Assignment-1

1 a.) Trade-off between file size and geometric accuracy while export STL file:

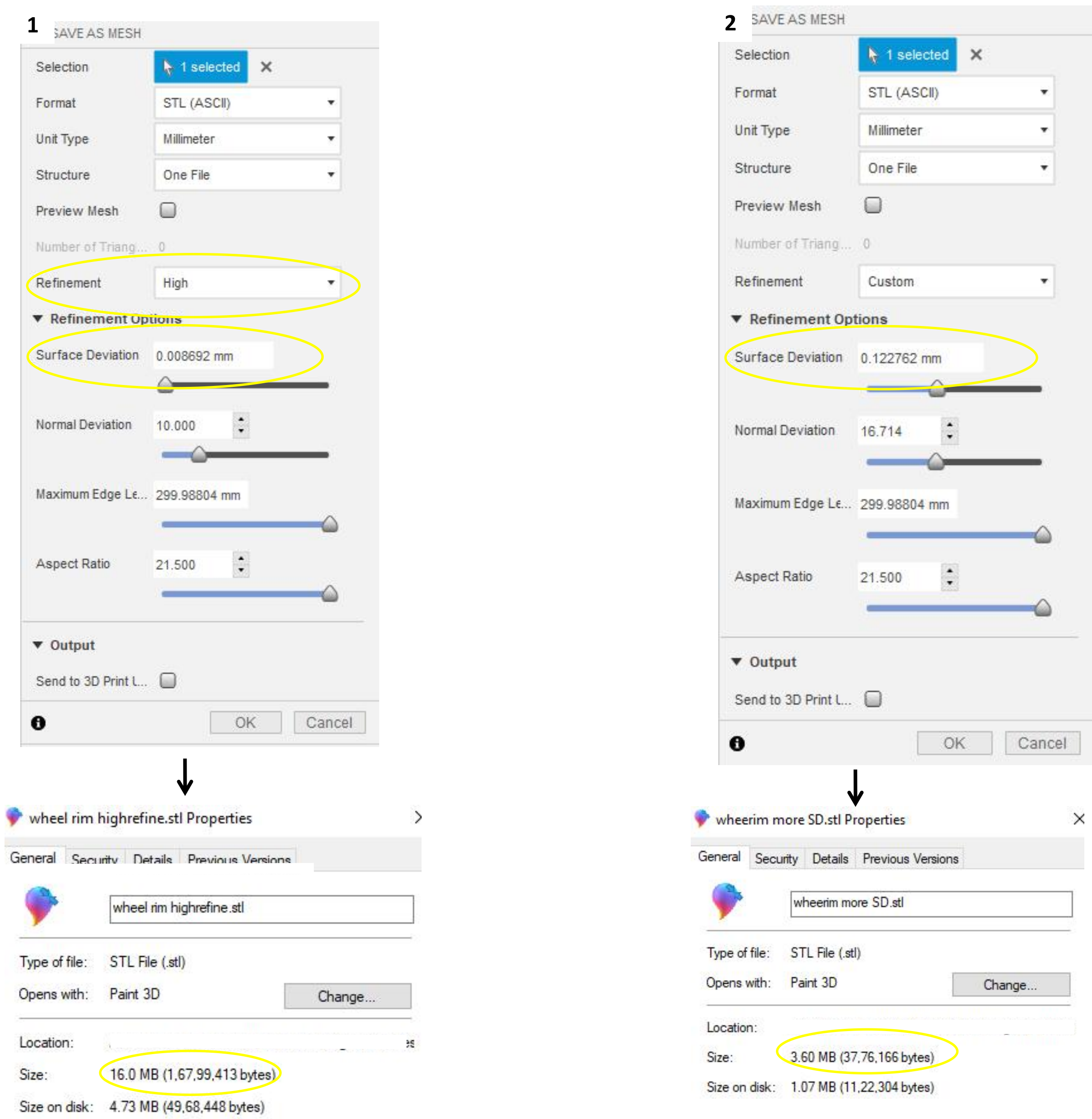
We might be tempted to simply crank up the resolution settings in our CAD programme to the highest resolution and call it a day because a higher resolution STL mesh generates a smoother, more accurate model. However, **raising the STL export resolution also increases the size of the STL file**, which typically results in longer software processing times for both creating the STL file and submitting it to Cura to slicing the STL and getting it ready for 3D printing.

