

EXPORTING STL FILES

Alibre

1. File
2. Export
3. Save As > STL
4. Enter File Name
5. Save

AutoCAD (Versions: R14-2000i)

Your design must be a three-dimensional solid object to output an STL file.

1. At the command prompt type "STLOUT"
2. Select Objects
3. Choose "Y" for Binary
4. Choose Filename

CATIA V5

1. Open one part at a time, not valid for assemblies
2. Open the file menu, scroll down to save as...
3. click new document
4. choose .STL format
5. Choose resolution of .001

I-DEAS

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

INVENTOR

1. Open the 3D design in 'part mode'
2. Open the 'File' menu and select 'save as'
3. At the bottom of the window, choose 'STL file type (*.stl)'
4. Click on OK

IronCAD

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

EXPORTING STL FILES (cont'd)

ProE

1. File > Export > Model
2. STL
3. Set chord height to 0. The field will be replaced by minimum acceptable value.
4. Set Angle Control to 1
5. OK

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.

Argon, Coblat or Xenon

1. Select the parts for conversion to mesh;
2. Choose a resolution for output (Edit – change resolution). The recommended settings are either very fine or super fine. This will automatically choose the correct default settings in the mesh parameters dialog box.
3. Go to EDIT – Change object type. Choose Mesh . . . OK.
4. Make sure that the “STL Facets” is selected in the mesh Parameters dialog.
5. Select OK

Rhino

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

SolidDesigner (Version 8.x)

1. File > Save
2. Select File Type > STL
3. Select Data
4. Click OK

EXPORTING STL FILES (cont'd)

Think3

1. File > Save As
2. Set Save As Type to STL
3. Save

SolidEdge

1. File > Save As
2. Set Save As Type to STL
3. Options export .0005
4. Set conversion Tolerance to Inches of Millimeters
5. Save

SolidWorks

1. File > Save As
2. Set Save As Type to STL
3. Options > Fine > OK
4. 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

(Please note: These are only general guidelines and may not work or produce the best possible STL file in some cases. Please consult your user's 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.)