

## INTRODUCTION

We offer a lot parameters that you can alter to suit the needs of your model. If you are new to 3D printing, we offer tips (in red) on our automatic quoting page. If you want to learn more about 3D printing and make more educated decisions on the 3D printing process, then this document is what you need.

## INFILL PERCENTAGE

As you may already be aware, most FDM objects are not printed as solids. Printing a solid object would use a large amount of filament and take a very long time to print. On the other hand, if you printed an object hollow with just thin outer walls it would print fast and use very little filament. A hollow print would not have much strength either.

Infill percentage is the amount of filament printed inside the object, between inner and outer walls and this determines the strength, weight and the amount of time to create the print.

In general, the strength of an FDM object is directly tied to the infill percentage used during printing. For example, a part utilizing 50% infill is approximately 25% stronger than a part that utilizes 25% infill.

However, the amount of strength gained by increasing infill percentage does not increase linearly. For example, increasing infill percentage from 50% to 75% will only result in an additional strength increase of 10%.

For a typical print that does not require a large amount of strength 12 to 15% is a good starting point.

## INFILL PATTERN

Infill patterns can also affect the model's strength. The weight may or may not be affected. We currently offer a rectilinear pattern and a honeycomb pattern. The rectilinear pattern lays down a layer of parallel lines that alternates direction by 90 degrees on each layer. For general use the rectilinear pattern is fine.

## WALLS

These are the exposed vertical walls of your model, typically built up vertically along the z axis. These can be specified to be either 1, 2 or 3 wall thicknesses. The thicker the wall the more strength your model has. If you will be using screws or bolts you'll want a higher number. Typically you will want 1 or 2 wall thicknesses.

## BOTTOM LAYERS

This is the part of your model that contacts the build platform and the bottom layer of any model part above a supported area. If you are printing a model just to test for fitment, you can reduce the cost by selecting zero (0) bottom layers, leaving the infill visible.

## TOP LAYERS

This is the part of your model that faces upwards and is exposed when looking downward at your model. If you are printing a model just to test for fitment, you can reduce the cost by selecting zero (0) top layers, leaving the infill visible.