Beyond a certain point, the resolution of the STL file may be significantly higher than the machine precision of 3D printer, resulting in a time cost for the resolution of the STL file that may not be reflected in the produced parts. See the below how refinement options will effect the file size.

- **How refinement options effects the STL file size**

a. If we increase this Surface deviation/chordal deviation values to smaller value it resulting in the big file size.
b. If we use ASCII STL format than Binary format STL will result in increasing file size
c. Increasing in Normal deviation/ Angular deviation value will result in smaller STL file size.

Figures: STL export settings in Fusion 360

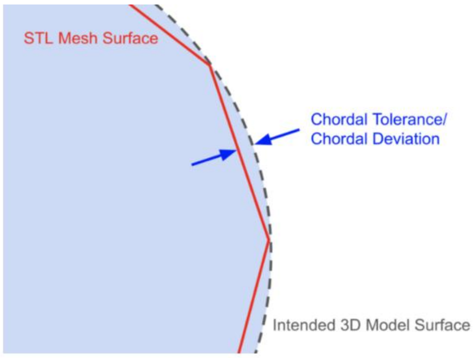


In the above figure1 refinement kept as high,and surface deviation is 0.008mm resulted in **16 MB** file size, where in figure 2refinement kept as custom,and surface deviation is 0.122mm resulted in **3.6 MB** file size. Surface/chordal deviation,Normal/ angular deviation explained below for clear understanding.

- **Chordal Tolerance/Chordal Deviation**

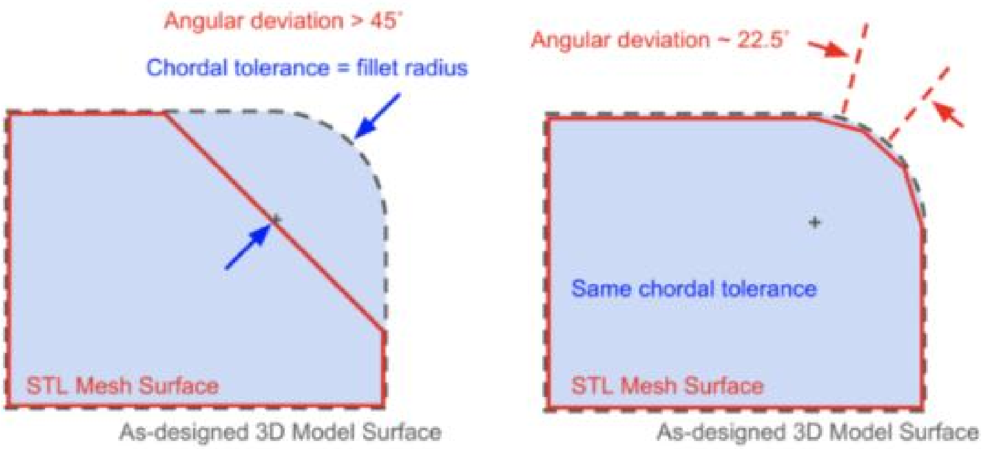
The chordal tolerance (or chordal deviation) is a setting that controls the global dimensional accuracy of the STL when compared with the as-designed 3D model. Chordal tolerance is usually specified as the maximum normal (perpendicular) linear deviation allowed from the surface of the as-designed 3D model and the nearest triangular face of the resulting STL, as seen in the following image.

Assuming that the chordal tolerance is the limiting factor in STL resolution, a **smaller chordal tolerance** value will result in a higher resolution STL, with more triangles and a **larger file size**.



