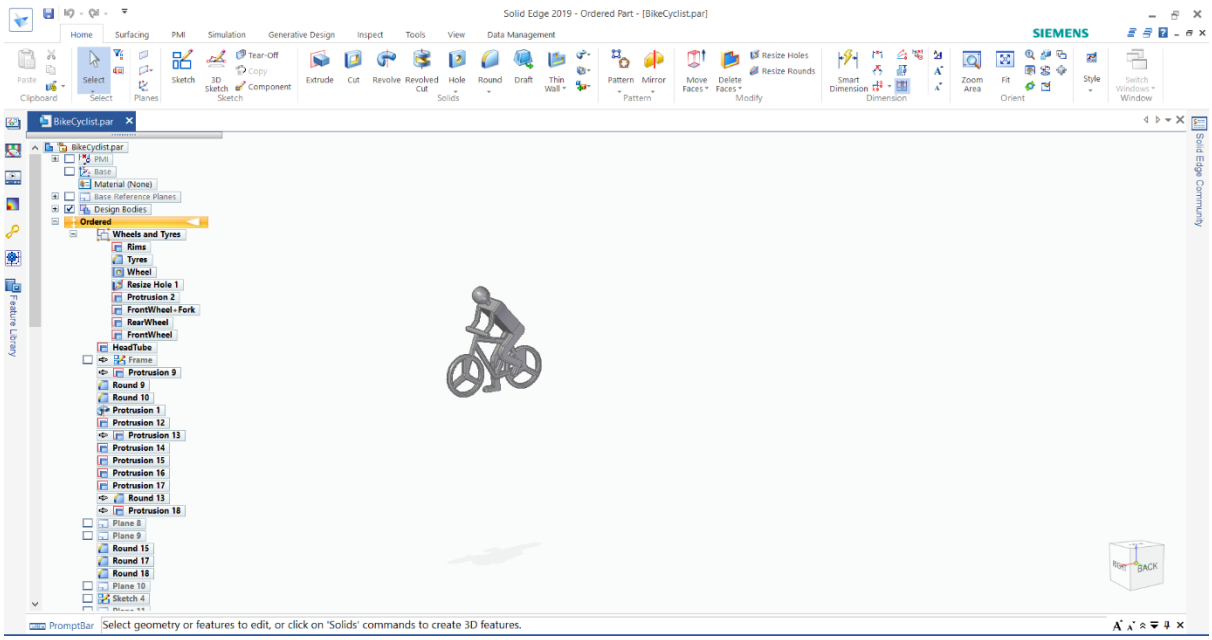
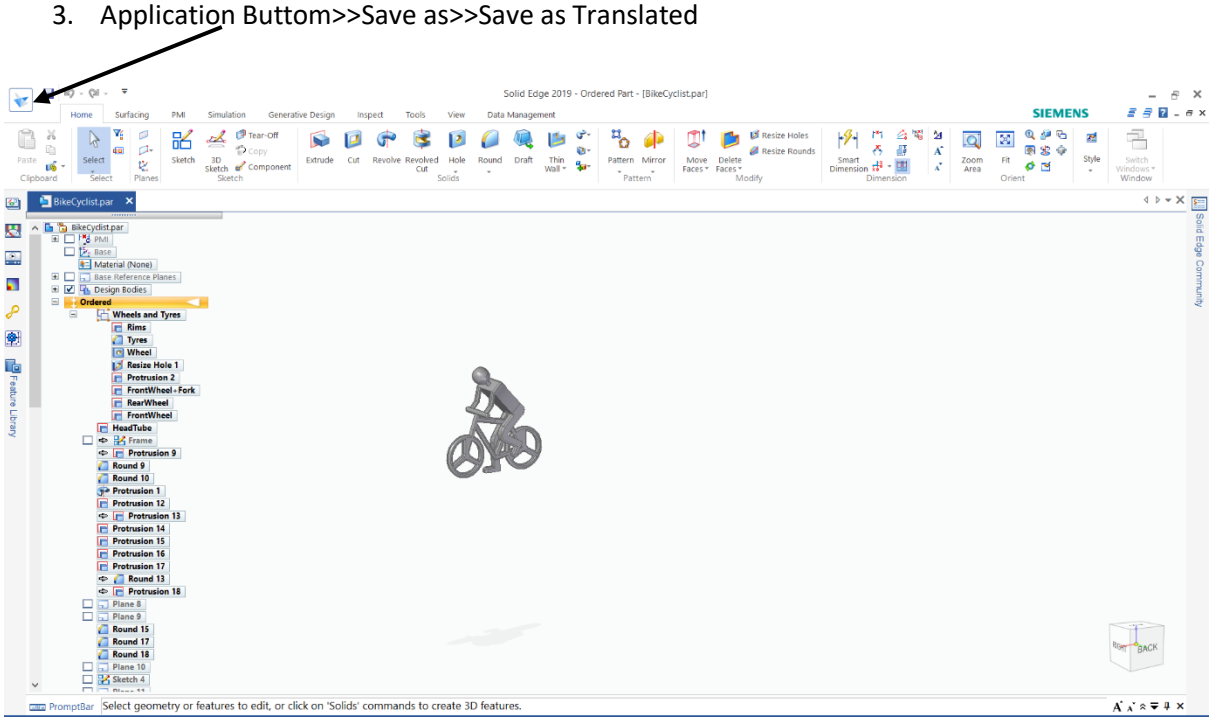
Exporting .stl from SolidEdge

