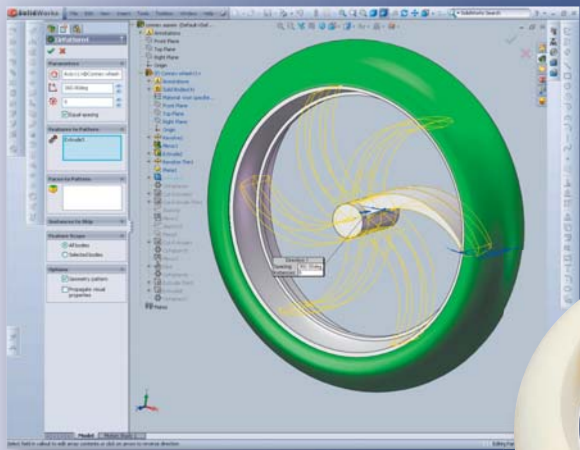
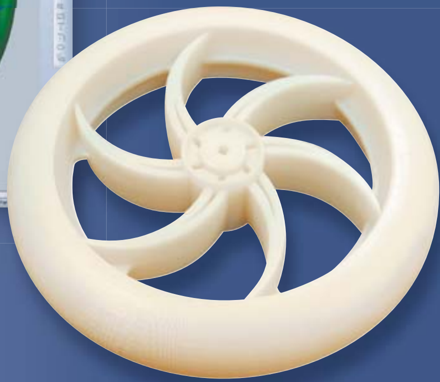




From CAD to Objet Studio Workflow for PolyJet™ Technology

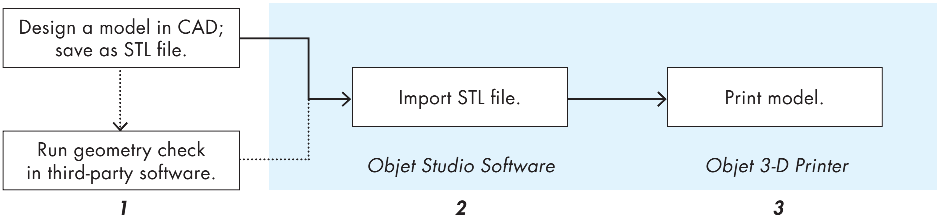
Content

CAD-Objet Studio Workflow – Overview

Understanding the interface between your CAD software and your Objet 3-D Printer enables you to get the most out of Objet's advanced PolyJet™ technology

Overview

The basic workflow for designing models in a CAD program and printing them on an Objet 3-D printer consists of three phases, as illustrated in the following figure.



Understanding STL Files

After designing a model in a CAD program, you save the design as an **STL** file. (Most CAD programs have this function.) An STL file renders surfaces in the CAD design as a mesh of triangles. The number and size of the triangles determine how accurately curved surfaces are printed. You control the number and size of the triangles by setting the following parameters when you create the STL file from the CAD design:

Chordal Tolerance / Deviation: The maximum distance between the surface of the original design and the tessellated surface of the STL triangle (see Figure 1).

Angle Control: The angular deviation allowed between adjacent triangles. This setting enables you to increase tessellation, necessary for surfaces with small radii. (The smaller the radii, the more triangles are needed).

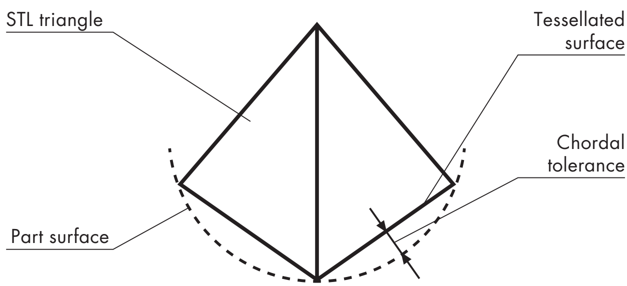


Figure 1. Tessellation of curved surfaces

The results of STL files created with different accuracy settings are shown in figures 3–5 and in the table following them.

Figure 2. CAD design

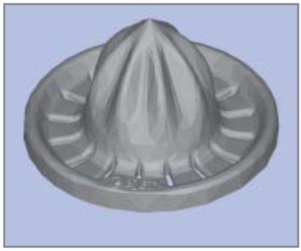


Figure 3. STL chordal tolerance = 2.0 mm

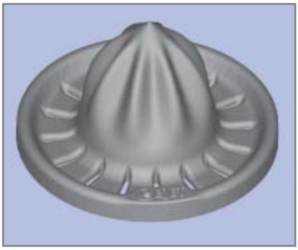


Figure 4. STL chordal tolerance = 0.1 mm

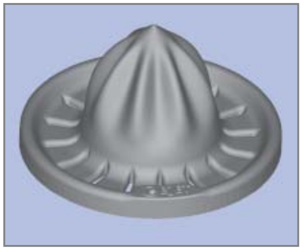


Figure 5. STL with chordal tolerance = 0.03 mm

Figure	Chordal Tolerance	Angle Control	Number of Triangles	File Size (Binary)
Figure 3	2.0 mm	30°	5080	0.25 MB
Figure 4	0.1 mm	10°	119732	5.7 MB
Figure 5	0.03 mm	5°	402746	19.2 MB

STL File Format

You usually have the option to save STL files in either binary or ASCII format. Binary files are smaller (by a factor of 6!), so this format is usually preferred. However, ASCII files can be visually read and checked.