- **Angular Tolerance/Angular Deviation/Normal Deviation**

The angular tolerance setting (sometimes referred to as angular deviation or normal deviation) controls the maximum angle allowed between the normal vectors of any two neighboring triangles in the mesh, and one can think of it as a parameter that 'refines' the mesh with higher resolution beyond what the chordal tolerance would otherwise allow. A good example of when the angular tolerance comes into play is typically with small curved surfaces, like fillets whose radii are similar in size to the chordal tolerance. Without the angular tolerance setting, these small fillets might have very visible flat spots, or be turned into a chamfer in the extreme case where the fillet radius is equal to the chordal tolerance as shown in the following diagram.



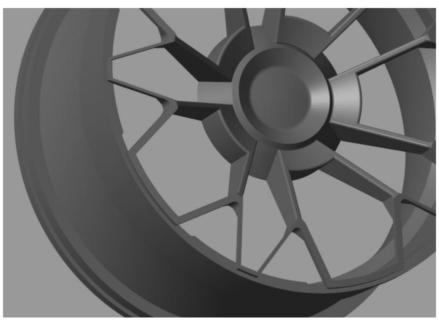
Note that while angular deviation is usually measured in degrees (with a lower value resulting in a higher resolution model), some CAD software specifies the angular deviation as a **dimensionless 'angle control'** parameter that varies in value from 0 to 1, with larger values specifying a higher STL resolution around curved surfaces

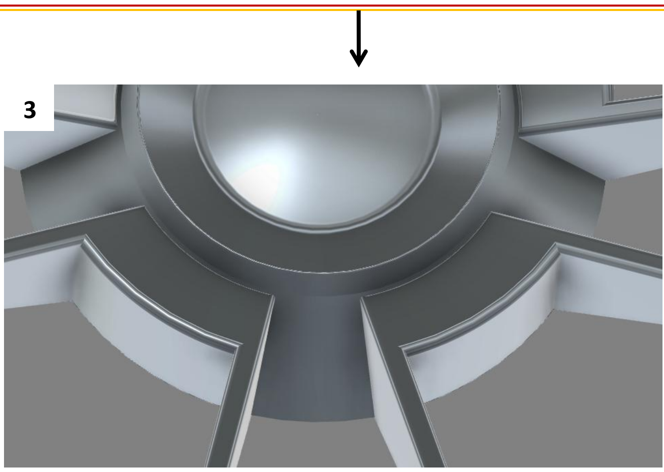
Exporting CAD geometry with the right STL resolution will result in 3D printed parts with the highest dimensional accuracy and surface finish, without slowing down the slicing process.

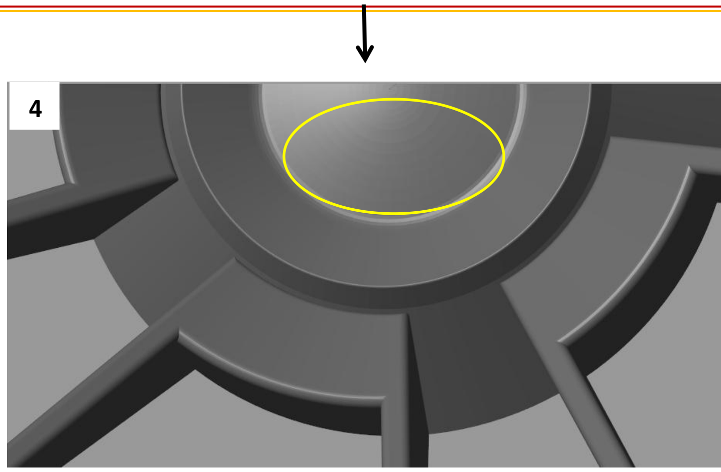
- **Recommended settings in Fusion 360 as a starting point:**

