Slicing Settings Explained

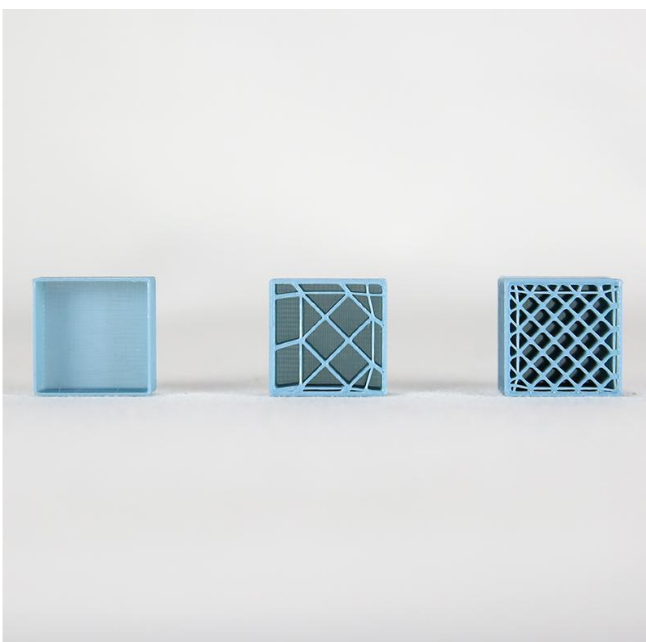
Created by Jane Leung & Lindsay Santos-Cox

Resolution

Change resolution settings. Choose from 300, 200, 100, and 50 Micron. Default set to Medium - 200 (0.2mm per layer). Printing at a lower resolution (300 micron) will allow for a faster print, while printing at higher resolutions (50 micron) will take the longest, but allow for the best finish. For more information on printing resolution.

Infill

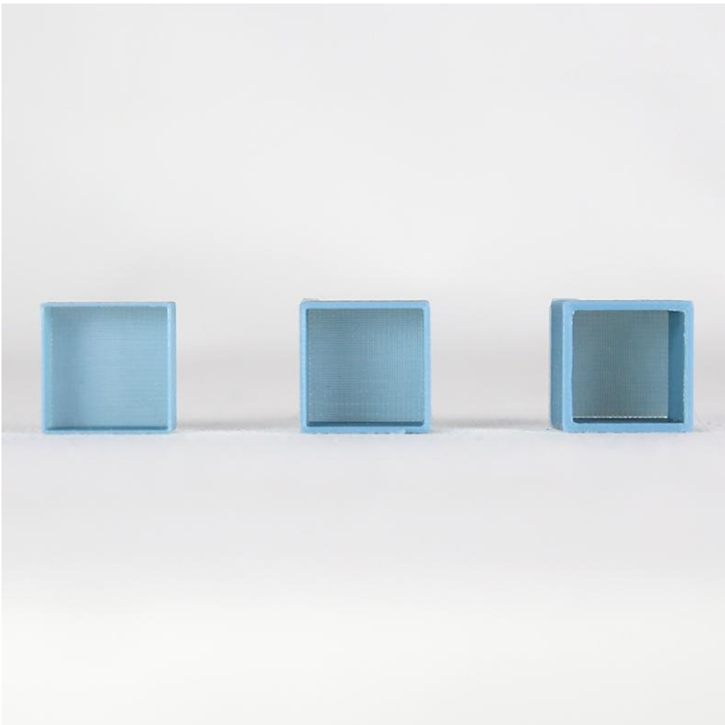
The infill setting adjusts the model's density, default is set to 10%.



From left to right: 0% Infill, 10% Infill, 20% Infill

Wall

Changes the thickness of the wall that make up the outer surface of the model. Default is set at 1.



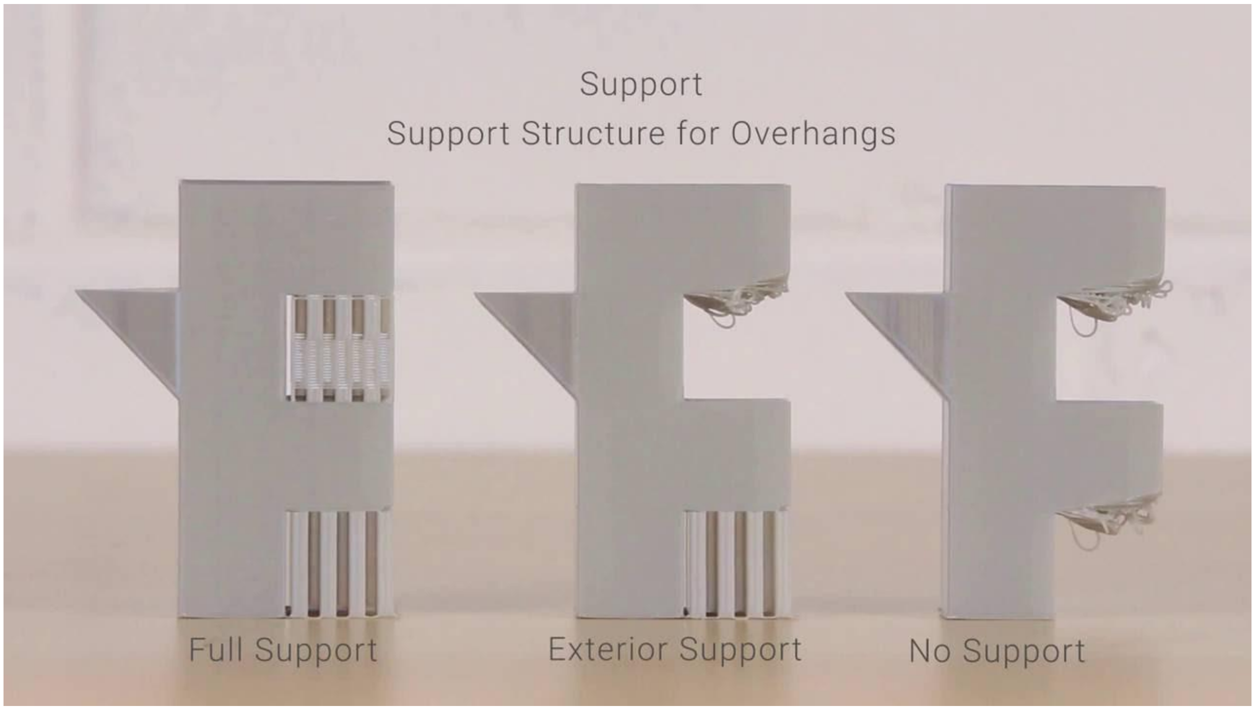
From left to right: 1 Wall, 2 Wall, 3 Wall.

Support

Activate support when you have steep angles or overhanging structures on your model; there are two modes of support:

Ext Mode (Builds support on the exterior of the object): This mode is useful when you don't want support structures in every nook and cranny of your model, and only on the exterior where it's easier to remove.

All Mode (Builds support everywhere it needs): This mode of support will print structures everywhere it is needed, which is useful when you have several overhanging points at different areas of the print.


Support
Support Structure for Overhangs
Full Support
Exterior Support
No Support