

PROTO3000

3D Engineering Solutions

STL File Guidelines

The following are general guidelines in producing adequate STL files for rapid prototyping and 3D printing. These are only guidelines, and procedures may vary according to software version and interface. For the best possible files, please contact Proto3000 today.

Alibre

- File
- Export
- Save As >STL
- Enter File Name
- Save

AutoCAD

- Your design must be a three-dimensional solid object to output an STL file.
- Make sure the model is in positive space
- Set Facetres to 10
- At the command prompt type STLOUT
- Select Objects
- Choose Y for Binary
- Choose Filename

CadKey / KeyCreator

- Choose Stereolithography from Export options
- Enter the filename
- Click OK

Catia

STL parts created as solids should be fine. If the part was created

as volumes from surfaces may create problems. We recommend using the CATIA toolbox to analyze all STL's once they have been generated

- Select STL Command
- Maximum Sag = .0003" or .0125 mm
- Select part(s) to be converted and Click YES
- Select Export
- Type filename and output the STL.

Inventor

- Save Copy As
- Select STL
- Choose Options >Set to High
- Enter File Name
- Save

IronCAD

- Right Click on the part
- Part Properties >Rendering
- Set Facet Surface Smoothing to 150
- File >Export
- Choose .STL

Mechanical Desktop

- Use the AMSTLOUT command to export your STL file.

- The following command line options affect the quality of the STL and should be adjusted to produce an acceptable file.
- Angular Tolerance - This command limits the angle between the normals of adjacent triangles.

- The default setting is 15 degrees. Reducing the angle will increase the resolution of the STL file.

- Aspect Ratio - This setting 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. The default setting is 0, ignored.
- Surface Tolerance - This setting controls the greatest distance between the edge of a facet and the actual geometry. A setting of 0.0000 causes this option to be ignored.

- Vertex Spacing - This option controls the length of the edge of a facet. The default setting is 0.0000, ignored.

ProE

- File >Save a Copy
- Set type to STL
- Set chord height to 0. The field will be replaced by minimum acceptable value.
- Set Angle Control to 1
- Choose File Name
- OK

SDRC I-DEAS

- FILE / EXPORT / RAPID PROTOTYPE / OK
- Select the Part
- Select: PROTOTYPE DEVICE / SLA500.dat / OK
- Absolute Facet Deviation: .0004
- Select Binary / OK

Rhino

- File >Save As
- Select File Type >STL
- Enter a name for the STL file.
- Save
- Select Binary STL Files

SolidDesigner

- File >External >Save STL
 - Select Binary mode
 - Select Part
 - Enter 0.001mm for Max Deviation Distance
 - Click OK
-

SolidEdge

- File >Save As
- Set Save As Type to STL
- Options
- Set Conversion Tolerance to 0.001in or 0.0254mm.
- Set Surface Plane Angle to 45.00
- Save

SolidWorks

- File >Save As
- Set Save As Type to STL
- Options >Output As=Binary, Unit = Inches
- Resolution > Custom, Deviation Tolerance = Minimum (Slide bar to right), Angle Tolerance = 5 degrees
- OK
- Save

Think3

- File >Save As
- Set Save As Type to STL
- Save

Unigraphics

- File >Export >Rapid Prototyping
- Set Output type to Binary
- Set Triangle Tolerance to 0.0025
- Set Adjacency Tolerance to 0.12
- Set Auto Normal Gen to On
- Set Normal Display to Off
- Set Triangle Display to On