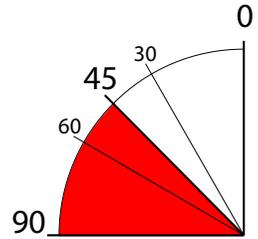


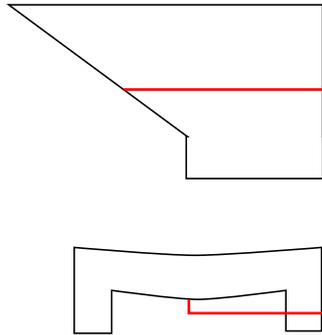
SECTION 6 : FDM PRINTING BASICS

SECTION 6.1 : OVERHANGS AND SUPPORTS

What is an overhang?



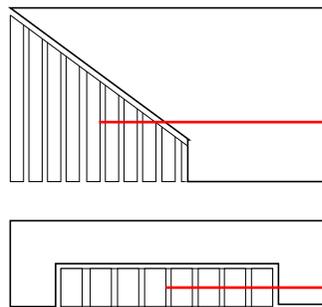
Support material is required once an overhang exceeds 45 degrees (1) or where layers are printing without the support from previous layers (2)



OVERHANGS

- An overhang occurs when part of a model's geometry on a newly printed layer is no longer fully supported by the layer below. This can occur when a model has protrusions or if angled walls are present. Some overhangs can be easily printed by 3D Printers, however if an overhang exceeds 45 degrees, support material will be required to successfully print the part.

What is support material?

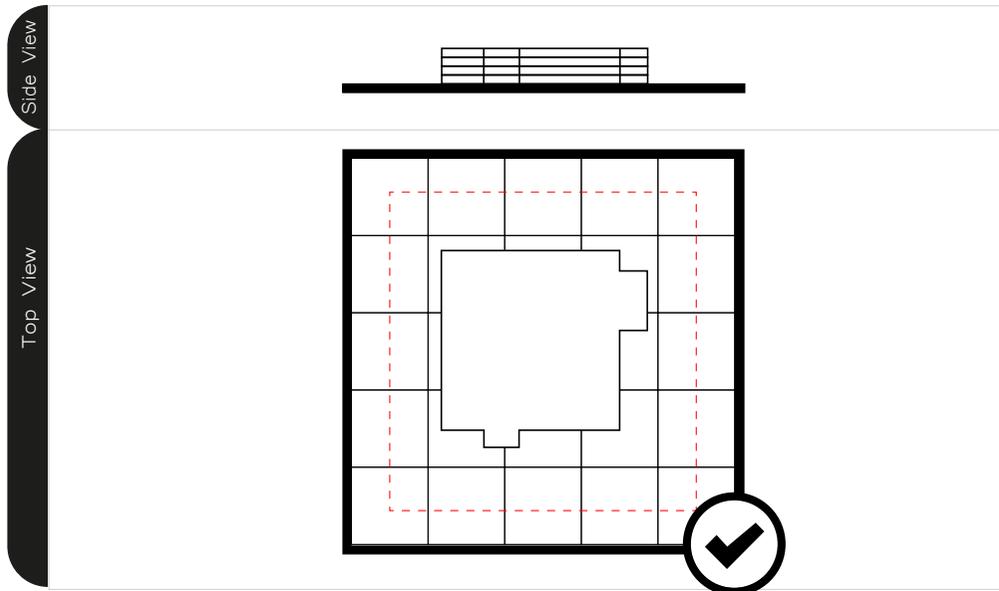


SUPPORT MATERIAL

- Support material is required for overhangs – i.e. Starting from the layer at which the print will not be building upon previous layers. The machine places extra material underneath these layers to support it during the printing process. This excess material is usually easy to remove, and does not affect your model structurally or visually once removed. If you have an overhang of 45 degree or more, you will need to place support material

Please be aware, if your model contains small / delicate sections it becomes difficult to remove FDM support material without damaging the model in the process.

SECTION 6.2 : BUILD PLATE ADHESION AND WARPING



BUILDPLATE ADHESION

Build plate adhesion refers to the raft and first few layers of your model “sticking to the build plate” when you start a print. Poor buildplate adhesion often occurs when there is only a small surface area of your model in contact with the build plate.

