1 Basics to 3D Printing: Overview

MODULE 3

Within this module, you will be introduced to basic rules and functions commonly used while 3D Printing. This module will ensure your ability to assemble a functioning 3D print file, and geometry for use in the CAP Fabrication Lab or a personal printer, as well as handcraft to digital model conversion methods to further enhance your abilities to interface with CNC equipment. This module's objectives are to expand your workflow capabilities, introduce new methods of model making, and improve comfortability with 3D Printers.

A1- Intro to 3D Printing

With this assignment you will be introduced to terms and concepts that are used while 3D Printing. Understanding these terms will allow you to interface with 3D Printer settings, and slicing softwares later in this module. Terms such as : Watertight Geometry, STL, Infill %, Wall Count, Support Angle, and Layer Height, are all commonly used while preparing a file for printing. For this assignment please read and understand how each of these terms applies to a model you wish to 3D Print.

Watertight Geometry - This pertains to how your model is constructed, whether that be through Rhino, Revit, Fusion 360,Sketchup, etc. A watertight model must have all holes closed and edges joined. Think of it as a boat, you don't want any water to slip into your boat, therefore the boat must be well constructed and thoughtfully crafted to ensure there are no leaks.

.STL File Format - File formats are important, they're the language in which software such as slicers are able to interpret your information. In 3D Printing .STL files are the most commonly used, also known as "Standard Triangle Language" this file type has no units attached to it (inch, ft, cm .etc.) however it does contain a number associated to scale. It is important when saving a model as an STL to track which units you are using so you are able to make the appropriate conversion to millimeters.

Infill % - Infill is how 3D Printers turn soft plastic into a sturdy model. Within the slicer software a model will be hollowed out and replaced with a honeycomb like inner structure to give strength to the out walls of your model. Higher infill amounts will result in a sturdier model however at the cost of time and materials. Balancing this setting will help with time and material costs.

Wall Count - The wall count represents how many layers of plastic a 3D Printer will generate as the "shell" of your model. Going back to the boat analogy, the walls would represent the bottom of your boat, with more walls, the thicker your boat becomes. With fewer walls the boat may become weak or "squishy" as its shell is thinner. Balancing this setting will help with time and material costs.

Support Angle - Generating supports for a model is very common when preparing for 3D Printing. Printers are unable to print in mid air. Support angle is the angle in which a printer is forced to add supporting material to build onto. Think of a vase, as the vase becomes wider while building upwards, extra material at the base may be required in order to help the vase continue outwards and upwards.

Layer Height - A model is generated by stacking molten plastic layer by layer until a specified height and shape is met. Layer Height is the variable that allows us to control just how many layers are in our model, the lower the layer height, the more layers to print, the higher the fewer layers needed.