1. Open SolidEdge
2. Load your file either .par or .asm



Make sure that the zero position and coordinate system directions match the tunnel coordinate system and zero location.

3. Application Buttom>>Save as>>Save as Translated



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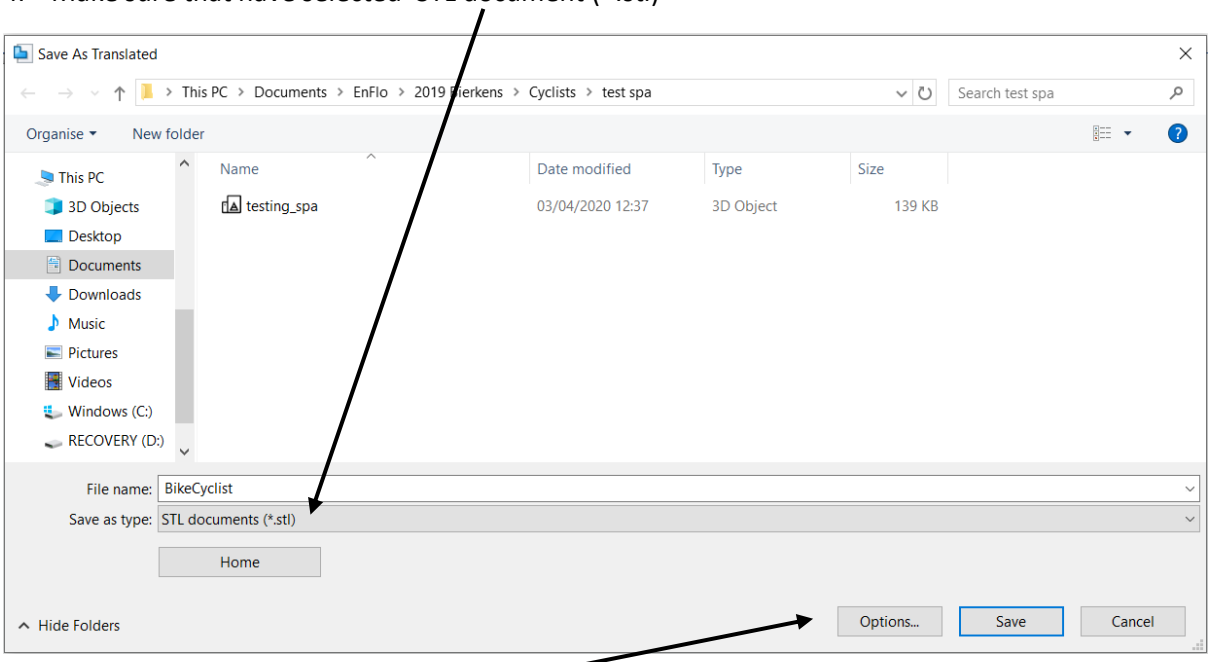
4. Make sure that have selected ‘STL document (*.stl)’