WARPING

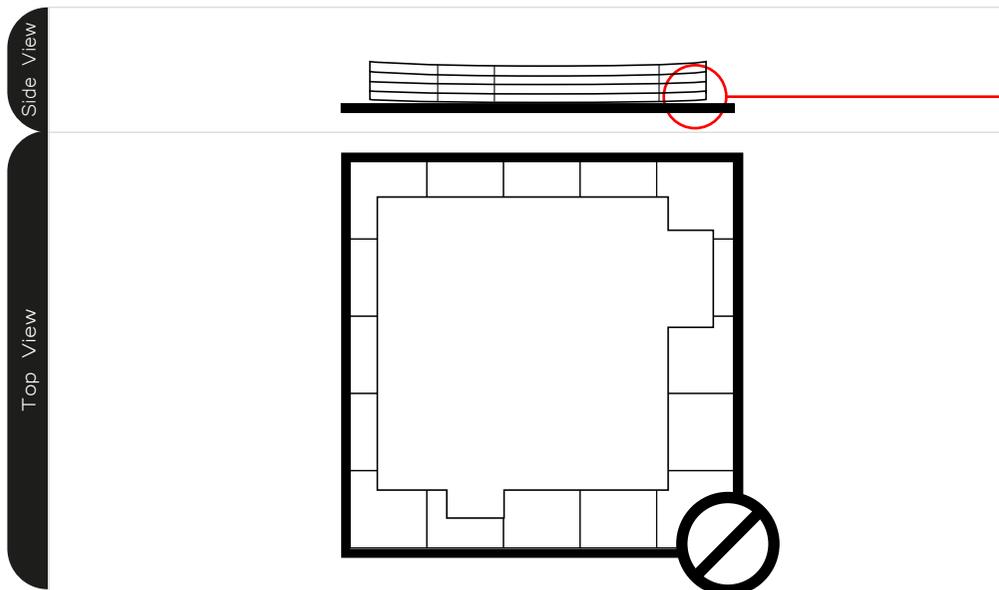
Warping occurs when a 3D model cools too quickly and detaches from the heated build plate. It is caused by the uneven contraction or shrinking of part of model. In the case of a bad print, the corners of your raft / model may start to lift up.

To improve buildplate adhesion

- Keep your model to 80% of the build plate area
- Always place model centrally
- Ensure you have a raft or brim enabled in your slicing software.
- Ensure you have oriented your model in a good orientation.
- Often the ‘heaviest’ part of your model should be in contact with the build platform.

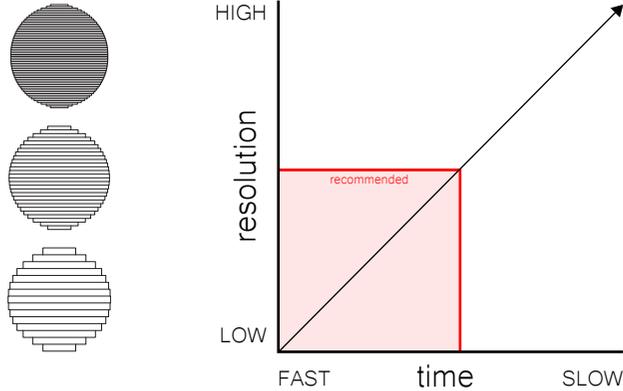
If warping continues, you may increase your infill. This will give your print a greater mass resulting in a model that will cool more gradually.

If you have tried all the options above but your print is still not adhering correctly, ask a D-Fab Staff member for some assistance.



SECTION 6.3 : RESOLUTION, LAYER HEIGHT AND INFILL

What is an overhang?

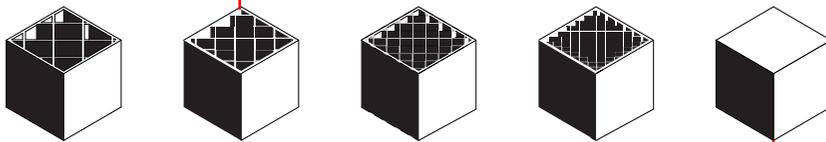


LAYER HEIGHT

Layer height refers to the thickness of each layer of a 3D print. The thinner the layer height, the higher the resolution of the print.

