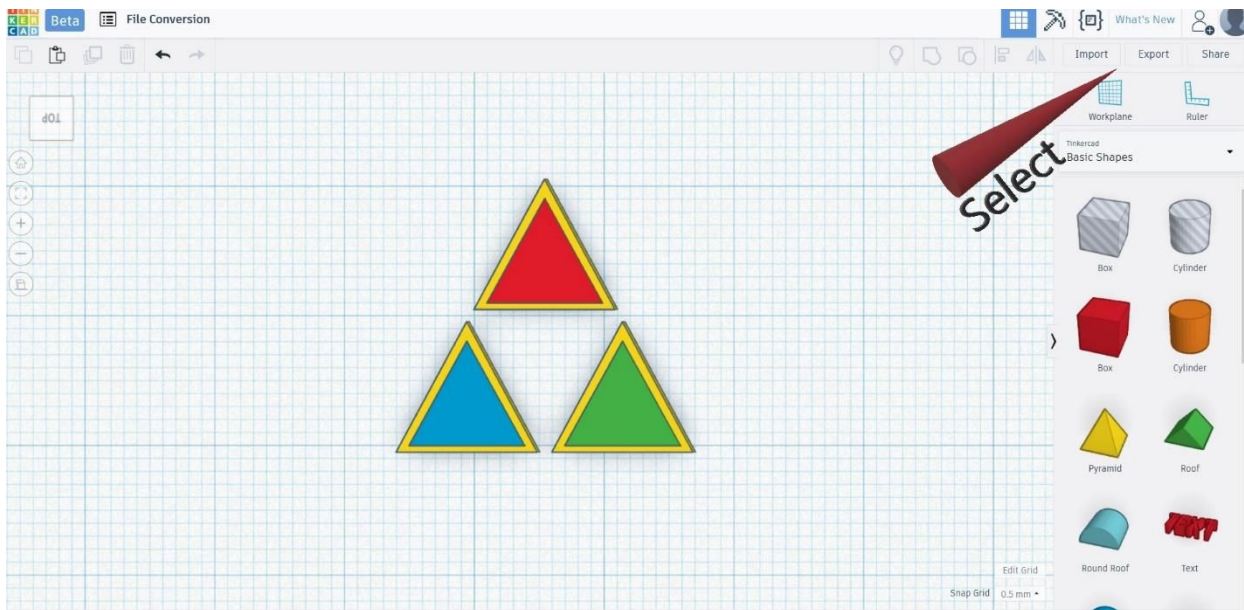# How to export for 3D printing from different software: Tinkercad, Onshape, Inventor, and Fusion360

Tinkercad

1) Complete model design in available work space
2) Select Export in the Top Right-hand corner

3) Choose Inclusion for Export
   a. Everything
   b. Selected Shapes (Must select prior to clicking export)
4) Choose File Type(s) for Download
   a. .STL – polygon mesh* (Standard Triangle Language)
   b. .OBJ – polygon mesh* with surface details (color, texture, material, etc…)
   c. .SVG – Scalable Vector Graphics (Laser Cutting/Engraving)
5) Clicking on the type will open a download menu, Select the "Save File" option
6) Click "Ok" to download your model for 3D printing

*Polygon Mesh: a collection of vertices, edges, and faces that define the shape of an object in 3D model