I. STL Format: Binary
II. Surface Deviation: 0.1 mm [0.004 in] (Note: this is the chordal tolerance)
III. Normal Deviation: 1 degree (Note: this is the angular deviation)
IV. Leave 'Maximum Edge Length' and 'Aspect Ratio' at their default values
V. [Optional] Minimum triangle side length set to 0.1 mm [0.004 in]

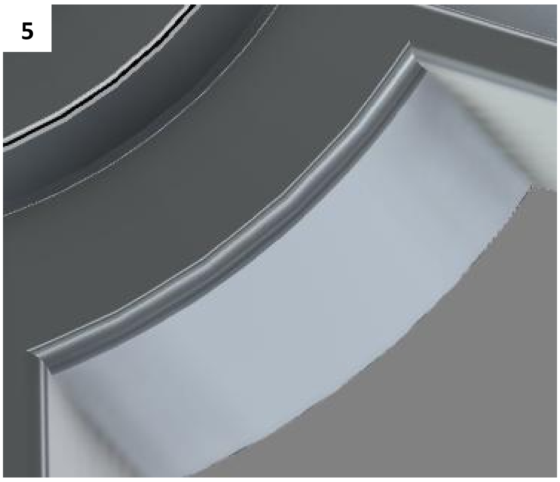
CAD geometry for the wheel rim (left) showing a smooth surfaces and the resulting exported part (right) STL file generated from the CAD. Note the flat, segmented **facets** on the wheel rim surface that are artifacts of the **low STL resolution**.see the zoom view in the figure3,figure4

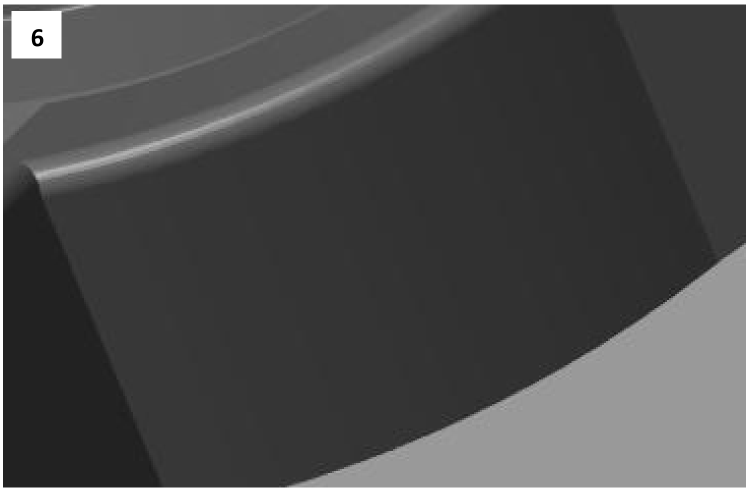






In this below figures also we can observe the facets on the surface of the STL file .figure 5 is CAD generated one, figure 6 is STL .





1 b.)Effect of layer thickness, raster parameters on build time.

Two most important parameters would be layer height and line width.

- **Layer Height:**

The layer height determines how much detail print will have in the z direction as printer prints it essentially works the same as a 2d printer only continues to move up in the z direction slowly building plastic on top of itself. If print with 0.2 millimeter layer heights will print twice as fast as one with 0.1 millimeter layer heights but we will get twice the detail in the z direction with the 0.1 millimeter layer height print the amount of material used should remain unaffected.