STL Geometry Check

Model designs containing holes and gaps adversely affect the quality of the printed model. Therefore, you should perform a geometry check of the STL files before continuing. Thirdparty software for this purpose attempts to fix the geometry of problematic files.

Preparing Parts for Printing

Objet Studio™ contains a full range of tools for preparing parts for printing. You can scale models and arrange them on a virtual build tray for printing, and you can save the build tray as a single file, enabling you to quickly reprint a job.

The following pages describe, in detail, how your CAD software fits into the workflow for printing models with Objet 3-D printers.

Phase 1: Saving a Model Design in STL Format

SolidWorks allows you to save model designs in STL format, at all levels of design, for both individual parts and assemblies, including the ability to save a multi-bodied model as a single STL.

To save a model or a model assembly in STL format:

1. From the **File** menu, select *Save as.*

The *Save As* dialog box opens.

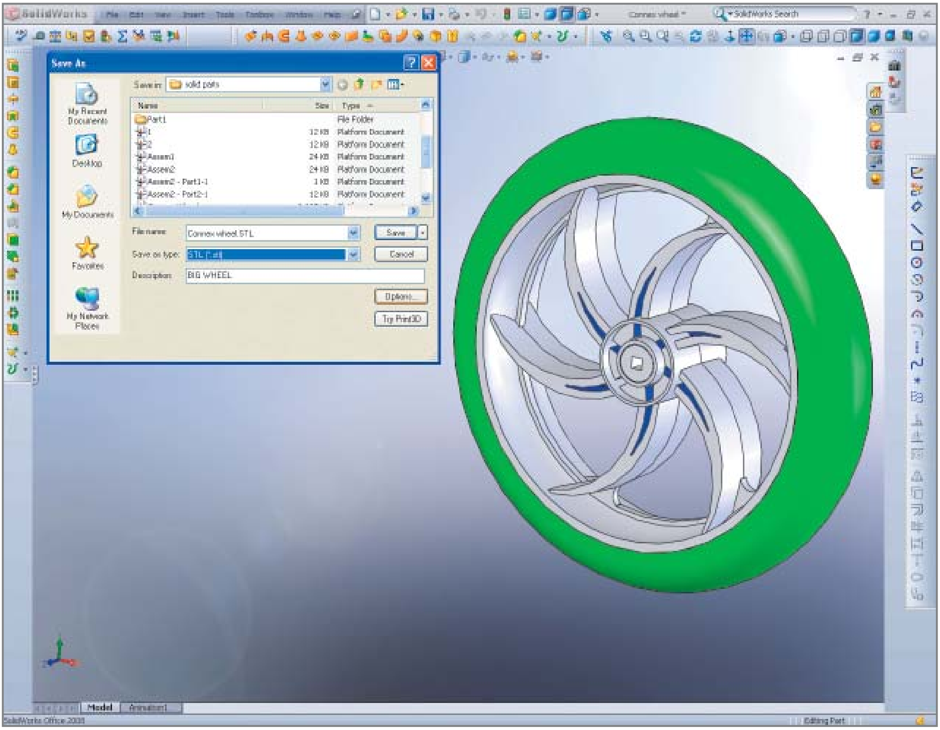


Figure 6. *Save As* dialog box in SolidWorks

2. From the *Save as type* drop-down menu, select **STL** (*.stl).

3. Click **Options**...

The *Export Options* dialog box appears, and the model is displayed in tessellated view. **STL** is the *File Format* selection (see Figure 7).

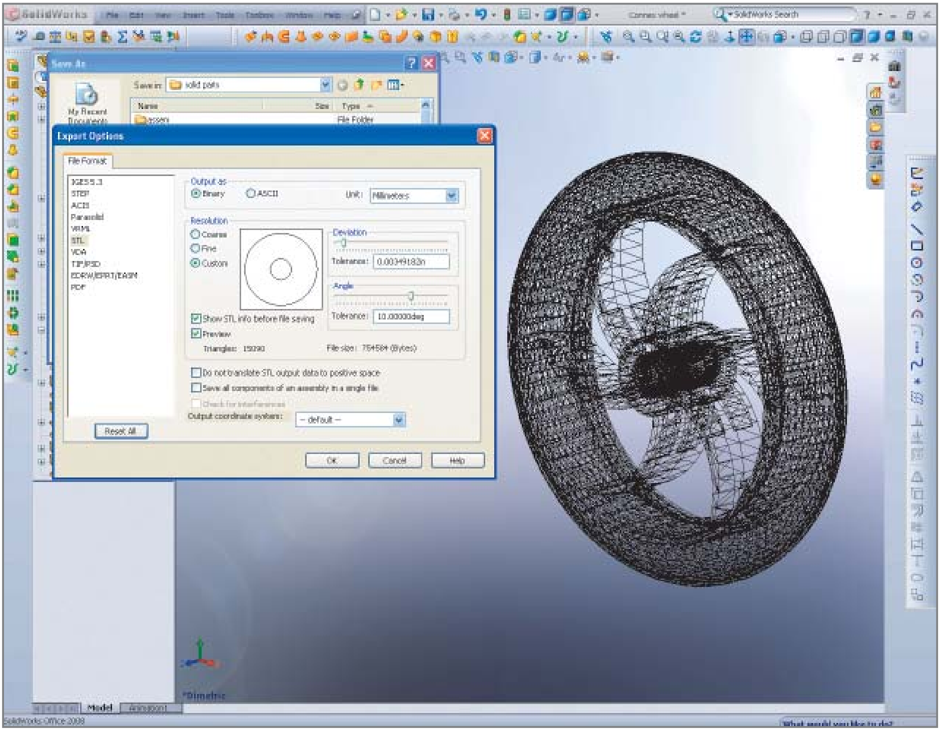


Figure 7. *Export Options* dialog box and SolidWorks screen

4. In the *Export Options* dialog box (see Figure 8), in the "Output as" section, select **Binary**. (The resulting file size will be much smaller than a file saved in ASCII format.)