Onshape

1) Complete model design in available work space
2) Locate the Parts Manager in the Lower Left Corner
3) Select parts ready for file conversion. (Select multiple by holding CTRL + Left Click)
4) Right click on selected part(s) and choose Export
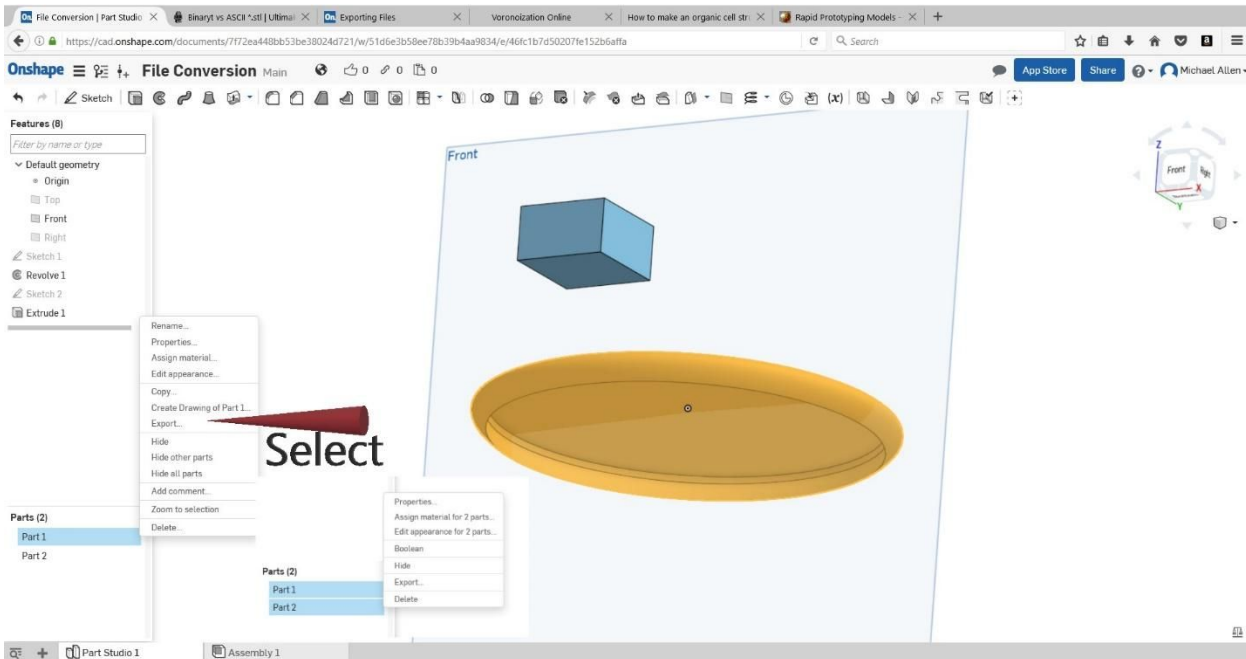
5) Choose applicable settings:
   a. File Name: Your_File_Name
   b. Format: STL
   c. STL Format: Text (ASCII*) or Binary** (Both are usable; Binary reduces file size)
   d. Units: (Determined by measurement system your design utilized)
   e. Resolution: (Determines surface area and angular difference of polygons: Coarse to Smooth Texture; higher resolutions produce more accurate curved surfaces for a better looking model)
   f. Options: Download

   (Separate files are available for individual parts, creates .ZIP with multiple .STLs)

6) Click "Ok" after you have the desired settings
7) Select the "Save File" option and click "Ok" to download your model for 3D printing

*ASCII STL: File type that stores information about the surface covering polygons

**Binary STL: File type that stores information about the surface covering polygons, condenses polygon coordinates to maintain a smaller file size

Inventor (2015 Professional)