Quality		
Layer Height	0.2	mm
Initial Layer Height	0.2	mm
Line Width	0.4	mm
Wall Line Width	0.4	mm
Outer Wall Line Width	0.4	mm
Inner Wall(s) Line Width	0.4	mm
Top/Bottom Line Width	0.4	mm
Infill Line Width	0.4	mm

Figure: Raster parameters settings in Ultimaker Cura

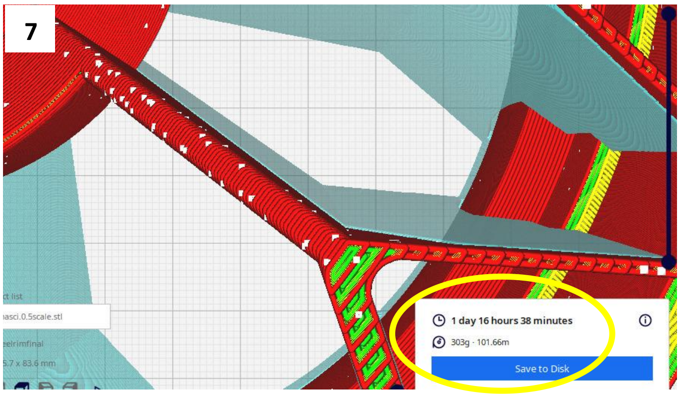
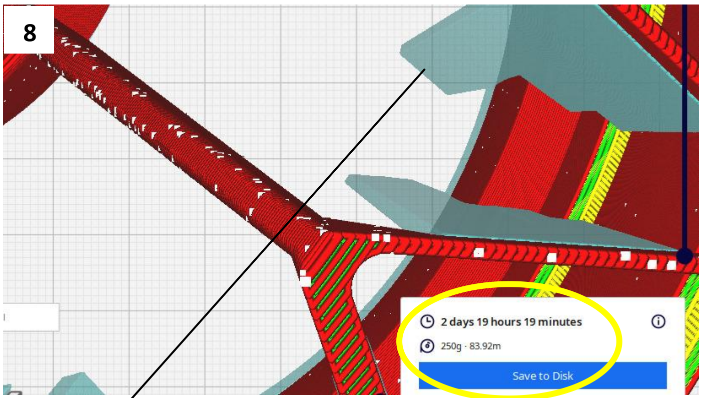


Figure 7: Layer height 0.2mm



Improper support stature created

Figure 8: Layer Height 0.1mm

Generally the layer height stay within roughly 25 to 75 of the nozzle diameter, most of the cases prints to being used 0.2 millimeter layer heights. When using a point 0.4 millimeter nozzle since it prints fairly fast and has enough quality. If layer height drop it down to 0.1 millimeter for higher quality prints and up it to 0.3 millimeters for faster prints that don't require as much detail go for the **initial layer height set to 0.3 millimeters** this will actually for **increased bed adhesion**.

In 3d printing is that having first layer lay down properly is key for the rest of print to finish successfully, so as long as the very **first layer of print doesn't** need to be an amazing quality it's smart to essentially maximum out the layer height based on nozzle diameter.

- **Line Width:**

The line width essentially the line width is dependent on nozzle diameter.The width of the line is exactly what it sounds like how wide each line of print will be. This will affect how high quality print will be in the x and y direction. If we just want to be safe we can keep line width the exact same as nozzle diameter.

The nozzle diameter is the main determining factor that said some people like to increase this to around 110 of the nozzle diameter which means line width is to be 0.44 millimeters resulting some **gaps in thin walls** this can often be changed via other methods but if model wall is let's say 1.3 millimeters thick then having line width be 0.4 millimeters can cause there be a gap in that wall since three shell walls will actually create a wall that is only 1.2 millimeters thick. If change the line width to be 0.433 millimeters then the gap shouldn't really exist.

