

STL Export Parameters

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

Inventor

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

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

Rhino

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

SolidWorks

- File >Save As
- Set Save As Type to STL
- Options >Resolution >Fine >OK
- 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

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

SolidDesigner

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