5. In the "Resolution" section, select the appropriate option.

If you select **Custom**, you can manually adjust the *Deviation* and *Angle* settings, as needed. These settings affect the tessellation of non-planar surfaces, as follows:

- Lower *Deviation* settings result in finer tessellation.
- Lower *Angle* settings result in with greater accuracy, noticeable in small details.

As a rule, the higher the resolution, the larger the size of the file, and the longer

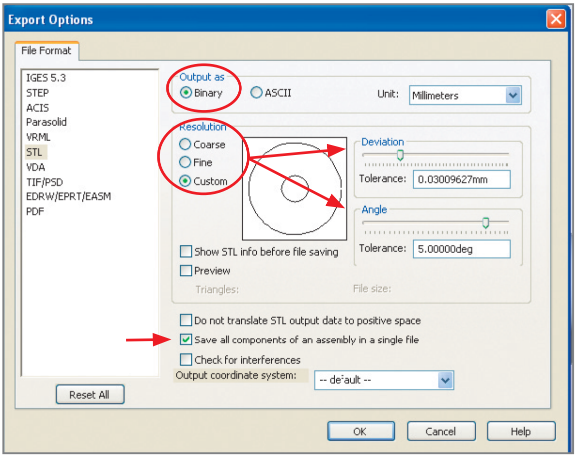


Figure 8. Export Options dialog box

6. Make sure that the following check box is selected:

   Save all components of an assembly in a single file.
   This ensures that all components are saved as a single STL file*.

7. Click **OK**.
8. In the *Save As* dialog box, click **Save**.
9. In the confirmation message, click **Yes** (see figure 9).

***Note:** This check box is *not* relevant when saving a single component model, since the model is saved as single STL file.

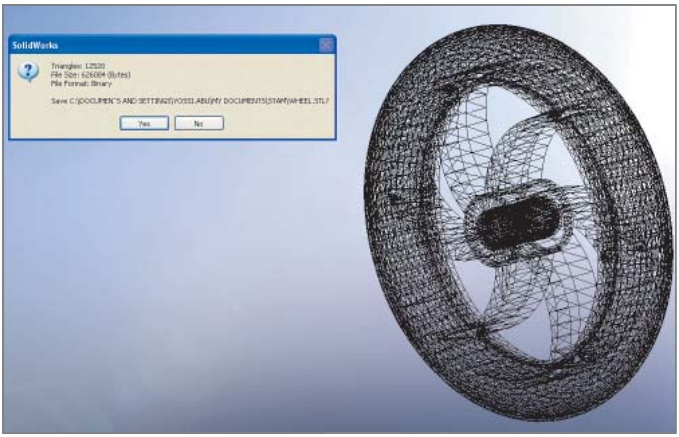


Figure 9. Save confirmation message

Phase 2: Opening model files in Objet Studio Software

To place an object on the build tray:

1. Open the *Insert* dialog box.

- From the *Object* menu, select **Insert**.
  or
- On the toolbar, click the *Insert Model* icon .
  or
- Right – click on the build tray, and select **Insert**.
  The *Insert* dialog box appears.

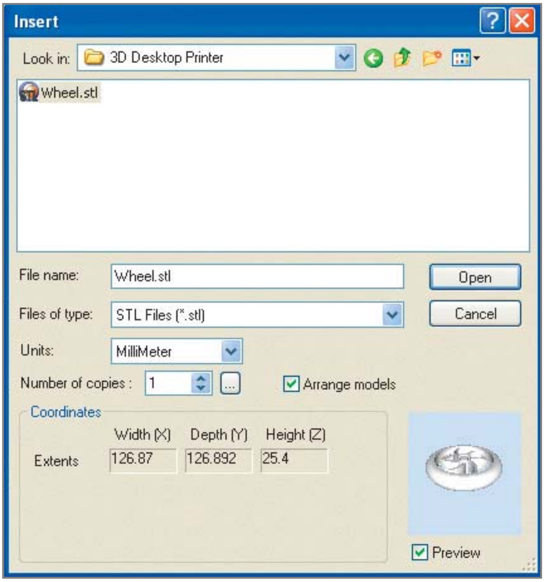


Figure 10. Insert dialog box

2. Select the desired file, and make sure that it appears in the *File name* field.

If the *Preview* check box is selected, the object is displayed in the dialog box, as shown in Figure 8.

3. Click **Open**.

Objet Studio places the object on the build tray, and in the list in the hierarchy pane (see Figure 11).

