

05 Introduction to 3D Printing

Objective:

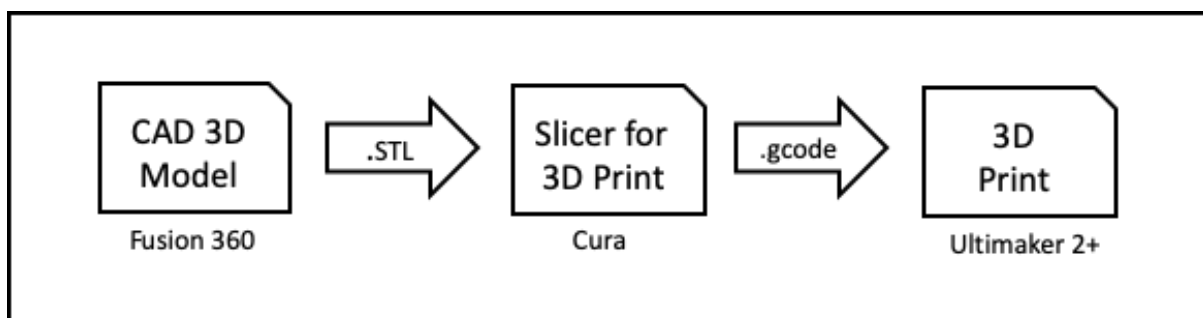
- To understand what additive manufacturing is and why it is important
- To understand the 3D printing workflow
- To learn how to create 3D models using extrusion and revolution
- To understand the parameters involved in creating 3D printed objects
- To print a small object using 3D printing

Additive Manufacturing

Earlier manufacturing processes used subtractive methods to develop models and objects. Subtractive methods involved the removal of material from a raw substrate block e.g. milling, turning or routing. There are some objects that cannot be created using subtractive methods because of holes and cavities within the object.

Additive Manufacturing uses a blank surface and adds material to form the object. This method is often called 3D Printing and can be done by a variety of machines e.g. FDM and Resin printers. In the lab the most common methods of 3D printing is using the FDM (Filament Deposition Method) which yields good models in relatively fast times.

The 3D Printing process



1. We first model the 3D object using Fusion 360.
2. Using Fusion 360, we generate the STL ("Standard [Tessellation](#) Language") file which is a representation of the 3D object.
3. We use a Slicer program (CURA), configured for the 3D printer e.g Ultimaker 2+ to load the .STL file.
The Slicer program, slices the 3D model into layers for printing. It also allows us to configure how we would like to print the object.
4. The Slicer program outputs the print job (.gcode) in a format that the 3D printer understands and prints it.

Creating 3D Models

There are 4 methods of creating 3D models:

- Extrusion of a 2D surface
- Revolution of a closed profile about the axis
- Creating a model using 3D shapes
- Free-hand sculpting the model

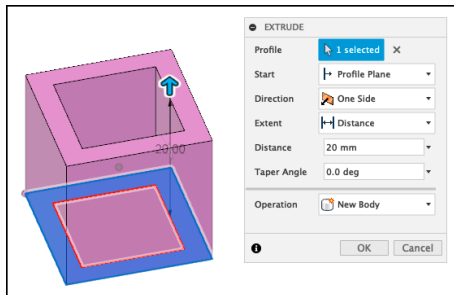
You have already seen how Extrusion works in your previous lab sessions. In this lab session, we will use Revolution to generate our 3D model. We will not cover the last 2 methods as it involves advanced 3D CAD processes.

Creating a 3D model using Extrusion

The method of creating a 3D model using extrusion is to

1. Create a 2D profile
2. Use the extrude command to extrude the profile in the 3rd axis.

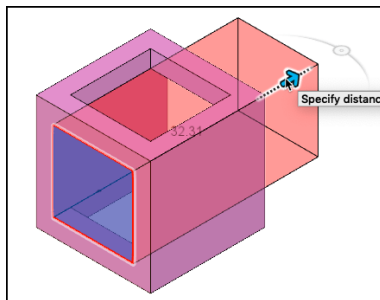
This method is useful when you can draw a 2D profile of the object.



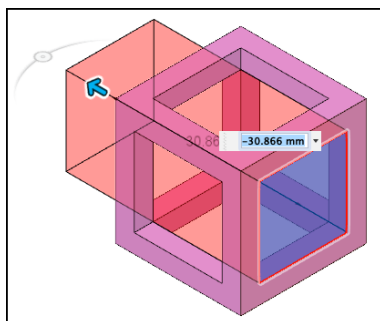
We would like to obtain the model of a hollow cube, length 20mm and with edges 3mm thick.

We start by drawing a 2D profile of a hollow square on the X-Y plane and extruding in the Z-axis by 20mm.

