



Q: Does Simplify 3D allow for easy break-away support?

A: Yes. Simplify 3D allows for a separation distance to be set on the x, y or z axis.

PRINT QUALITY

The support material is adhered too strongly to the desired part. It took about 20 minutes of work to aggressively remove it with a screw driver and needle nose pliers, the print is scarred badly and may not be usable.

PROBLEM

The separation layers are not set correctly. The support material will print directly in contact with the solid layers.

- Horizontal Offset from Part: 0.00mm
- Upper Vertical Separation Layers: 0
- Lower Vertical Separation Layers: 0

CORRECTIVE ACTION

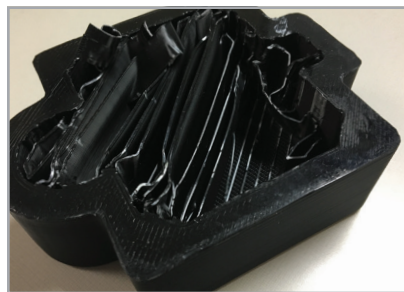
Adjusting the separation layers and distances allows for easy removal of the support material and minimizes scarring.

- Horizontal Offset from Part: 0.55mm
- Upper Vertical Separation Layers: 1
- Lower Vertical Separation Layers: 1

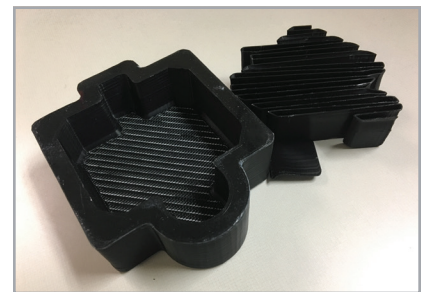


Zoomed View

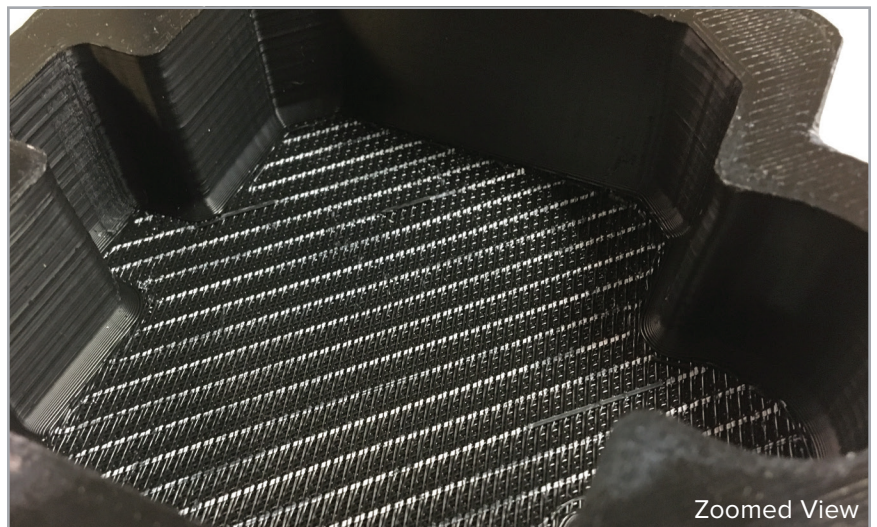
Failed Print: After - The support material is extremely difficult to break away. The print will be scarred badly when all of the support material is removed. (20 mins. of attempting removal with screwdriver and pliers)



Failed Print



Fixed Print

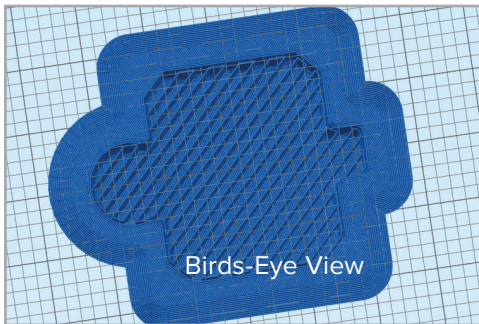


Zoomed View

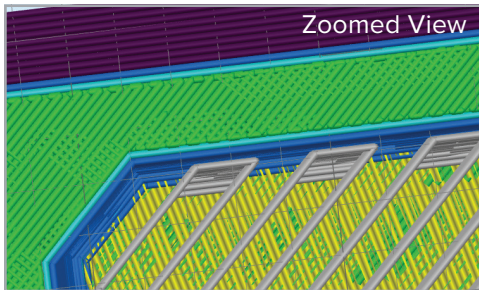
Fixed Print: There is minimal scarring, the surface is acceptably smooth, and the support removal process was greatly simplified. (5 minutes vs. original 20 minutes using the same tools.)



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



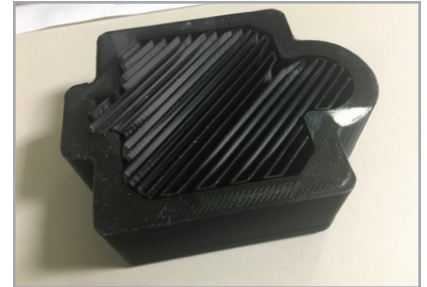
Model, Sliced:



Model, Sliced: The horizontal separation offsets the support material from the solid part by the set mm distance. (.55mm)

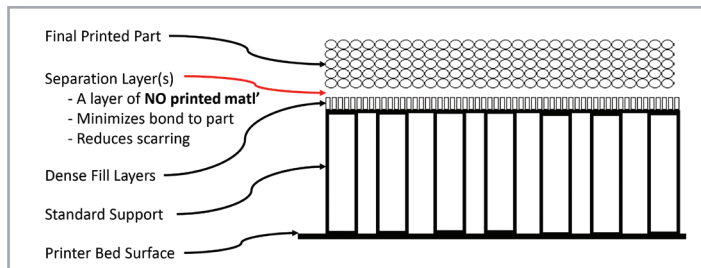


Failed Print: Before - The support material is extremely close to the outer walls of the part (horizontal offset).



Fixed Print: After - The space between the support material and outer wall allows for a tool to be inserted to aid in support removal (horizontal offset).

PRINT PROCESS SETTINGS	BEFORE FIX BAD PRINT	AFTER FIX GOOD PRINT
Material Type	PLA	PLA
Bed Temperature	80C	80C
Nozzle Size	.6mm	.6mm
Nozzle Temperature	200C	200C
Flow Rate (Extrusion Multiplier)	1.00	1.00
Extrusion Width	.72mm	.72mm
Print Speed	100mm/s	100mm/s
Layer Height	.2mm	.2mm
Number of Perimeters	2	2
Top Layers	5	5
Bottom Layers	5	5
Infill Percentage	15%	15%
Support Structures	17%	17%
Horizontal Offset	0.00mm	0.55mm
Upper Separation	0	1
Lower Separation	0	1



KEY POINTS

The print process training on the 3DP website allows for print users to determine the support cube print settings before attempting larger prints.