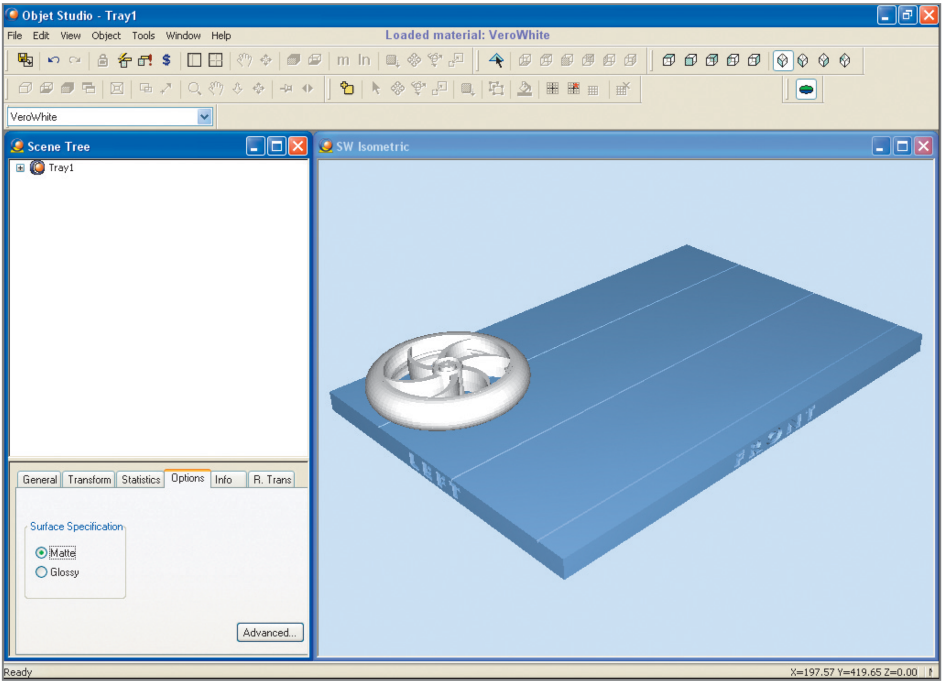


Figure 11. Default screen layout

You can place additional objects on the build tray by repeating this procedure.

Phase 3: Printing the Model

When the build tray (in *Objet Studio*) is ready to be printed, you send it to Job Manager, where it is placed in the print queue.

From the *File* menu, select **Build Tray**.

Click the *Build Tray* icon .

The Objet printer builds the model.

Figure 12: Finished model

Phase 1: Saving a Model Design in STL Format

Pro/E allows you to save model designs in STL format, at all levels of design, for both individual parts and assemblies. When dealing with assemblies, you can specify parts of an assembly to either include or exclude from the resulting STL file.

Use the procedures below for saving both parts and assemblies as STL files for printing with Objet 3D printers.

To save a Pro/E part as an STL file:

1. Check that the model design is continuous and "watertight."

This step is especially important if the design was imported from a neutral design format. Non-continuous bodies are likely to result in defective models. You cannot always check for continuity by examining the model displayed in shaded view. Therefore, use the following method:

(a) View the model with hidden lines displayed.

(b) From the *View* menu, select **Display Setting** > **Scheme** > **PreWildfire**. The model surfaces are displayed in magenta. If the design is continuous, the contour lines are white. If there are gaps, the lines are yellow.

(c) Fix the model design, if necessary, before saving it as an STL file.

2. From the *File* menu, select **Save a Copy**.

The *Save a Copy* dialog box appears.

3. From the *Type* pull-down menu, select **STL** (see Figure 13).

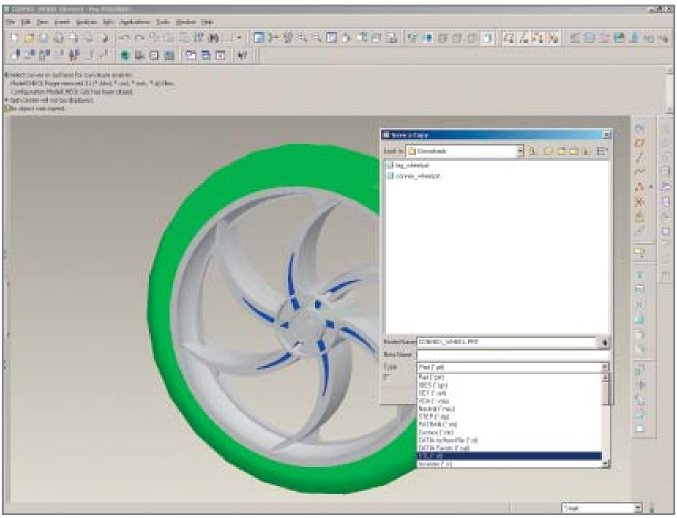


Figure 13. Saving a part as a STL file

The Export STL dialog box appears.

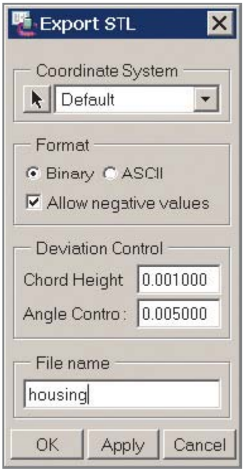


Figure 14. Export STL dialog box

Deviation Control

The *Deviation Control* settings in the Export STL dialog box affect the accuracy of the model and the size of its file.

Chord Height – Also known as "chordal tolerance," this setting specifies the maximum distance between the surface of the original design and the tessellated surface of the STL triangle (the chord), as shown in Figure 15. Therefore, the chord height controls the degree of tessellation of the model surface. The smaller the chord height, the less deviation from the actual part surface (but the bigger the file).

