



Q: What are my options for using support structures?

A: Supports can be printed using soluble material or the same material as the printed part. The latter requires settings which allow the supports to break away.

PRINT QUALITY

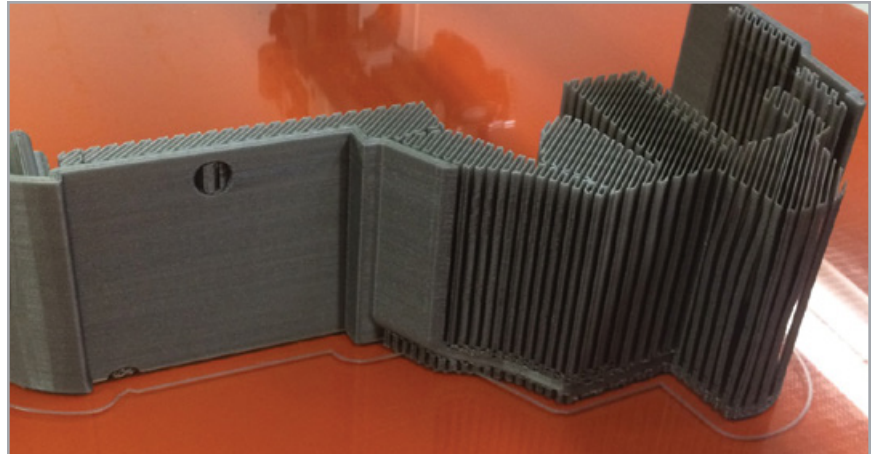
1. Thin walls of the model
2. Steep overhangs require supports
3. Support structure settings did not allow for clean removal—damaging the part in some areas

PROBLEM

- Outside perimeters and infill settings created a weak, thin part
- Support structure settings were too dense, without enough separation from the part

CORRECTIVE ACTION

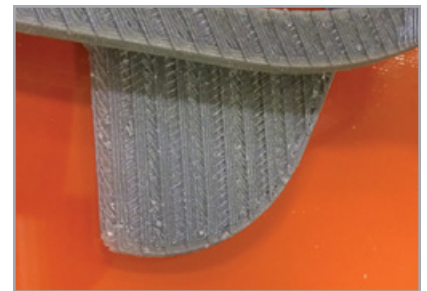
- Thin wall adjustments:
 - Increased the number of outside perimeters to match the intended wall width
 - Increased infill percentage to 100%, creating a solid thin part
- Support structure adjustments:
 - Used 2 dense support layers, with 55% density
 - Utilized 1 upper and 1 lower separation layer to aid in the removal of support structures



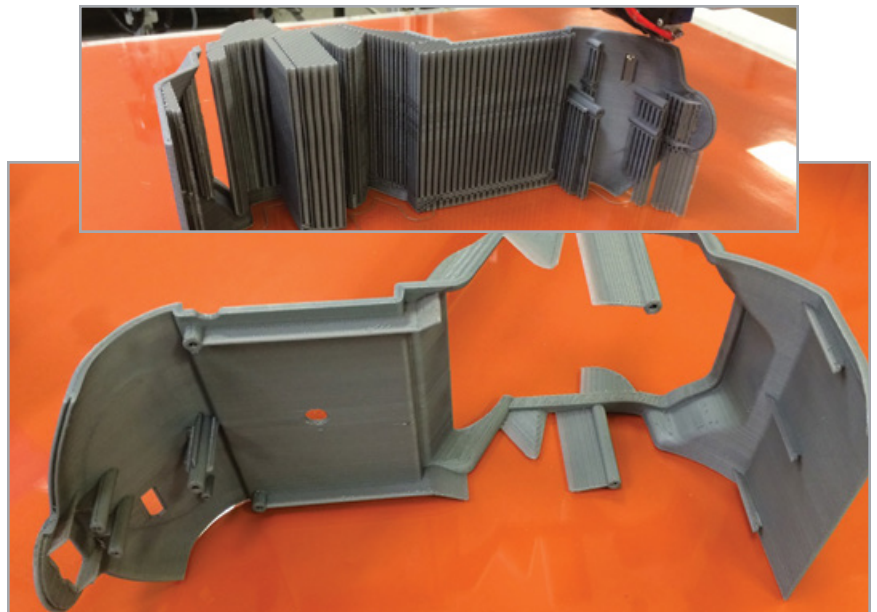
Failed Print: The model walls were thin and the part required substantial support structures. The process settings made the walls extremely fragile and the supports did not cleanly break from the part.