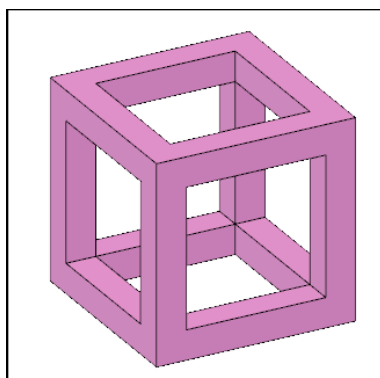
The hollow square is drawn using a rectangle and then applying the Offset command to put in the sides.



We now create a new sketch on the X-Z face of the cube. We use the Offset command to create the inner rectangle and then Extrude the inner rectangle to cut and obtain the edges.



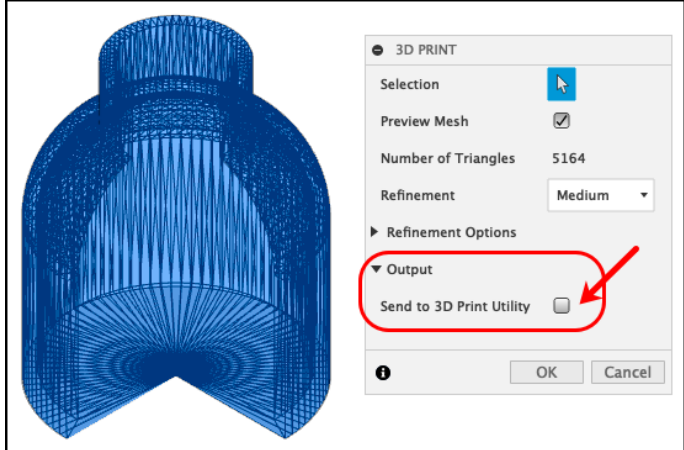
We apply the same process as above on the Y-Z face of the cube and the result is a hollow cube.



A hollow cube, ready for 3D printing!

Preparing your object for 3D printing using Cura

We will now prepare your object for 3D printing using Cura. Cura is an open source slicer, available for most OS platforms and is commonly used by manufacturers of 3D printers. You may have a 3D printer that uses a different slicer, but the process is still the same.



Select MAKE > 3D Print from the Ribbon bar.

When the 3D print dialog appears

- Click on the bottle to select it
- Enable Preview Mesh to see the tessellations created
- Select “Medium” refinement
- Uncheck the “Send to 3D Print Utility” to create the STL file
- Click OK

Fusion 360 will now ask you for the name of the STL file to be generated. Create the STL file on the desktop

Preparing for 3D print

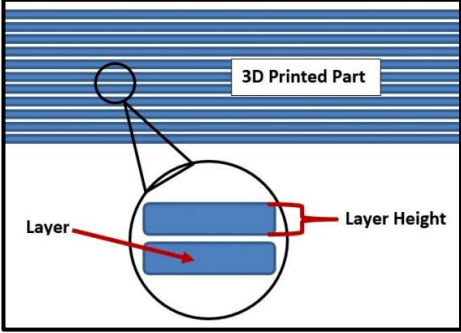
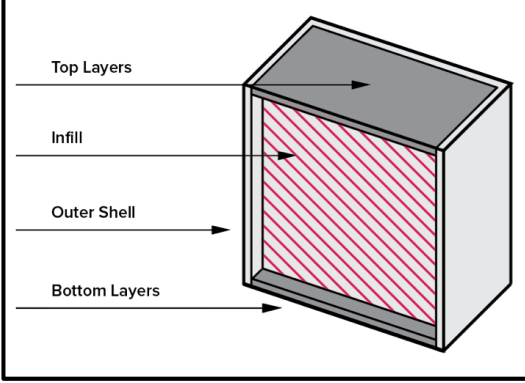
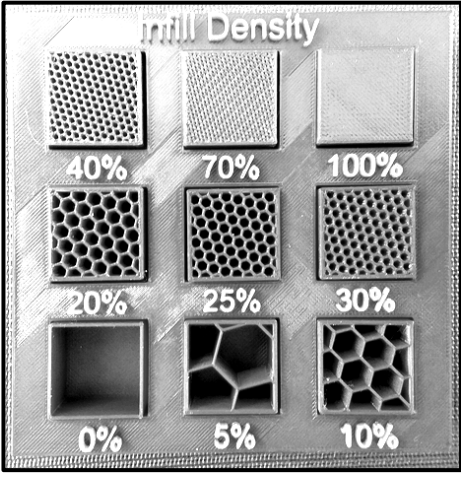
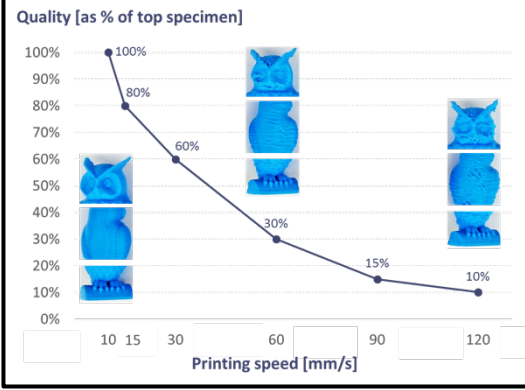
When you have obtained your .STL file, you are now ready to prepare your object for 3D printing on a 3D printer. There are many types of Slicer programs, however the function and use are almost the same. We will look at one of the most popular (and Open Source) slicers: Ultimaker Cura (<https://ultimaker.com/software/ultimaker-cura>).