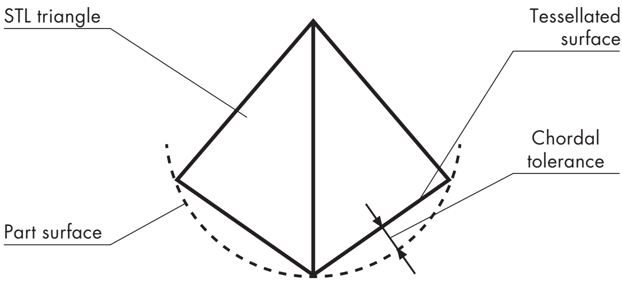


Figure 15. Chord Height

Angle Control – This setting regulates how much *additional* tessellation occurs along surfaces with small radii. The smaller the radii, the more triangles are used. The setting can be between 0 and 1. Unless a higher setting is necessary, to achieve smoother surfaces, 0 is recommended.

4. When you have made all of the required settings, click **Apply** and **OK** to create the STL file.

To save a Pro/E Assembly as an STL File:

1. From the *File* menu, select **Save a Copy**.

The *Save a Copy* dialog box appears.

2. From the *Type* pull-down menu, select **STL**.

The Export STL dialog box appears. In addition to the settings used when exporting a *part* STL, this dialog box enables you to specify the parts of an assembly to either include or exclude from the resulting STL file. In the dialog box shown in Figure 16, one of the parts of the assembly (the tire) has been excluded, leaving two parts (the hub and the main wheel) to be exported to the STL file. The design resulting from these settings (when you click **OK**) is shown on the left.

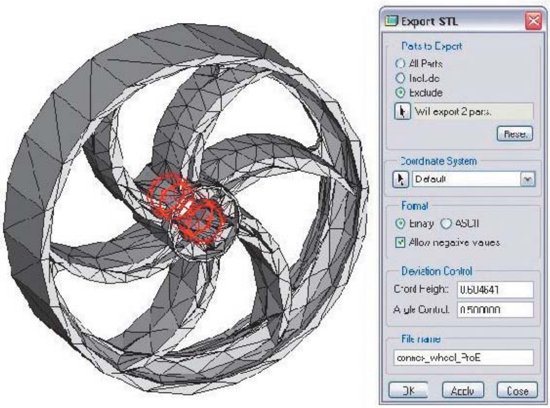


Figure 16. Export STL settings for assemblies and the resulting model saved

3. When you have made all of the required settings, click **Apply** and **OK** to create the STL file.

For phases 2–3, see pages 7–8.

Phase 1: Saving a Model Design in STL Format

Catia can import almost any design-file format, but only those that include solid data (IGES, STEP, Parasolid, etc.) can be saved as STL files – with a special add-on module. *Catia* V5 is capable of creating STL files from parts (*CatiaPART* files), but not from assemblies (*CatiaPRODUCT* files) or geometrical representations (*car* files). Therefore, source files, including those saved in a neutral format (STEP or IGES, for example), must be saved as *parts*. If the source design was saved as an assembly, it is imported to Catia as a *product*. To create an STL file from it, you must first convert it to a multi-bodied part. The procedure described below is one of several methods for doing this.

Opening and Preparing the Source File

1. From the *File* menu, select **Open**, and open the source file.

If the source design was saved as an assembly, it is imported as a *CatiaPRODUCT* model (see Figure 17).

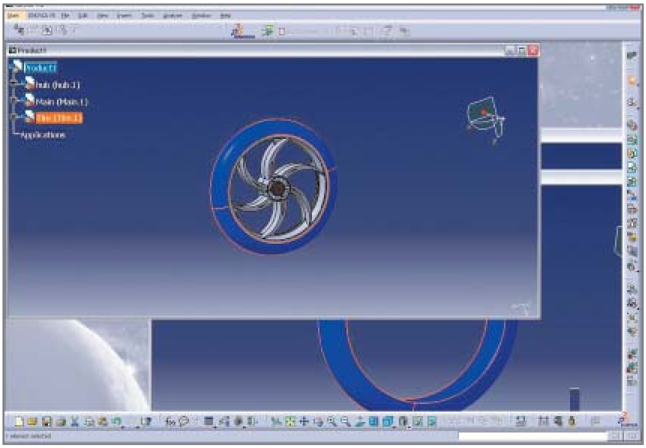


Figure 17. An assembly imported into Catia

2. Save the product file.

3. From the *File* menu, select **New > Part**, and give it the name of one of the components.

4. In the product window, select this component (Figure 17), and copy it (with the *Edit* menu or the right-click pop-up menu).

5. In the part window, paste the component.

6. Repeat steps 4 and 5 until you have copied all of the components and pasted them as individual parts.

Figure 18 shows the result of pasting the components of the product from Figure 17 into individual part files.