1) Finalize model and save .ipt file to desired location
2) Click on the I in the top left corner
3) Hover over Print, Select "Send to 3D Print Service"
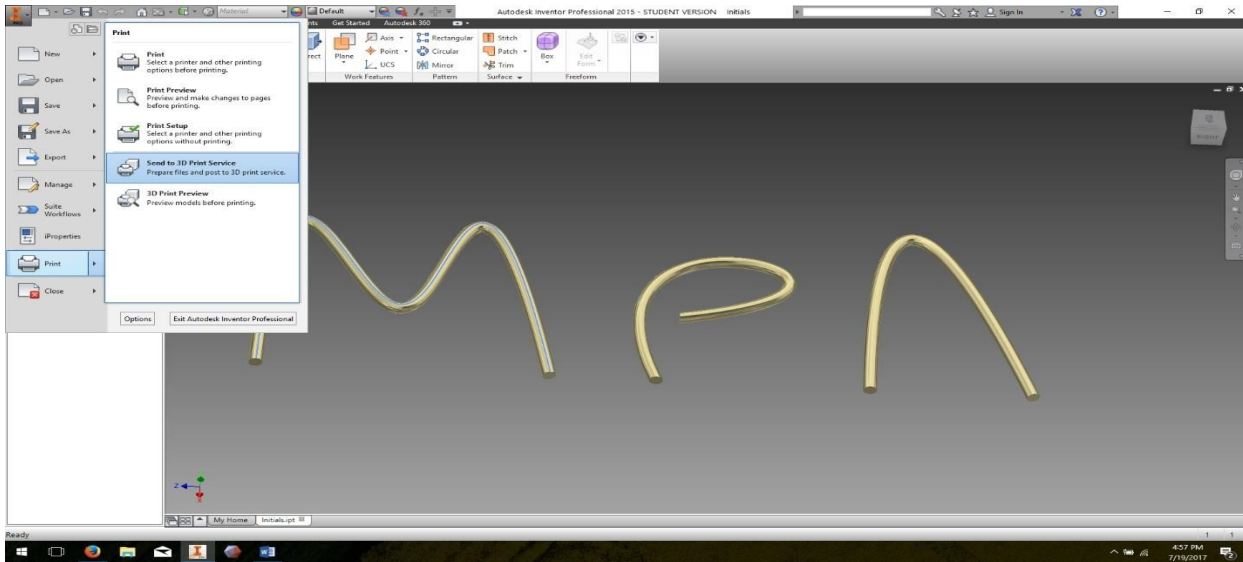
4) Change object Scaling if desired
5) Advanced Options:
   a. File Type: ASCII/Binary (Similar Usability; Binary reduces file export size)
   b. Units for exported file and Resolution settings
   c. Resolution: increasing or decreasing surface area and angular difference of polygons for final model surface (higher resolutions produce more accurate curved surfaces for a better looking model)
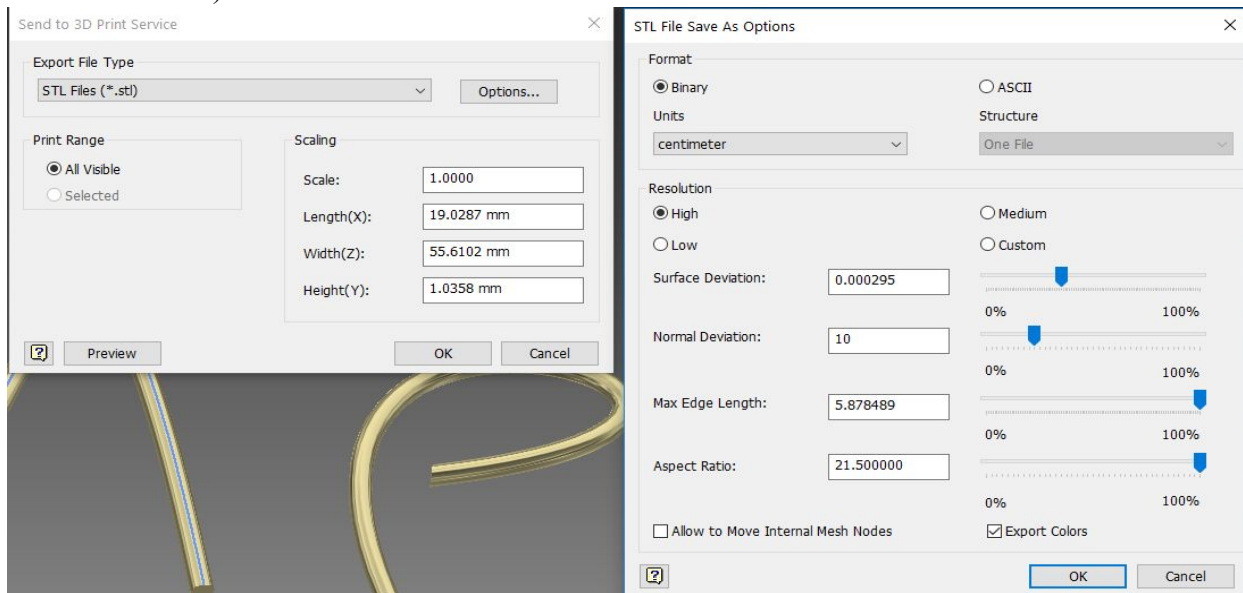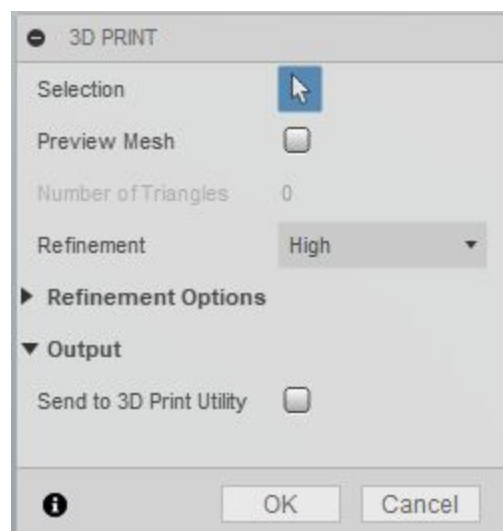
Fusion360