It is important to note low layer height (higher resolution) prints take significantly longer to print. High resolution prints should only occur when printing final models. All sketch and prototyping models should be completed at a low or medium resolution.

Infill Density



INFILL

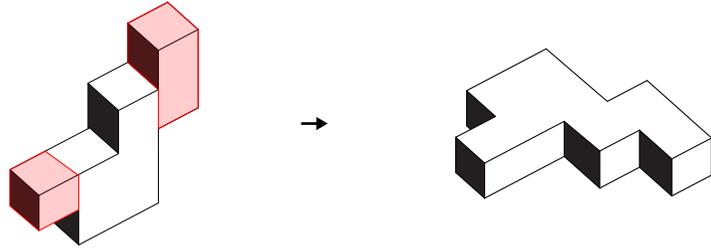
Infill is the amount of material used to fill your 3D print. The density of infill can be varied depending on how strong your print needs to be.

■ Most models will only need 20% infill or less.

Carefully consider whether you need to have a density greater than 20% as this will increase the amount of material used, as well as the printing time with no added benefit.

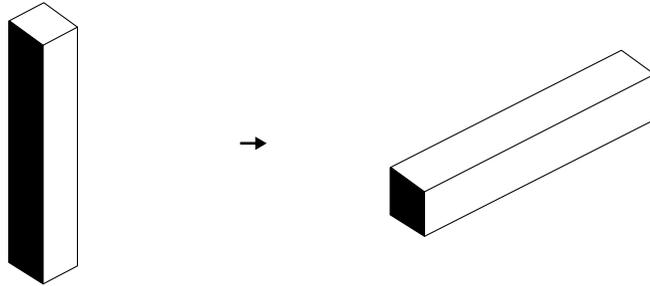
■ NEVER USE SOLID INFILL

SECTION 6.4 : ORIENTATION



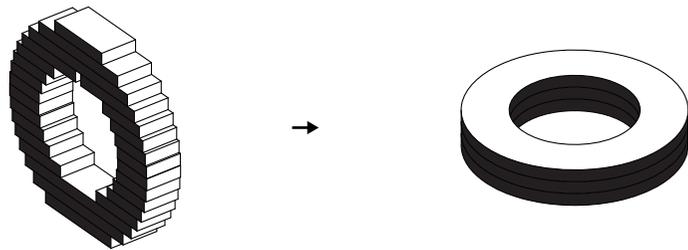
Speed:

If you are wanting a fast print turn around, it is best to orient you model in a way that it will require little to no support material. This can be done by limiting overhangs, printing on a flat base and on occasion, splitting models.



Strength:

Due to the layered nature of 3D printing, parts will always be more likely to spilt or fracture in the z-direction. To strengthen parts, models should be placed with more surface are covered by the x and y axis.



Surface Finish & Details:

As a model is printed, layer lines become visible. When printing a model, it is best to orient the print with most details in the x and y axis. This will allow for smoother curves, unaffected by the shifting layer height.

SECTION 6.5 : COMMON TERMS

.gcode:	Coding language generated in Cura containing the instructions for every movement required to complete a specific task for Ultimakers printers.
.zcode:	Coding language generated in Z-Suite containing the instructions for every movement required to complete a specific task for Zortrax printers.
Cura:	Slicing software for Ultimaker printers.
Extruder:	An extruder is a common name given to the part of a 3D printer, which contains the nozzle in control of heating and extruding (pushing out) filament.
Filament:	Filament is the general name for printing material used by FDM printers. This material is has the appearance of a wire and is normally coiled around a spool.
Heated Print Bed:	A heated surface on which a 3D model will be printed. The heating of the bed allows for greater build plate adhesion and reduces the likelihood of print warpage.
Print Bed:	A surface on which a 3D model will be printed.
Slicing:	The practice of using printer specific software to convert .stl files to either .zcode or .gcode.
Spool:	A spool is a small wheel used to store 3D printing filament. The spool will often contain information regarding what material
STL:	Stereolithography, the most common file format used for 3D printing
Z-suite:	Slicing software for Zortrax printers.