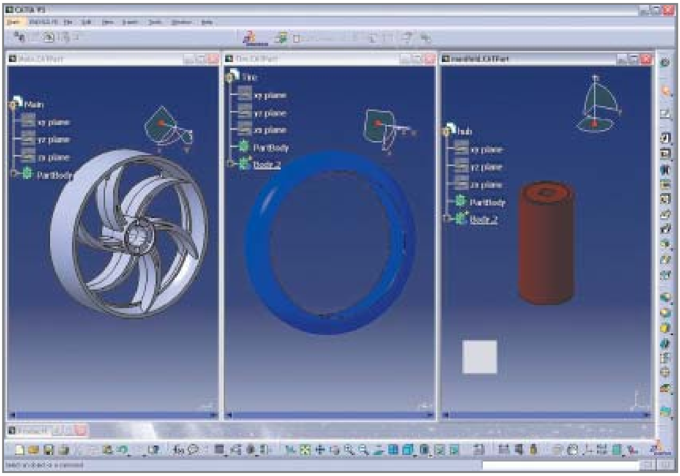


Figure 18. Assembly components saved as individual parts

7. From the *File* menu, select **New > Part**, and give it a name suitable for the combined model.

8. Copy each of the individual components (parts) from the working files (Figure 18) and paste them into the new (combined) model file (Figure 19).

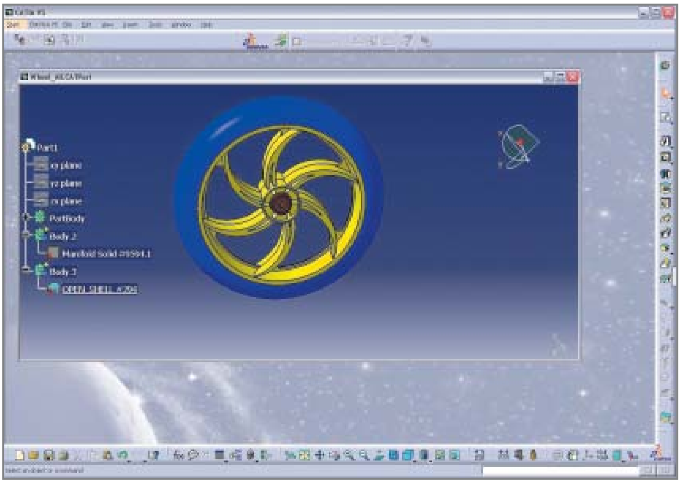


Figure 19. Part assembled from individual parts

Since the geometries of all of the parts are retained, they should be aligned correctly in the combined part. The new part is now ready to be exported as an STL file.

Re-aligning parts (if necessary)

Occasionally, because of the way the original assembly was designed, some of the components may not align correctly in the combined part. If so, you must align them, using the Constraints feature, from the *Insert* menu (see Figure 20).

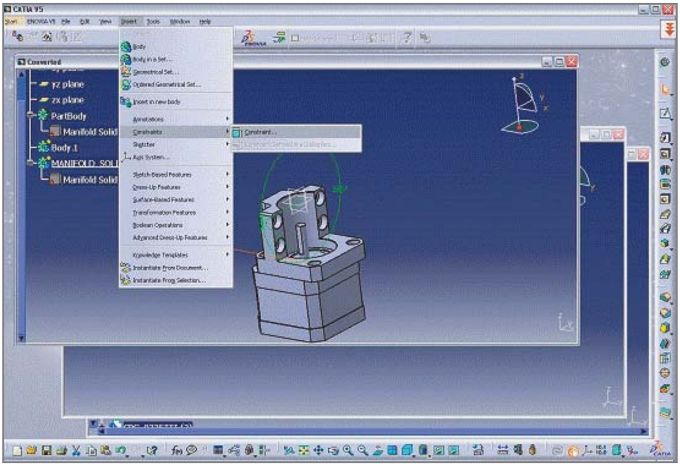


Figure 20. Re-aligning parts

Before saving the file, it is advisable to review the settings that determine the accuracy of the model – and the size of the file. To see these parameters:

1. From the *Tools* menu, select **Options**.

2. In the *Options* dialog box, display the *Performance* tab.

3. Under the *General* category (on the left), select **Display**.

4. Pay attention to the 3D Accuracy settings (see Figure 21).

Fixed – The lower the setting, the finer the details of the model in the STL file.

A very small setting results in a very large STL file.

Curves' accuracy ratio – The higher the setting, the smoother the surface will be, when dealing with complex geometries, especially if surfaces contain sudden small changes with small radii (like the bumps on a golf ball).

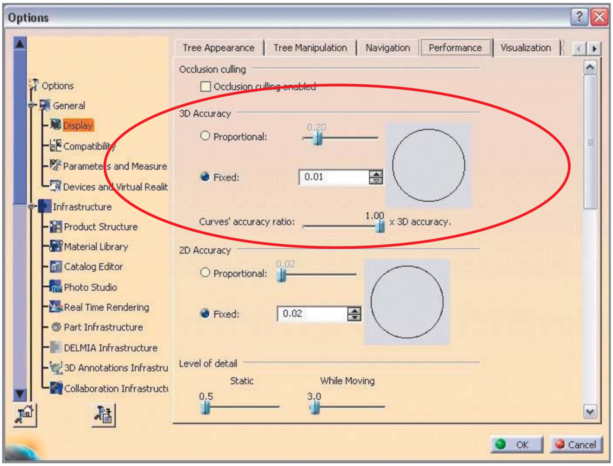


Figure 21. Setting the 3D Accuracy settings in the Options dialog box

Saving the Part as an STL File

After preparing the part, as described above, proceed as follows:

1. From the *File* menu, select **Save As**.

2. In the *Save As* dialog box, select **stl** from the *Save as type* pull-down list.

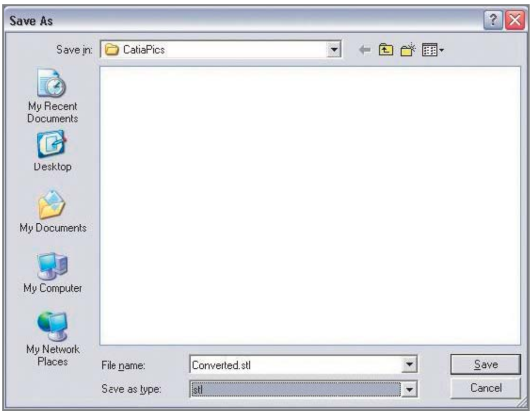


Figure 22. Saving a *Catia* file as an STL file

3. Click **Save**.

Since Catia5 supports non-continuous model designs, importing geometry into a part by copying and pasting is not problematic.