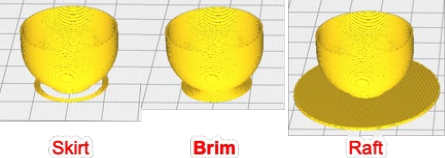
Although originally written for the Ultimaker series 3D printer, this slicer program has been adapted to suit many types of 3D printers in the market. Your instructor will assist you in downloading and setup of the correct printer driver for the 3D printer.

The workflow of this process is as follows:

1. Load the STL file
2. Orientate the object such that it best suits 3D printing
3. Set the parameters:
 - a. Layer Height (resolution)
 - b. Wall thickness (min)
 - c. Fill Density
 - d. Speed of print
 - e. Supports
 - f. Bed adhesion
4. Slice the object using the above parameters and generate the 3D printer code
5. Save the code on Removable Memory (e.g. Flash drive, SD card)
6. Use the file on the Removable Memory to print the object.

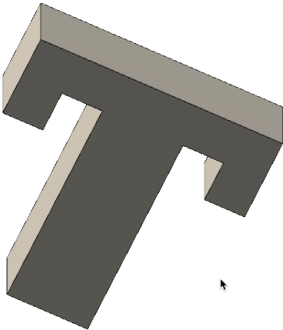
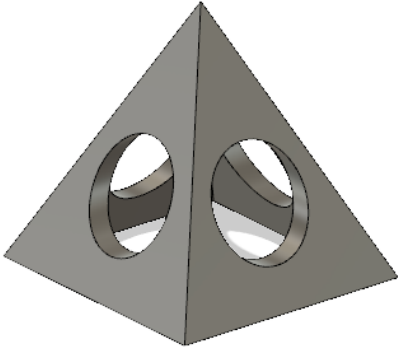
Let us look at some of these parameters and how it affects the print

 <p>The diagram shows a stack of blue horizontal layers representing a 3D printed part. A circular callout provides a magnified view of a single layer, with a red arrow indicating the thickness of the layer, labeled 'Layer Height'. The entire stack is labeled '3D Printed Part'.</p>	<p>Layer height refers to the thickness of each sliced layer. The 3D printer will control the layer height through the extrusion of the filament. Fine prints : 0.1mm Fast prints : 0.2 mm (our prototypes) Fine prints take a longer time to print.</p>														
 <p>The diagram shows a 3D printed rectangular box. Labels with arrows point to different parts: 'Top Layers' at the top surface, 'Infill' for the internal hatched structure, 'Outer Shell' for the side walls, and 'Bottom Layers' at the base.</p>	<p>Wall thickness refers to the thickness of the inner (bottom) and outer (top) walls of the object. Typically this is set as a minimum value of 1mm. Usually the middle part of the object is left hollow, hence the wall thickness determines the "hardness" of the skin of the model.</p>														
 <p>The image shows a grid of 10 square samples of 3D printed material with different infill densities. The densities are labeled as follows: 0%, 5%, 10%, 20%, 25%, 30%, 40%, 70%, and 100%. The 100% sample is a solid block, while the 0% sample is completely hollow.</p>	<p>Fill Density In order to save time and material, the solid sections of the object are printed with a infill pattern. Increasing the density increases the strength and time taken to print. Typical infills are 20%</p>														
 <p>The graph plots Quality (as a percentage of the top specimen) against Printing speed (in mm/s). The quality drops significantly as the printing speed increases. The data points are as follows:</p> <table border="1"> <thead> <tr> <th>Printing speed [mm/s]</th> <th>Quality [as % of top specimen]</th> </tr> </thead> <tbody> <tr> <td>10</td> <td>100%</td> </tr> <tr> <td>15</td> <td>80%</td> </tr> <tr> <td>30</td> <td>60%</td> </tr> <tr> <td>60</td> <td>30%</td> </tr> <tr> <td>90</td> <td>15%</td> </tr> <tr> <td>120</td> <td>10%</td> </tr> </tbody> </table>	Printing speed [mm/s]	Quality [as % of top specimen]	10	100%	15	80%	30	60%	60	30%	90	15%	120	10%	<p>Speed of print The speed of print determines how fine and accurate your 3D object will look. Higher speeds results in rough work, while lower speeds result in fine work but take much longer to print. Typical print speed: 50mm/s Image source: http://my.3Dmatter.com <u>What is the influence of infill %, layer height and infill pattern on my 3D prints?</u></p>
Printing speed [mm/s]	Quality [as % of top specimen]														
10	100%														
15	80%														
30	60%														
60	30%														
90	15%														
120	10%														