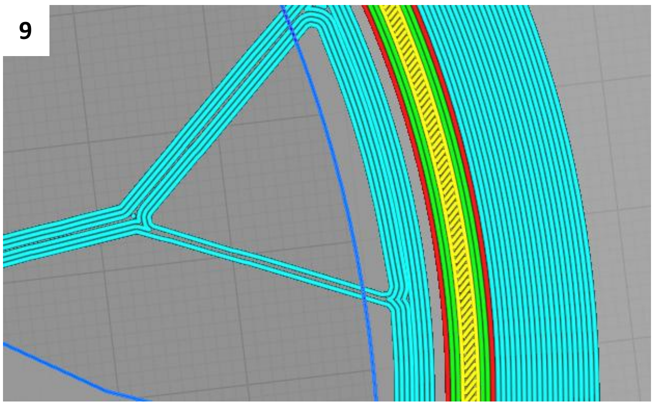


Figure 9: Line width (= Nozzle diameter)0.4mm

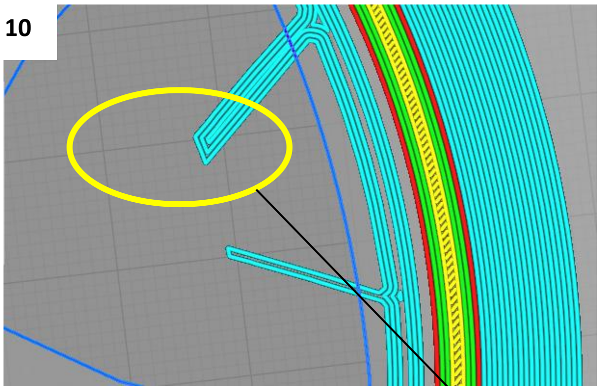
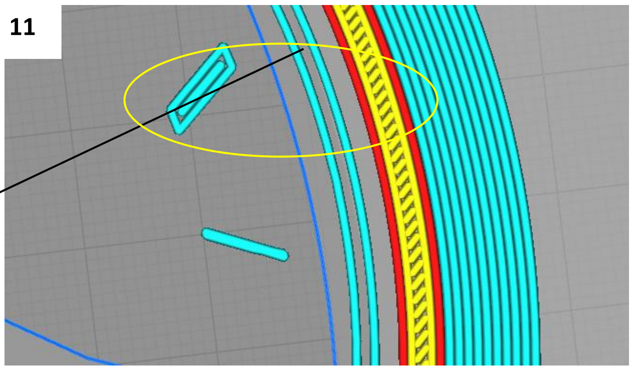


Figure 10: Line width is 0.44mm

Loss of support structures



Gaps in support structures

Figure: Line width is 0.88mm

Because we now create a shell wall just about exactly 1.3 millimeters thick but normally line width can just keep same as nozzle diameter this means that the nozzle diameter will determine the quality of print in the x and y direction as well as how long print will take to cover the same surface area.

Example if have text on print and it's really not legible it likely means we want to print with a smaller diameter nozzle the same is true with any print that has walls thinner than0.4 nozzle diameter.Another example like knife, which have sharp tip ,by using a 0.4 millimeter nozzle and the 0.4 millimeter line width is thicker than the **thinnest tip** this means we need a smaller nozzle diameter and line width for these(want higher quality in the x ,y and z direction) will need to reduce nozzle diameter and layer heights but reducing both of these will cause print to exponentially take longer since now the surface area takes longer to complete.

This problem is solved by having **line width different for outer walls** than inner walls which means in that above example we can have outer wall be 0.4 millimeters thick and then inner walls be 0.45 millimeters which would then add up to a perfect 1.3 millimeters .

Figure: Line width settings in Ultimaker Cura

- ## Additional Information

3d print these are the settings of the walls that hold in part infill, both the wall thickness and the wall line count as options only one of these is needed. Since they refer to the same thing if line width is 0.4 millimeters and have three shell walls then the shell wall thickness will be 1.2 millimeters line width is not a simple number, the amount of shell walls will depend on a couple of things like how thick is line and **how strong do we need part** to be **in that direction.** For example using a normal 0.4 millimeter nozzle with 0.4 millimeter line widths for the vast majority of **non-strong parts**.

If won't ever require more than two shell walls or 0.8 millimeter wall thickness for **non-mechanical part.** The fact is that depend on wall thickness, we can decide how strong is our part to be when being bent in that direction. For example hooks goes under tensile load(pulled apart)that means the stress is being put on these shell walls and not really the infill.

Name: K Sowjanya
Roll No: MM22M023

END