For phases 2–3, see pages 7–8.

Phase 1: Saving a Model Design in STL Format

With **IronCAD**, you can only save *parts* as STL files. When working in *assembly* mode, you must save its component *parts* as individual STL files. The procedures for doing so are described below

To save an IronCAD part as an STL file:

1. Right-click on the part and, from the pop-up menu, select **Part Properties...**

Figure 23. Selecting part properties

Note: If you opened an assembly, right-click on the part you want to save as an STL file.

The *Part* dialog box appears (see Figure 24).

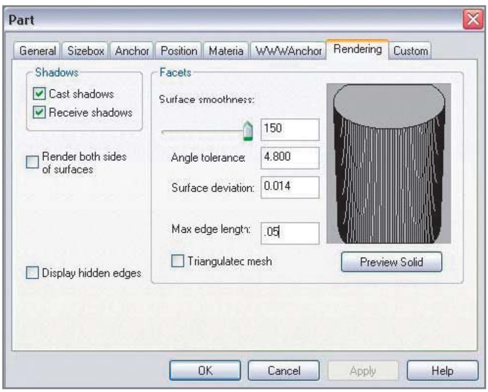


Figure 24. Opening the *Part* dialog box

Note: Setting *Surface Smoothness* affects *Angle Tolerance* and *Surface Deviation*.

2. Make sure that the *Rendering* tab is displayed (as shown in Figure 24).

3. Change the *Surface Smoothness* setting to an appropriate value for your model.

If you have not established an appropriate value, try *150*. The higher the number, the smoother the model surface will be.

4. Change the *Max edge length* setting to an appropriate value for your model.

If you have not established an appropriate value, try *0.05*. This setting produces good results, but increases file size and may require several minutes to render the model to STL format.

5. To create smoother model surfaces, when designing spherical and torus geometries, select the *Triangulated mesh* check box.

Selecting this check box results in larger STL files, but may produce smoother curves in models. If the surfaces of the model design are planes, this setting does not improve the results.

6. Click **OK** to save the settings and close the dialog box.

7. From the *File* menu, select **Export > STL**.

8. In the *Stereolithography Write* dialog box, make sure **PC** is selected, and select the *Binary output* check box.

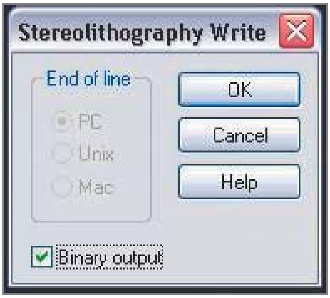


Figure 25. *Stereolithography Write* dialog box

9. Click **OK** to save the settings and create the STL file.

For phases 2–3, see pages 7–8.

Phase 1: Saving a Model Design in STL Format

Rhino enables extensive control of STL properties when saving designs as STL files. Because Rhino software is surface-based, the complete model design (even if an assembly) is saved as a single STL part

To save a Rhino model design as an STL file:

1. From the *File* menu, select **Save As**.

The *Save* dialog box opens.

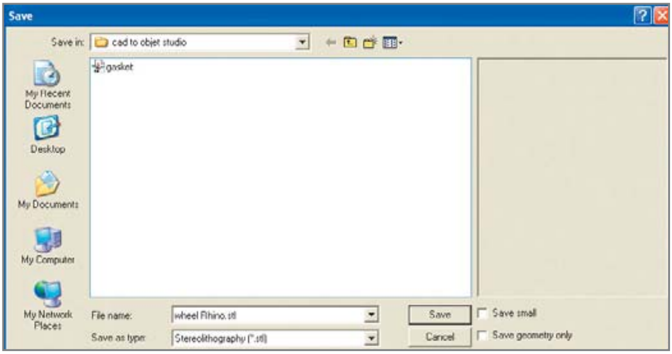


Figure 26. *Save* dialog box

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

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

4. Click **Save**.

5. In the *STL Mesh Export Options* dialog box, set the STL tolerance – the maximum distance allowed between the surface of the design and the polygon mesh of the STL file.

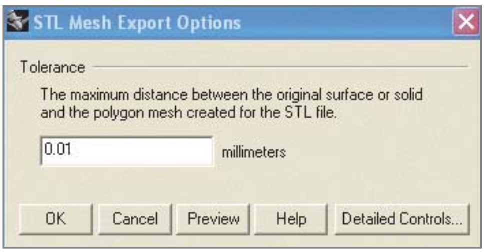


Figure 27. STL tolerance setting in the *STL Mesh Export Options* dialog box

Note: The *Detailed Options* dialog box (Figure 28) may open instead of the *STL Mesh Export Options* dialog box. You can display each of these dialog boxes by clicking **Detailed Controls…** and **Simple Controls…**, alternately.