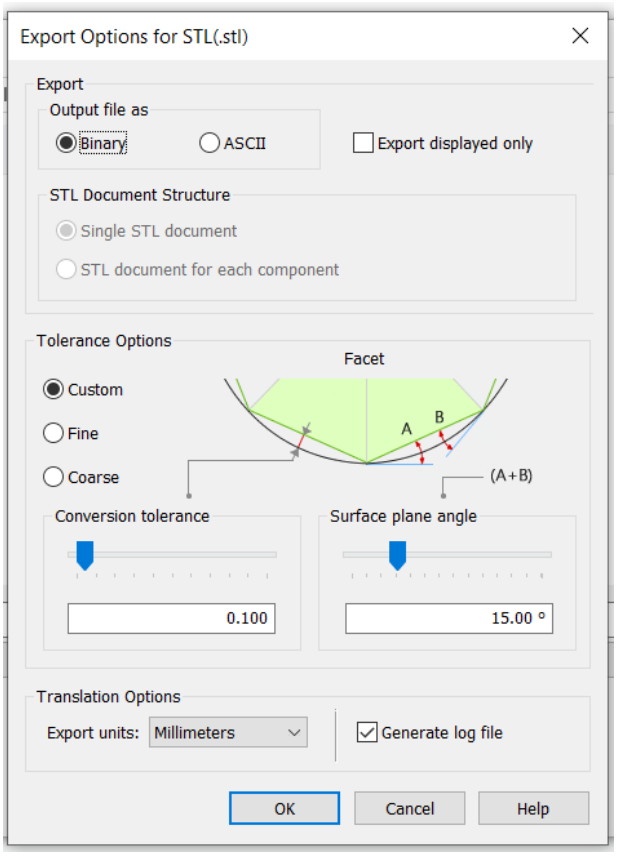
5. Click ‘ Options… ‘ to edit the export settings.

6. The following settings should ALWAYS be selected

   a. Binary
   b. Single STL document

7. Now the tolerance options can be adjusted to suit your needs. Whether you want the exported file to look exactly like the SolidEdge is upto you.

   a. In general increasing the tolerance reduces file size and reducing tolerance decreases file size
   b. Increasing surface plane angle reduces file size but makes your exported file look different, e.g will make rounded objects look more angular. Decreasing surface plane angle will make the file size larger but it will look more closely to the original file.



8. After you chosen your settings, click ‘ok’ and save in your desired location.

9. Here are some examples of .stl files exported with various tolerances and surface angles. If you have a 3D viewer on your computer you may be able to have a look at the appearance of the .stl file.

