Overview

How CAD files are exported to STL is an important process for accurate building of parts. The step by step process for converting CAD files to STL was taken straight from the mentioned companies websites.

Definition

STL is the standard file type used by most or all rapid prototyping systems. A STL is a triangulated representation of a 3D CAD model (figure 1). The triangulation of a surface will cause faceting of the 3D model. The parameters used for outputting a STL will affect how much faceting occurs (figure 2 and 3). You cannot build the model any better or smoother than the STL file, so if the STL is coarse and faceted, that is what you can expect in the final model.

In the CAD package, when exporting to STL, you may see parameters for chord height, deviation, angle tolerance, or something similar. These are the parameters that affect the faceting of the STL.You don't necessarily want to go too small. The finer the STL the larger the file is in size, which will affect processing time in Insight as well as build time. Below is some information found on the Internet regarding exporting to STL from various CAD packages.

STL Creation

Note: Please consult your user's guide or the software developers for more information or technical support. If your CAD software is not listed below, please contact your CAD software technical support for information about exporting to an STL.

3D Studio Max

1. First check for errors

2. An STL object must define a complete and closed surface. Use STL-Check modifier to test your geometry before exporting your object to STL.

3. Select an object.

4. Click Modify

5. Click More ...

6. Select "STL-Check" under Object-Space Modifiers

7. Select Check

8. If there are no errors, continue to export the STL file by:

9. Clicking File>Export

10. Select "StereoLitho [*.STL]" in Save as type

```
Solid test
Facet normal 0 1 0
Outer loop
Vertex 0 4 0
Vertex 0.517638 3.93185 0
Vertex 0.5 3.93185 -0.133975
Endloop
Endfacet
Endsolid test
```
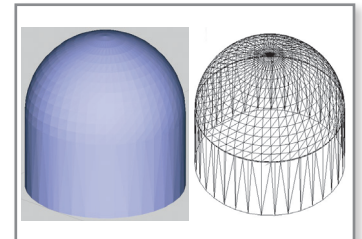
Figure 1: Sample triangulation code

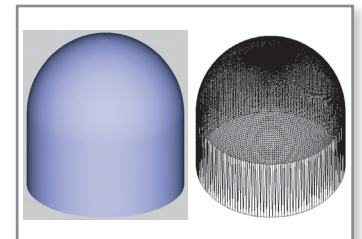*Figure 2: Example of course triangulation*

Figure 3: Example of fine triangulation

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11. Select location in Save in

12. Enter a name in File name

13. Click Save; OK

14. Export To STL dialog:

15. Object Name: Enter a name for the object you want to save in STL format.

16. 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.

17. Selected Only: Exports only objects that you selected in the 3D Studio scene.

Alibre

1. File

2. Export

3. Save As > STL

4. Enter File Name

5. Save

AutoCAD

1. Click Output > Send panel > Export. At the Command prompt, enter export.

2. In the Export Data dialog box, enter a file name.

3. Under Files of Type, select Lithography (*.stl). Click Save.

4. Select one or more solid objects. All objects must be entirely within the positive XYZ octant of the world coordinate system (WCS). That is, their X, Y, and Z coordinates must be greater than zero. The file extension .stl is automatically appended to the file name.

AutoDesk Inventor

1. Save Copy As

2. Select STL

3. Choose Options > Set to High

4. Enter Filename

5. Save

CADKEY

1. File>Export>STL
2. Type a file name and select OK
3. Change format to binary
4. Use default facet tolerance
5. Additional export tolerance options are in the Solids99 Configuration window accessed from the Tolerance section of Configuration.

DataCAD

1. In the File pull-down menu, select Export STL file.

2. Click Save

Deskartes-3D Data Expert

1. Transfer the data into 3Data Expert through IGES file

2. Give command Solid/Triangulate in 3Data Expert

3. A watertight STL file is generated

4. Surface accuracy control during the process is easy

5. To check for a good STL file:

6. Orient surface normals and check for errors with Solid/Repair command

Google Sketchup

1. Download Sketchup to DXF or STL plugin.

2. To use the plugin, download the file below (skp_to_dxf.rb) to the Sketchup plugins folder on your computer. [VERSION] stands for the Sketchup version number (6, 7 or 8).

3. On a Windows PC: If you've installed Sketchup on the C: drive, this folder will be at C:\program files\google\google sketchup [VERSION]\plugins.

4. On Mac OSX: The sketchup plugins folder is /Library/Application Support/Google SketchUp [VERSION]/SketchUp/Plugins

5. After copying this file, start Sketchup. You should now have an extra menu option (Export to DXF or STL) in the Sketchup Tools menu.

IronCAD

1. Select Part Properties, then Rendering

2. Set facet surface smoothing to 150

3. Select File, then Export

4. Select .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 > Export > Model (or 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

Pro E Wildfire

1. File > Save a Copy > Model

2. Change type to STL (*.stl)

3. Set Chord Height to 0. The field will be replaced by minimum acceptable value.

4. Set Angle Control to 1

5. OK

Revit

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

1. Go to 3D view

2. Go to File menu , select Export CAD format

3. A dialog box opens

4. Select Option

5. Scroll down the dropdown menu (3D view only) and select 3D polymesh

6. Select "AutoCAD 2004 DWG" in Save as type

7. Next, open the saved file AUTO CAD

8. Enter "Explode" on the command menu

9. Select the object and press Enter

10. All the objects are converted into 3D solid

11. Select a single solid for STL output (must be one solid to export to STL)

12. Enter "stlout" or "export" on the command menu

13. Select objects: Use an object selection method and press Enter when you finish

14. Create a binary STL file? [Yes/No]: Enter y

RhinO

1. Open the model design in Rhino.

2. From the File menu, select Save As. The Save dialog box opens.

3. In the File name box, enter a name for the new STL file.

4. In the Save as type box, select Stereolithography [*.stl].

5. Click Save.

6. In the STL Mesh Export Options dialog box (Simple Controls), set the STL tolerance – the maximum distance allowed between the surface of the design and the polygon mesh of the STL file.

7. In the Polygon Mesh Detailed Options dialog box, set the STL tolerance in the field labeled Maximum distance, edge to surface, as shown in the figure (please refer to the STL help in Rhino for setting definitions)

8. Click OK.

9. In the STL Export Options dialog box, set the file type as Binary and click OK.

NOTE: In order to build on additive manufacturing technologies, STL files must contain completely closed (watertight) polygon mesh objects. Refer to Rhino's STL help for more information.

SolidEdge

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

1. File > Save As

2. Set Save As Type to STL

3. Options > Resolution > Fine > OK

4. Save

Think3

1. File>Save As

2. Set Save As Type to STL

3. Save

UGS NX 6

1. File > Export > STL

2. Set Output type to Binary

3. Set Triangle Tolerance to 0.001

4. Set Adjacency Tolerance to 0.001

5. Set Auto Normal Gen to On

6. Set Normal Display to Off

7. Set Triangle Display to Off

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