



Q: Why does 3DP request the source file, .stl, and .gcode for failed prints?

A: So we can replicate the problem here at the factory, allowing us to effectively troubleshoot and determine root cause—which then is provided back to the customer.

### PRINT QUALITY

1. Thick, wide beads of filament
2. Bulging, oozing beads of filament
3. Oversized, not to specification

### PROBLEM

A combination of settings:

- Nozzle temperature too high
- Print speed too slow
- Flow rate (extrusion multiplier) too high pushing too much material through the nozzle

### CORRECTIVE ACTION

- Reduce the temperature of the nozzle. Optimal temperature can be verified by raising the Z axis, heating the nozzle, extruding a length of filament. The bead should resemble a smooth lasso.
- Increase the print speed to 2100 mm/min. Speeds too slow allow the material to get too hot and ooze.
- Decreased the flow rate (extrusion multiplier)—this reduces the amount of material being pushed through the nozzle.



Zoomed View

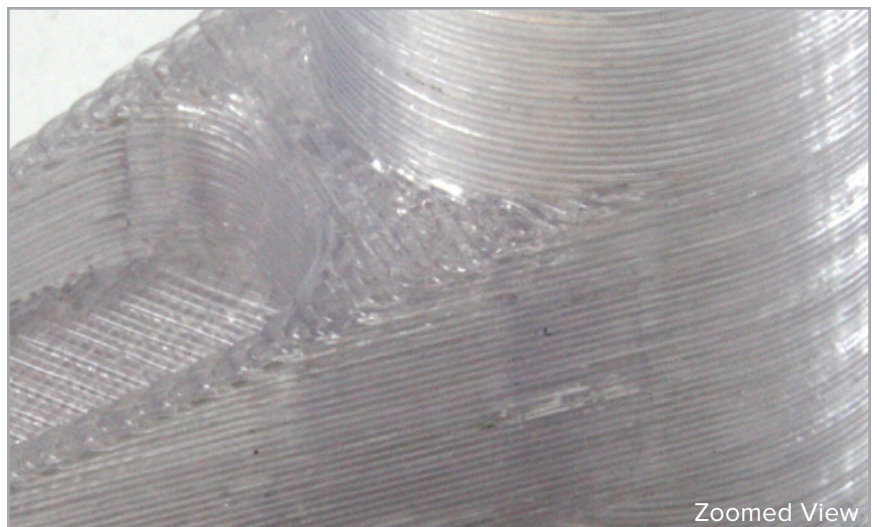
**Failed Print:** The layers are not smooth as they should be. Instead, each bead of filament is bulging and oversized.



Failed Print



Fixed Print



Zoomed View

**Fixed Print:** Smooth, even layers are essential to a successful print. The material is not bulging between layers and is smooth on the top.





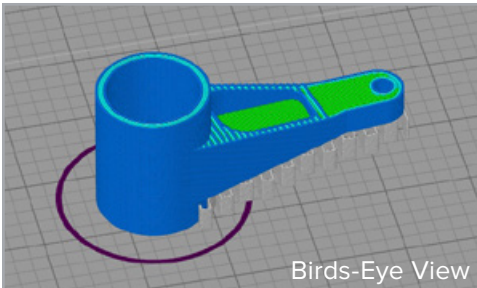
**Rule of Thumb:** Only make one adjustment at a time, so you can see the result of the change.



Failed Print

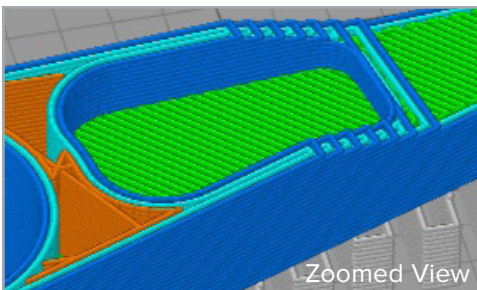


Fixed Print



Birds-Eye View

**Model, Sliced:** The previewed file shows the skirt, support structures, and color-codes print features such as outside perimeters, infill, and top/ bottom solid layers.



Zoomed View

**Model, Sliced:** Look carefully at each layer of the file in the software to identify errors in the model or inappropriate slice settings.

PRINT PROCESS SETTINGS	BEFORE FIX BAD PRINT	AFTER FIX GOOD PRINT
Material Type	PLA	PLA
Bed Temperature	70° C	70° C
Nozzle Size	0.8 mm	0.8 mm
Nozzle Temperature	235° C	210° C
Flow Rate (Extrusion Multiplier)	1.25 (125%)	0.80 (80%)
Extrusion Width	1 mm	0.83 mm
Print Speed	800 mm/min	2100 mm/min
Layer Height	.55	.55
Number of Perimeters	2	2
Top Layers	4	4
Bottom Layers	3	3
Infill Percentage	10	10
Support Structures	yes 25%	yes 25%

### KEY POINTS

Use this case study to visually compare print problems you are experiencing. Make one adjustment at a time. Fine tuning your print process settings can be the difference between success and failure with 3D printing.

### OTHER NOTES

**Elephants Foot**—when the lowest layers of a print flare out—this is caused by the nozzle being too close to the build area; or the first layer width set too high. Increase the Z-Gap or reduce the first layer width to achieve a first layer that adheres but doesn't flare.