	<p>Supports A good 3D printer can usually handle an overhang angle of 50 deg before it fails. Hence, in order to have good overhanging prints, we need supports. Supports can be easily removed after printing.</p>
 <p style="text-align: center;"> Skirt Brim Raft </p>	<p>Bed Adhesion The first couple of layers in a 3D print usually determines whether the print is going to be successful. To start the print, the filament is laid on the printer bed and the layers added on. Depending on the type of print and amount of bed adhesion required, there are 3 options:</p> <ol style="list-style-type: none"> a. Skirt – no adhesion to object, a boundary is printed around the object. b. Brim – a number of lines are laid next to the object to secure the object to the printer bed. c. Raft – a layer of one or two lines of filament are laid on the bed first, the object is then printed onto the bed. This provides of excellent adhesion.

Worked Examples

Examine each of the objects below and suggest how the object should be placed for optimal 3D printing. All objects will be printed using Fast/Normal settings.

	<p>Example 1:</p> 	<p>Example 2:</p> 
Mode:	Fast/Normal	Fast/Normal
Orientation:		
Layer Height:		
Wall thickness:		
Fill Density:		
Printing speed:	50 mm/s	50 mm/s
Supports (Yes/No)		
Bed adhesion:		
Orientation		

Task

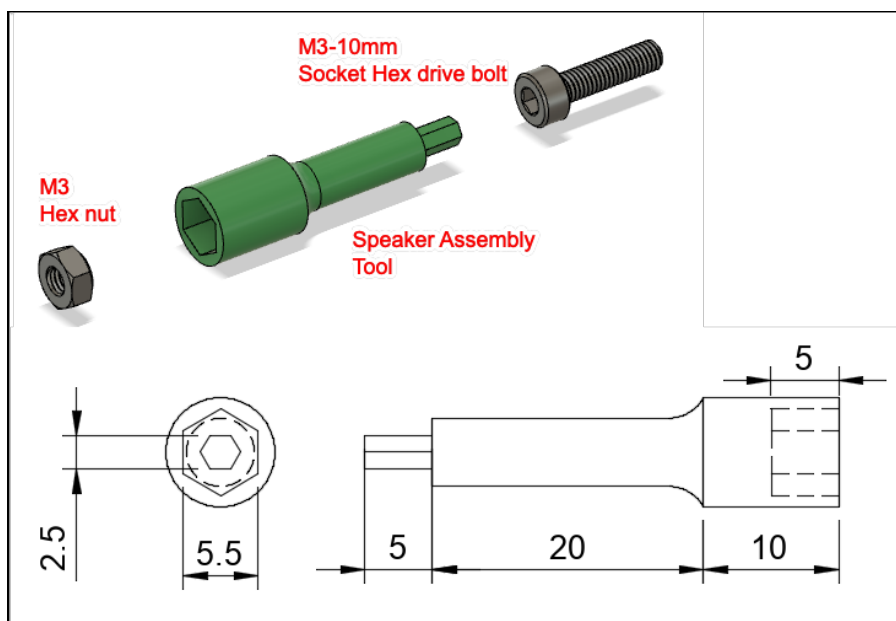
You will design and 3D print a tool for fixing your Speaker Box.

The Speaker has a large magnet at the back of the cone which is used to help move the cone producing air movements (or sound). However, when you try to attach screws and nuts to the speaker, the metal pieces are easily attracted to the magnet. We will make a tool that will allow us to hold the nut in place so that we can attach the speaker to the front panel. As the tool is 3D printed using PLA (a form of plastic), it will not be attracted by the magnet.

The hex (6-sided) nuts that we use are known as standard M3 nuts. Use the internet to find the dimensions of the nut so that we can design our tool. The tool must be able to hold the nut securely.

On the other hand, the bolts that we use to secure the speaker are known as M3-10mm Socket Hex drive bolts. The head of the bolt is round, however the drive is a hex-shaped drive. The hex shaped drive has the dimensions of 2.5mm across the flat sides. The bolt itself has a diameter of 3mm and a length of 10mm

Design a combination tool that can perform both tasks and 3D print it for your project. The diagram below will help in the design dimensions.



3D print your tool and test it with the hex nuts and bolts provided. Does it fit?

Adjust your sizes with the “**tolerance**” provided by your lecturer and reprint the object. You should have a working tool now.