1) Complete model iteration and save file
2) Locate "Make" in the tool selection ribbon, select "3D Print"
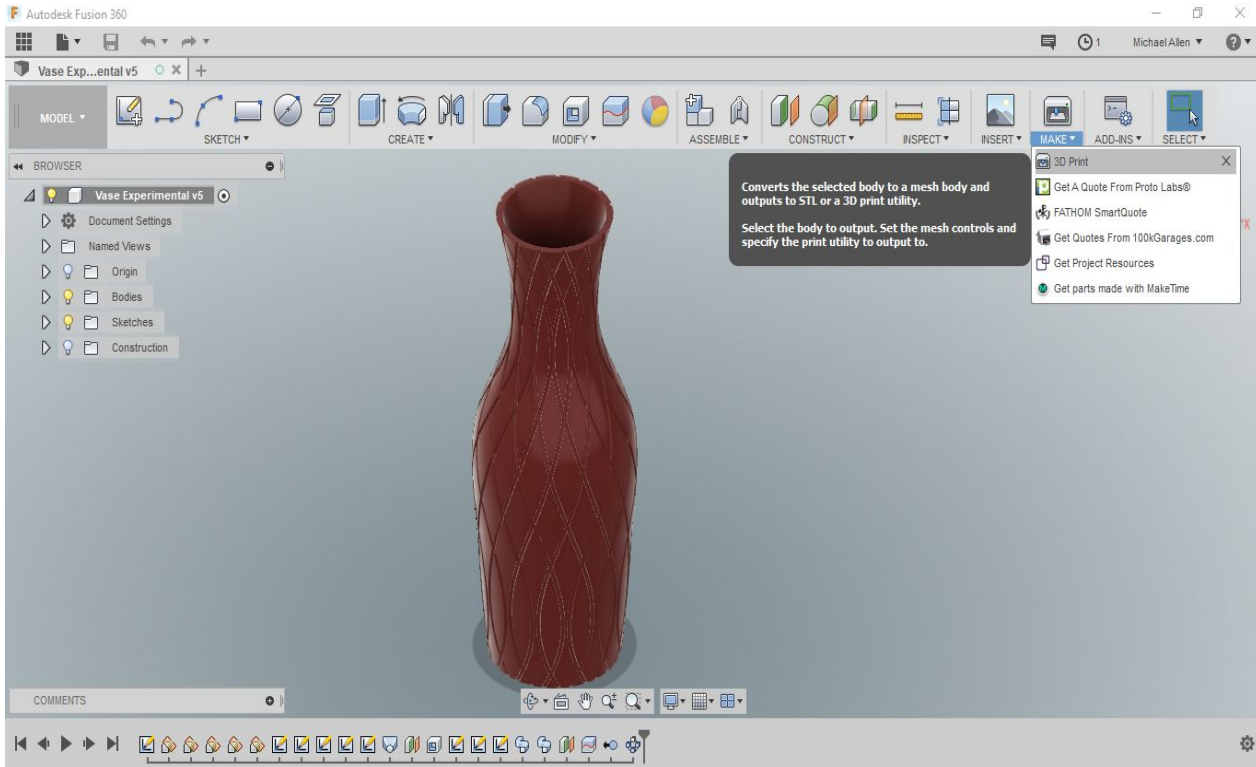
3) Chose the object(s) ready for export by left-clicking on the model or in the browser. Choose refinement and unselect 3D Print Utility. Select Ok and Save to desired destination

SketchUp Pro 2018

1) Open a Template that uses "Units: Millimeters" and complete your model
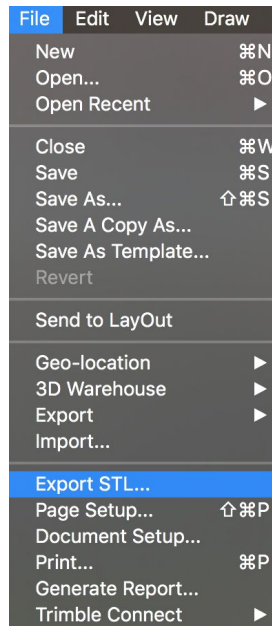2) Click on the "Extension Warehouse" that looks like a diamond in a box

3) Search "STL" in the search field and install "SketchUp STL" by the SketchUp Team

SketchUp STL by SketchUp Team
Import and Export STL files for 3D printing

4) Select File > Export STL

5) Export Unit: "Millimeter" and File Format: "Binary" to keep your file easier for the slicer to read

6) Name your file and save to desired destination