Failed Print



Fixed Print

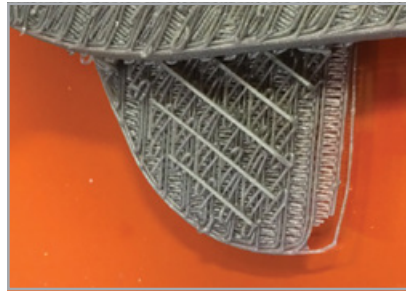


Fixed Print: Successful print with strong outside walls and little sign of scarring where the break-away support structures were removed.

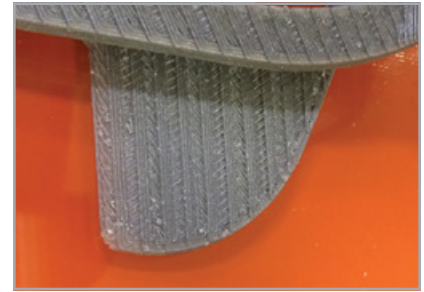
CASE STUDY PROBLEM | SOLUTION



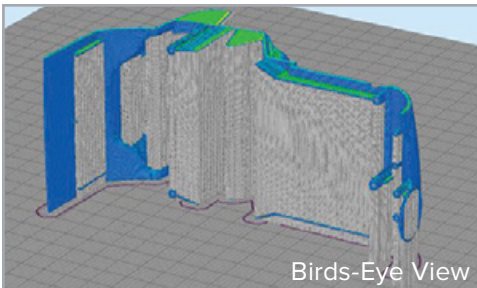
Rule of Thumb: Only make one adjustment at a time, so you can see the result of the change.



Failed Print

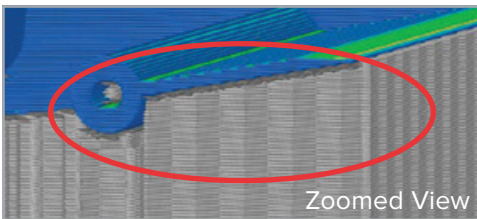


Fixed Print



Birds-Eye View

Model, Sliced: Supports can be removed without damaging the part—with the use of dense support and separation layers.



Zoomed View

Model, Sliced: Dense support layers create a 'shelf' for the part to print on—providing a better surface finish.

KEY POINTS

Thin wall parts can be challenging. Consider the actual wall thickness of the model when setting the number of outside perimeters.

Using separation layers in your support process settings can eliminate damage often caused by support structure removal. Additionally, adding one or two dense support layers can increase the quality of the surface finish of the printed part.

PRINT PROCESS SETTINGS	BEFORE FIX BAD PRINT	AFTER FIX GOOD PRINT
Material Type	PLA	PLA
Bed Temperature	70° C	70° C
Nozzle Size	0.6 mm	0.6 mm
Nozzle Temperature	210° C	210° C
Flow Rate (Extrusion Multiplier)	1.0 (100%)	1.0 (100%)
Extrusion Width	0.72 mm	0.62 mm
Print Speed	3600 mm/min	3600 mm/min
Layer Height	0.40	0.36
Number of Perimeters	2	3
Top Layers	3	3
Number of Bottom Layers	3	3
Infill Percentage	18	100
Support Structures	Yes 30%	Yes 30%
Dense Support Layers	1	2
Separation Layers	0	1

OTHER NOTES

Gap Fill settings within Simplify3D provide assistance when dealing with thin wall models. In this example, the gap fill produced visual resonance and a rough surface finish in the printed part (see image to the right). This was resolved by increasing the number of perimeters to match the wall thickness, instead of utilizing the gap fill function.

