

Digital Manufacturing Process



Saving CAD files in STL format

RedEye On Demand
8081 Wallace Road
Eden Prairie, MN 55344

phone: 952-906-2725
toll free: 866-882-6937
email: godigital@redeyeondemand.com

www.redeyeondemand.com

Saving your CAD file in STL format

Most CAD packages will have a couple of options that affect the quality of the STL. Changing a "Deviation" type of value will alter the overall output or tessellation. Changing an "Angle Tolerance" type of value will alter smaller details in your file. The tighter these parameters, the more triangles placed on the surface of the model. Simple geometries tend to be a few hundred kilobytes in size. Complex models will range from 1-5MB in size and still produce good parts. For many models, files larger than 5MB may be unnecessary and often result in more time to get your quote and models back. In all cases, export your STL file as a binary file. This saves on time and file size.

Please note, these are general guidelines and may not work or in some cases, produce the best possible STL file. Please consult your user's guide or your software developer for more information or technical support.

Click on the CAD software you use to be taken to step-by-step directions:

[3D Studio Max](#)

[ADT](#)

[Alibre](#)

[AutoCAD](#)

[CADKey](#)

[I-DEAS](#)

[Inventor](#)

[Mechanical Desktop](#)

[ProE](#)

[ProE Wildfire](#)

[Revit](#)

[Rhino](#)

[SolidDesigner](#)

[SolidEdge](#)

[SolidWorks](#)

[Think3](#)

[Unigraphics](#)

[Alibre](#)

- File
- Export
- Save As > STL
- Enter File Name
- Save

STL settings: How to change STL settings and Redeye's recommendation.

- Tools > Options
- File Types tab
- Configure File Types: STL
- Normal Deviation: 5

(Smaller deviations will produce a smoother file, but the file size will get larger)

AutoCAD (Versions: R14-2000i), 2009, 2010

- At the command prompt type "FACETRES"
- Set FACETRES between 1 and 10. (1 Being low resolution and 10 high resolution for STL Triangles)

Note: RedEye recommends using a higher resolution for small parts and parts with a lot of detail. For larger parts, dimensions around 20 inches or more, use a medium resolution.

- At the command prompt type "STLOUT"
 - Select Objects
 - Choose "Y" for Binary
 - Choose Filename
-

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
-

IronCAD

- Right Click on the part
 - Part properties > Rendering
 - Set Facet Surface Smoothing to 150
 - File > Export
 - Choose .STL
-

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 triangle. 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 > Export > Model
 - STL
 - Set chord height to 0. The field will be replaced by minimum acceptable value.
 - Set Angle Control to 1
 - OK
-

ProE Wildfire

- File > Save a Copy > Model
 - Change type to STL (*.stl)
 - Set Chord Height to 0. The field will be replaced by minimum acceptable value.
 - Set Angle Control to 1
 - OK
-

Rhino

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

SolidDesigner (Version 8.x)

- File > Save
 - Select File Type > STL
 - Select Data
 - OK
-

SolidEdge

- File > Save As
- Set Save As Type to STL
- Options
- Set Conversion Tolerance to Inches of Millimeters
- Save

SolidWorks

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

STL settings: Redeye's recommendation for smaller parts and parts with many contoured shapes and detailed features.

- File > Save As
- STL > Options
- For a smoother STL file, change the Resolution to Custom
- Change the deviation to 0.0005in (0.004 mm)
- Change the angle to 5

(Smaller deviations and angles will produce a smoother file, but the file size will get larger)

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
-

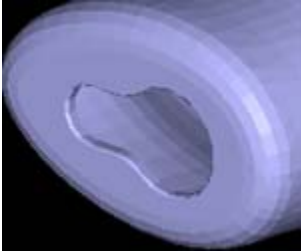
CADKey

- Choose Stereolithography from Export options
- Enter the filename
- Click OK

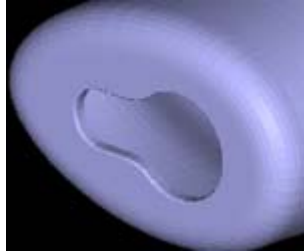
Inventor

- Save Copy As
- Select STL
- Choose Options > Set to High (for highest quality surface)
- Enter File Name
- Save

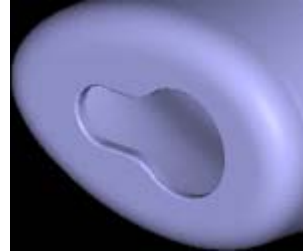
Note: The "High" setting will also produce the largest file size. From Low, Medium to High, the Hairdryer sample file in Inventor went from about 6.7MB to 17.6MB to 50MB. Below are images from this sample file with the various settings:



Low



Medium



High

3D Studio Max

- First check for errors
- An STL object must define a complete and closed surface. Use STL-Check modifier to test your geometry before export your object to STL.
- Select an object.
- Click <Modify>
- Click <More...>
- Select "STL-Check" under Object-Space Modifiers
- Select <Check>
- If there are no errors, continue to export the STL file by:
- <File> <Export>
- Select "StereoLitho [*.STL]" in <Save as type>
- Select location in <Save in>
- Enter a name in <File name>
- <Save>
- <OK>
- Export To STL dialog:
- Object Name: Enter a name for the object you want to save in STL format.
- Binary/ASCII: Choose whether the STL output file will be binary or ASCII (character) data. ASCII STL files are much larger than binary STL files.
- Selected Only: Exports only objects that you selected in the 3D Studio scene.

ADT

- Select AEC object . Go to 3D SOLID menu & select convert to 3D SOLID (ENTER)
 - After that you will have an option: Erase selected object [Yes/No] <Yes>: *Enter Y*
 - All the objects are converted into 3D Solid using the same procedure for each AEC objects
 - Select a single solid for STL output... (Must be ONE solid to export to STL)
 - Command entry: *stlout*
 - Select objects: *Use an object selection method and press ENTER when you finish*
 - Create a binary STL file? [Yes/No] <Yes>: *Enter Y*
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Revit

Revit doesn't allow direct export to STL files. We have to first save in dwg file and open in AutoCAD to create STL files.

- Go to 3D view
- Go to File menu , select Export CAD format
- A dialog box opens
- Select option
- Scroll down the drop down menu (3D view only) & select 3D polymesh
- Select " AutoCAD 2004 DWG " in <Save as type>
- Next open the saved file AUTO CAD
- Enter < Explode > on the command menu
- Select the object and press <enter>
- All the objects are converted into 3D solid
- Select a single solid for STL output... (Must be ONE solid to export to STL)
- Enter < stlout> or <export > on the command menu
- Select objects: Use an object selection method and press < ENTER > when you finish
- Create a binary STL file? [Yes/No] <Yes>: *Enter y*