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STL FILE CREATION

The STL file is the Rapid Prototyping & Rapid Manufacturing industries standard data transmission output. It drives all of the RP machines available today including SLA, SLS, POLYjet, FDM, EOS, etc... This format approximates the surfaces of a solid model with triangles. For a simple model such as the box shown in figure 1 the surfaces can be approximated with twelve triangles, as shown in figure 2. The more complex the surface, the more triangles produced as shown in figure 3.

Most CAD systems today are capable of producing an STL file. 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 and not ASCII. The Binary format is much smaller and easier to work with.

As a general rule, changing options such 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. For simple geometry (not a lot of curves) the file may only be a couple hundred kilobytes. For complex models, files sizes in the range of 1-5MB will produce good parts. Files larger than 5MB are unnecessary and the RP machine will not produce a better quality part.

The minimum feature thickness for most RP processes and materials is 0.020". Any features less than this thickness have a chance of not building correctly. Check your models and make appropriate changes to critical features which we refer to DFP, "design for prototype. One possible exception is High Resolution POLYjet where the machine builds at 16 microns which is .00063 or 6 tenths.

(Please note that the mentioned above are general guidelines and may not necessarily produce the best possible STL file. Please consult your software user guide for specific information or technical support. If we determine that your STL file is not sufficient for RP processing we will contact you accordingly.)

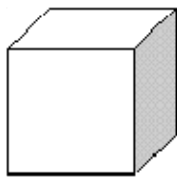


Figure 1

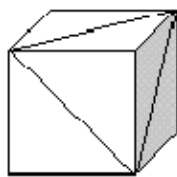


Figure 2



Figure 3

Catia

1. Select STL command
2. Maximum Seg to 0.0125mm
3. Select the model and > select Yes
4. Select Export > type File name > OK

AutoCAD

1. Make sure the model is in positive space
2. Set Facets to 10
3. At the command prompt type STL out
4. Select Objects
5. Choose Y for Binary
6. Choose Filename

CAD-Key

1. Choose Stereolithography from Export options
2. Enter the filename
3. Click OK

I-DEAS

1. File > Export > Rapid Prototype File > OK
2. Select the Part to be Prototyped
3. Select Prototype Device > SLA5000.dat > OK
4. Set absolute facet deviation to 0.000395
5. Select Binary > OK

Inventor

1. Save Copy As
2. Select STL
3. Choose Options > Set to High
4. Enter File Name
5. Save

Iron CAD

1. Right Click on the part
2. Part Properties > Rendering
3. Set Facet Surface Smoothing to 150
4. File > Export
5. Choose .STL

Mechanical Desktop

1. Use the AMSTLOUT command to export your STL file.
2. 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.

Pro-E

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

Rhino 3D

1. File > Save As
2. Select File Type > STL
3. Enter a name for the STL file.
4. Save
5. Select Binary STL Files

Solid Designer

1. File > External > Save STL
2. Select Binary mode
3. Select Part
4. Enter 0.001mm for Max Deviation Distance
5. Click OK

Solid Edge

1. File > Save As
2. Set Save As Type to STL
3. Options
4. Set Conversion Tolerance to 0.001in or 0.0254mm.
5. Set Surface Plane Angle to 45.00
6. Save

SolidWorks

5. File > Save As
6. Set Save As Type to STL
7. Options > Resolution > Fine > OK
8. Save

Unigraphics

1. File > Export > Rapid 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