired resin
- Debris/clouding/damage in the resin tank
- Contaminated optical surfaces
- Restricted resin flow due to improper model orientation or overly dense supports

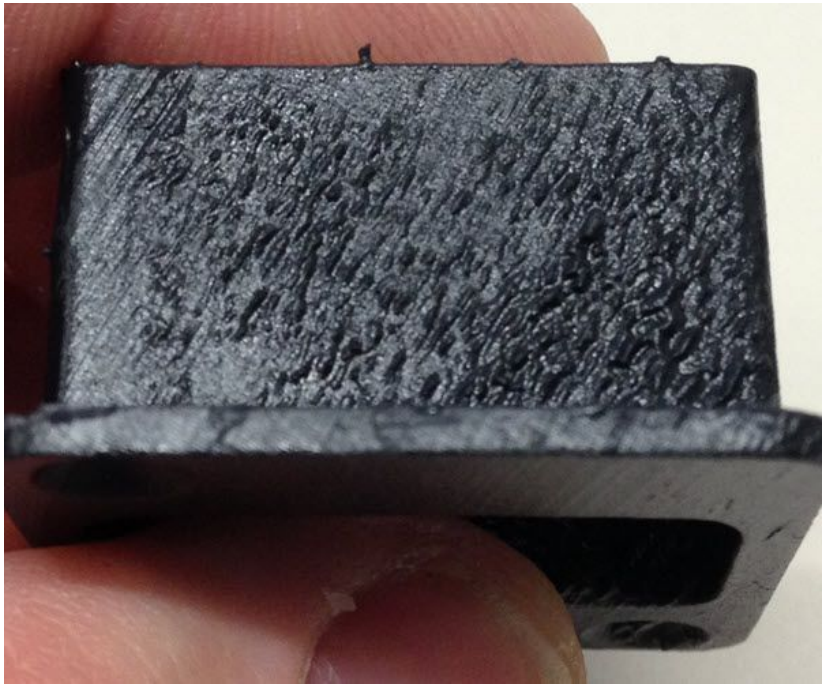
How will I know?
For prints created on Formlabs printers, rashing produces an uneven or bumpy surface finish on one or both sides of a part.

Rashing cont.

Solutions

- Check the resin's expiration date
  - Parts printed with expired resin may have poor surface finish and exhibit ragging and rashing. Formlabs resins last for 12 to 24 months in the resin cartridge, depending on the resin type. Learn more about resin lifetime and storage to determine if your resin has expired.
- Check the resin tank
  - Any damage or debris in or on the resin tank can block the laser at that location, preventing the resin directly above it from curing. Inspect the inside and underside of the tank for dirt, dust, and/or fingerprints. The bottom of the tank must be clean for the best print results.
    - Inspect the resin tank for debris with the tank tool.
    - Remove any failed prints and cured resin using the tank tool.
    - Alternatively, remove any failed prints and cured resin by printing a cleaning mesh.
    - Filter the resin to remove any contamination or cured resin.
    - Inspect the flexible film at the bottom of the resin tank for scratches or wear. Excessive wear means the resin tank
- Check the optical window
  - Any contamination, dust, or debris on the printer's optical surfaces can cause the laser light to diffuse or weaken, resulting in a print failure. Contamination is most likely to occur on an exposed optical surface in the printer.
- Check for restricted resin flow

Rashing cont.



Over Compression

What is it?
“Over compression” describes a print defect in which reduced space between the build platform and resin tank causes the initial layers to be very thin. The rest of the part prints, but the initial layers are flattened and may be very difficult to remove from the build platform.

What causes it?
Over compression is caused by insufficient space between the build platform and the elastic layer or the flexible film in the resin tank.

How will I know?

- The raft is compressed and thinner than expected.
- Parts are especially difficult to remove.
- The raft is completely flat and supports are significantly shorter than expected.

Over compression cont.

Solutions

- Contact support