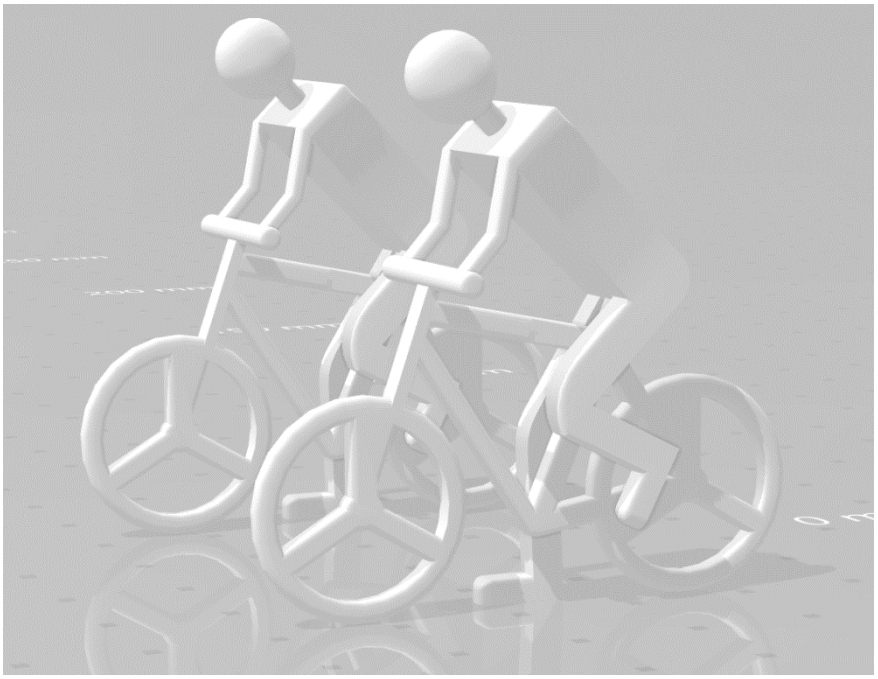


Figure 1 Tolerance = 1, Surface Plane Angle = 15

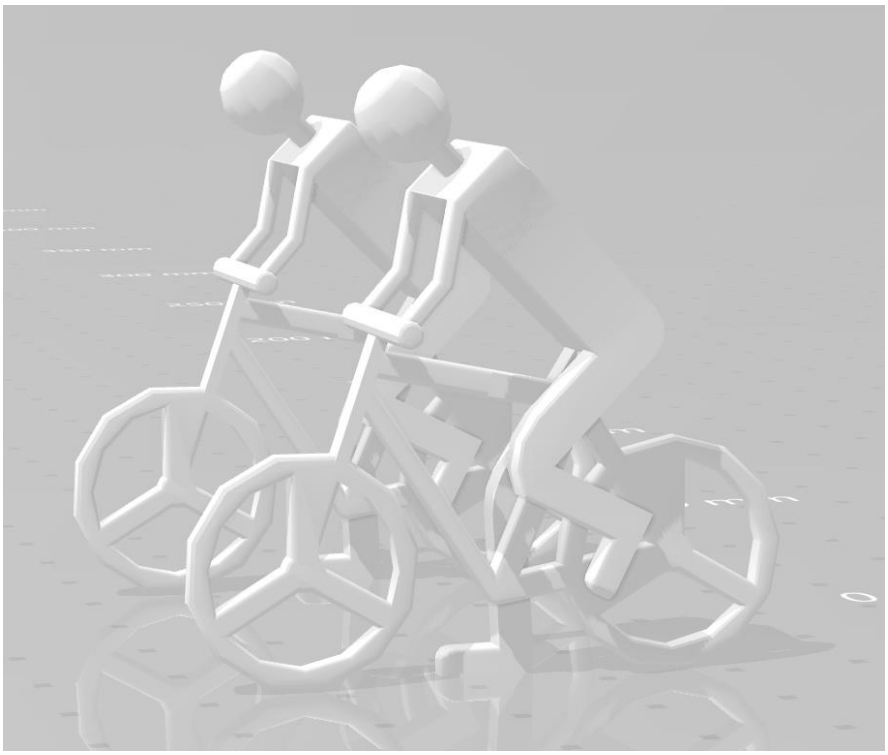


Figure 2 Tolerance = 1, Surface Plane Angle = 30

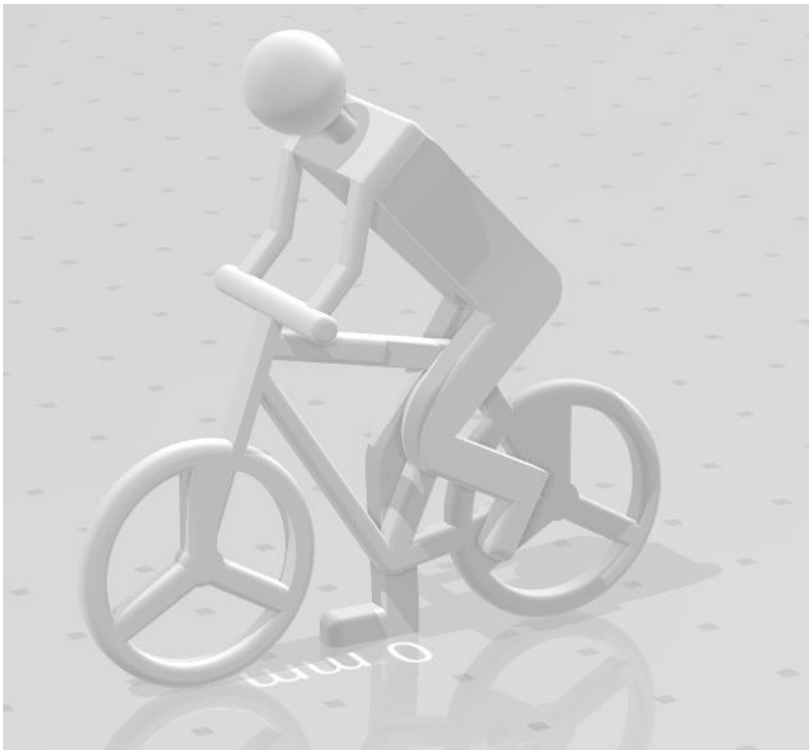


Figure 3 Tolerance = 0.001, Surface Plane Angle = 30