



Outputting STL and VRML Files for Z Corp 3D Printers

Alibre

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

Autodesk 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

Autodesk Inventor

1. Go to the **File** menu, then select the **Save Copy As...** choice
2. Select **STL** from the **Types** drop-down
3. Click the **Options** button, and choose the **High** detail level
4. Click the **Save** Button.

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

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

IronCAD

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

Pro/E – STL Output

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

Pro/E – VRML Output

1. Perform a “save a copy” and select the VRML output and select “ok”.
2. In the Export VRML window select the “setup” button in the corner.
3. In the setup window select the “quality” pull-down and select “high”

Pro/E Wildfire – VRML Output

1. In Wildfire, from the tools menu, select OPTIONS.
2. In the lower left Option window, type in "Vrml_export_version"
3. In the "Value" window just to the right, the drop down needs to be set to 1.0.
4. Once the value has been set to 1.0, click on the "Add/Change" button, then OK.

To adjust the resolution

1. In Wildfire, from the tools menu, select OPTIONS.
2. In the lower left Option window, type in "Vrml_export_resolution"
3. In the "Value" window just to the right, the drop down needs to be set to “High”.
4. Once the value has been set to “high” click "Add/Change" button, then 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

SolidEdge

1. Open your model and from the **File** menu choose **Save As**.
2. Select the file type **STL**.
3. Click on the **Options** button in the **Save As** dialog box.
4. Set the **Conversion Tolerance** to **0.001 inches** or **0.0254 mm**.
5. Set the **Surface Plane Angle** to **45°**.
6. Select the **Binary** radio button and click the **OK** button.
7. Name and **Save** your STL file.

SolidWorks – STL Files

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

SolidWorks – VRML Files

1. Tools > Options menu
2. Select "Document Properties" tab at the top
3. Select "Image Quality" on Left side
4. Move "Resolution" slider bar to "high"
5. If working with an assembly you must select "Apply to all referenced part documents" first.
6. Select OK
7. Select File > Save As
8. Select .VRML format for "Save as type"
9. Select Options
10. Select "VRML 1.0" on the Version Drop down list
11. Select "OK"
12. Select "Save"

Think3

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

CATIA version four

1. Select STL command.
2. Set Maximum Sag to 0.0005 inches or 0.0125 mm.
3. Pick the model to be converted to STL.
4. Select Yes to generate triangles.
5. Select Export, type a file name, and click OK to output an STL file.

Note: CATIA has a utility for checking and analyzing STL files. It is a good practice to check the direction of normal vectors (they should point outward) and verify that the STL surface is closed. Parts created as CATIA solids should be no problem. Parts created as volumes from surfaces are more likely to have flaws.