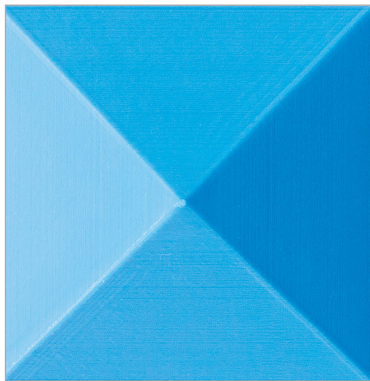


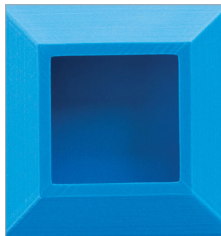
SHELLS AND INFILL

Print settings can dramatically change the strength, appearance, print time, and other properties of your printed parts.

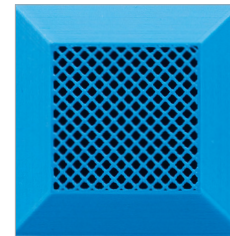
Shells are the perimeter on each layer; they make up the walls of your part. Infill is the internal structure of your part. You can set the infill of your part to be anywhere from 0% (hollow) to 100% (solid). Increasing the infill and number of shells will make your parts stronger, but will increase print time and filament use.



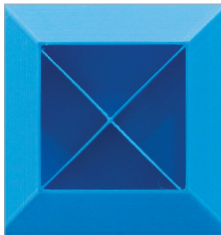
PRINTED PYRAMID
10% Infill / 02 Shells
without supports



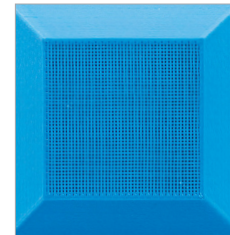
PRINTED PYRAMID
0% Infill
08 Shells



PRINTED PYRAMID
25% Infill
02 Shells



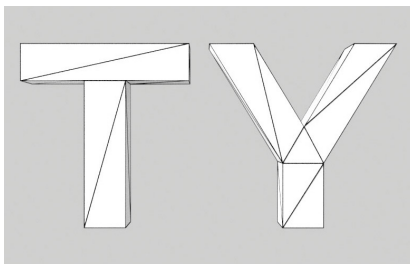
PRINTED PYRAMID
02% Infill
02 Shells



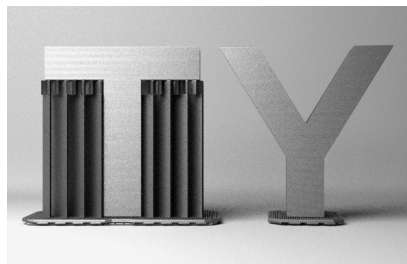
PRINTED PYRAMID
50% Infill
02 Shells

SUPPORTS AND RAFTS

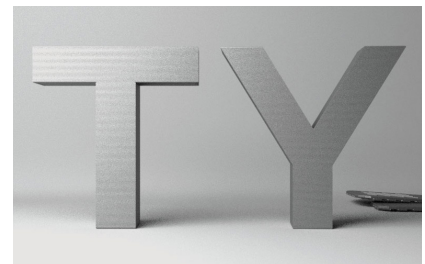
Supports are printed scaffolding for overhangs. If your model has overhangs greater than 68 degrees (measured from the vertical axis) then you will need to print with supports. A raft helps the part adhere to the build plate by laying down an even, flat foundation to print on.



3D Model: The T model has overhangs greater than 68 degrees and needs support material. The Y model does not need support material.



Supports: After printing, the T will need support material removed. Both printed with rafts.



Final Print: Final parts after removing supports and rafts.