

STL Files:

The .STL file format has become the Rapid Prototyping industry's defacto standard data transmission format and is the format required to interact with Skeg's Printing machine. This format approximates the surfaces of a solid model with triangles. The more complex the surface, the more triangles produced as shown in figure 1.

Almost all of today's CAD systems are capable of producing an STL file. For the user the process is often as simple as selecting File, Save As, and STL. Below are steps for producing high quality STL files from a number of today's leading CAD systems. In all cases, export your STL file as a Binary file. This saves on time and file size.

As a general rule, changing options such as Chord Tolerance or Angular Control will change the resolution on your STL file. The larger the STL file, the more triangles placed on the surface of the model.

Also, for most RP processes and materials, the minimum feature thickness is 0.4mm. Anything less than this, features will not develop. Please check your models and make appropriate changes to critical features.

(Please note, these are only general guidelines and may not work or produce the best possible STL file in some cases. Please consult your users guide or the software's developers for more information or technical support. Should we determine that your STL file is not adequate for production we will contact you for an updated file.)

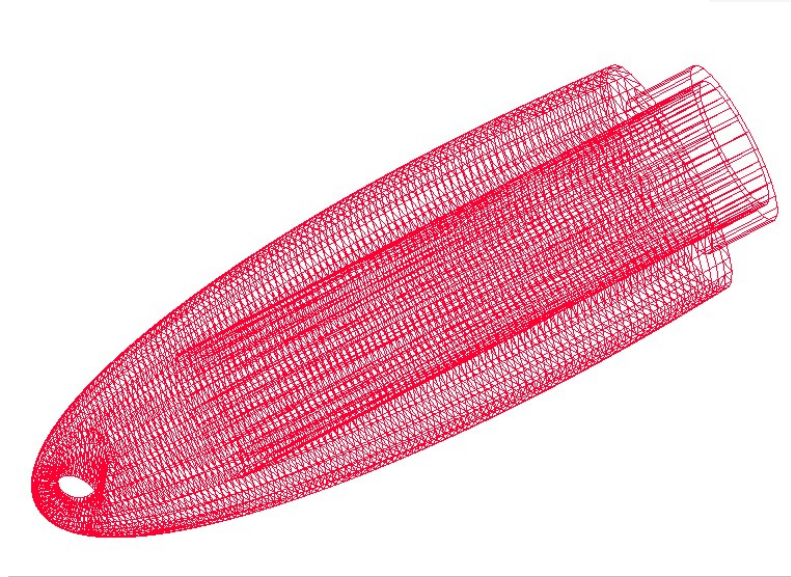


Figure 1



How to save a STL file:

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

Choose Stereolithography from Export options
Enter the filename
Click OK

I-DEAS

File >Export >Rapid Prototype File >OK
Select the Part to be Prototyped
Select Prototype Device >SLA500.dat >OK
Set absolute facet deviation to 0.000395
Select Binary >OK

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



How to save a STL file:

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

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



How to save a STL file:

SolidWorks

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

