How to Create STL Files

Contents

When creating STL files for Rapid Prototyping a number of key considerations should be taken into account when converting CAD data to STL format, in order to ensure the final prototype model received matches expectations.

4 Key considerations for creating STL files

1. <u>Faceting & Smoothness</u>
Faceting refers to the relative coarseness or smoothness of a curved area. Most CAD packages allow you to control faceting through chord height, angle control and angle tolerance. Excessively fine faceting will ensure a smooth surface area but is likely to increase build times and production costs. Coarse faceting will lower production costs and build times however is likely to result in flat surfaces rather than smooth curves, for example a heavily faceted file of a pound coin would most likely create a coin similar in shape to a fifty pence piece. Ideally to keep build times and costs to a minimum without compromising on surface finish your file should be just detailed enough to allow required features to build while maintaining a manageable file size.

2. <u>Wall Thickness</u>
It is important to take into account that while modern Rapid Prototyping machinery allows the creation of high-resolution parts, certain restrictions on wall thickness still exist. Where walls fall below these minimum levels holes, missing pieces or weak walls may appear on the final prototype model. When sourcing prototypes it is therefore advisable to use the below wall thickness guideline:

   SLA: No less than 0.5mm
   High Resolution SLA: No less than 0.3mm
   SLS: No less than 0.5mm-0.8mm (dependent on part geometries)

3. <u>Nested/Tabbed Parts</u>
It is advisable to save nested or assembly parts as individual pieces as this will not only affect the rapid turnaround of your quote but also ensure your build can be placed in the optimal position on the build platform for part quality, cost and build time. It is also recommended to avoid supplying tabbed files as part clean up can prove difficult reducing quality of the final prototype model. While some users believe the provision of tabbed parts will actually lower the cost of prototyping the opposite is in fact true as we will always position parts on the build to ensure best possible pricing and build quality.

4. <u>Surfaces, Edges & Inverted Angles</u>

   Prior to sending your files to LPE you should check for any missing surfaces, bad edges and inverted normals or overlapping surfaces. LPE technicians will check for these issues upon receipt of files however we do not have the same working knowledge of the final design and may not always spot these issues. To avoid issues early on we recommend the use of a STL viewer prior to submitting your final files (STL viewers are available free online).

How to save files in STL format

How to save your files in STL format depends on your chosen software package. View the instructions for yours below.

a) IDEAS

1. Select FILE > EXPORT > RAPID PROTOTYPE FILE.
2. Click on the model to be converted to STL format.
3. Select the following settings from the dialogue box:

– Prototype Device: SLA500.DAT
– Part positioning: Origin
– Absolute facet deviation: 0.01
– Output file type: Binary

4. Select OK.

b) AutoCAD

1. Design to be 3D solid object.
2. Set FACETRES = 10.
3. STLOUT to export STL file.

c) Mechanical Desktop

1. Use the AMSLOUT command to export the file.

Note:

– Angular Tolerance: reducing the default by 15 degrees increases the STL file resolution.
– Aspect Ratio: controls the height/width ratio of the facets. A setting of 1 would mean the height of a facet is no greater than its width. Default is 0 (ignored).
– Surface Tolerance: controls the greatest distance between the edge of a facet and the actual geometry. A setting of 0.000 causes the option to be ignored.
– Vertex Spacing: controls the length of the edge of a facet. Default setting is 0.000 (ignored).

d) ProEngineer

1. Select File > Export > Model > STL Menu.
2. Use the default on the pop up menu for Coordinate system, and Binary Format. Allow negative values.
3. At the Chord Height prompt type 0.01 and press ENTER.
4. If Chord Height changes to a value between;
– 0.01 – 0.02mm: enter 0 as the Angle Control
– 0.02 – 0.04mm: enter 0.5 as the Angle Control
– 0.04 – 0.05mm: enter 1 as the Angle Control
We recommend Chord Height of between 0.01 – 0.02mm.
5. Click OK.

Notes:

- Good accuracy and overall physical size of a part will affect quality of STL file.
- Default accuracy (0.0012) is usually satisfactory for parts that can be contained in a cube of 100 x 100 x 100mm and which possesses the smallest radii feature of R0.5. For larger parts or parts with a smaller radii feature we recommend increasing the default accuracy prior to or near the start of the modelling process.

e) Solidworks

1. Select Tools > Options > Export.
2. STL Options:
– Set Quality to Fine.
3. File > Save As
– Set Save as Type to STL.
4. Save.

f) Solid Designer

1. Select File > External > Save STL.
2. Select Binary Mode.
3. Select Part.
4. Enter Filename.
5. Enter Max. deviation distance of 0.01mm.
6. OK.

g) Solid Edge

1. Select File > Save As.
2. Set Save As type to STL.
3. Options:
–Set conversion tolerance to 0.001in or 0.0254mm
–Set surface plane angle to 45.00

h) Unigraphics

1. Select File > Export > Binary Prototyping.
2. Set output type to BINARY.
3. Set Triangle Tolerance to 0.0025.
4. Set Adjacency Tolerance to 0.12.
5. Set Auto Normal Gen to On.
6. Set Normal Display to Off.
7. Set Triangle Display to On.