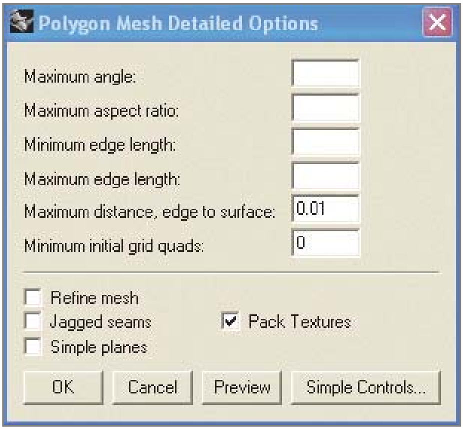


Figure 28. STL tolerance setting in the *Polygon Mesh Detailed Options* dialog box

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.

If you do not know the other settings appropriate for your model design, try these:

Maximum angle – clear
Maximum aspect ratio – clear
Maximum distance edge to surface (Tolerance) – less than half of the printer's resolution. For example, the setting shown in the figures above (0.01 mm) is a good setting for printing models at a resolution of 30 μm (0.03 mm).

6. Click **OK**.

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

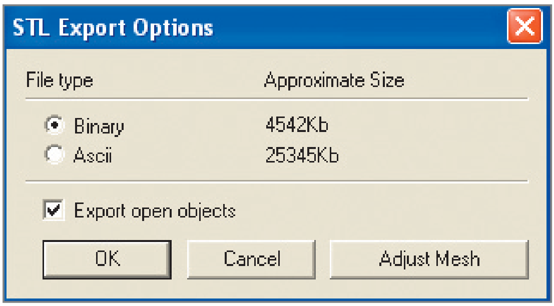


Figure 29. *STL Export Options* dialog box

If the *Export open objects* check box is selected, STL files will be created for each of the objects currently open. If this check box is cleared, an STL file is created for the selected object.

Important: STL files are suitable for 3-D printing if the models they describe are "watertight" – that is, they do not contain holes or gaps. If the following message appears, click **Cancel** and fix the model design (see instructions below) before saving it as an STL file.

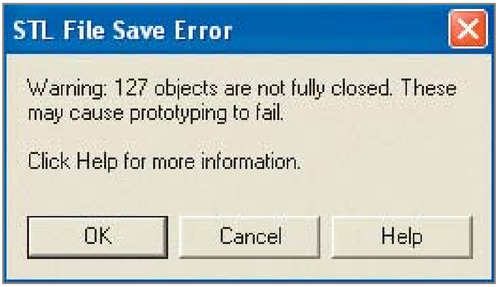


Figure 30. STL warning message

For phases 2–3, see pages 7–8. To troubleshoot the model design and make it suitable for STL conversion, continue below.

Troubleshooting Model Designs

If a model design contains holes or gaps, it is not suitable for 3-D printing. Before saving it as an STL file, you must make it "watertight."

To close holes and gaps in a model design:

1. Select the entire object.

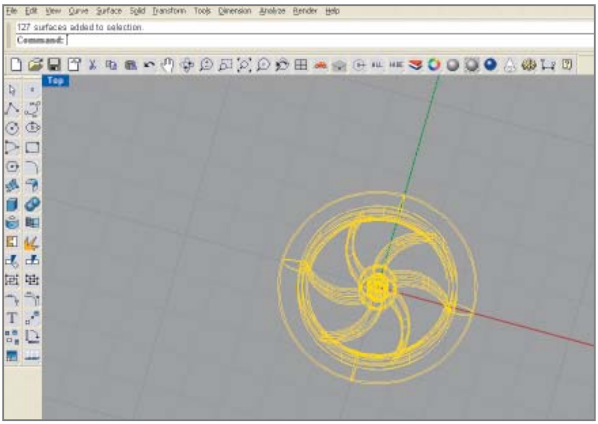


Figure 31. Selected object before making it watertight

2. From the *Edit* menu, select **Join**.

or –

Click the *Join* icon on the side toolbar.

This command reduces the number of surfaces and fits them together tightly. (The entities are *not* fused together into one unit.) The message in the command bar indicates this.

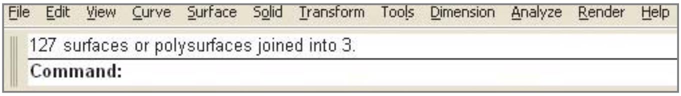


Figure 32. Command bar after joining surfaces

3. Select the object.

You can select the entire object, but to save time, you may select only the problematic entity.

4. From the *Tools* menu, select **Polygon Mesh > From NURBS Object**.

or –

Click the *Mesh from Surface/Polysurface* icon on the side toolbar. The *Polygon Mesh Options* dialog box opens.

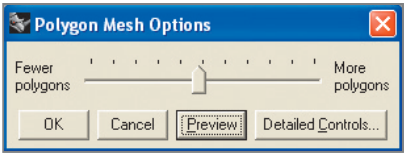


Figure 33. *Polygon Mesh Options* dialog box

5. Click **Detailed Controls**…

The *Polygon Mesh Detailed Options* dialog box opens (see Figure 28).

6. Enter the same settings as before (see step on page 18) and click **OK**.

7. Select the entire object.

8. From the *Tools* menu, select **Polygon Mesh > Weld**.

9. In the command bar, type `180` for the angle tolerance, and press Enter.

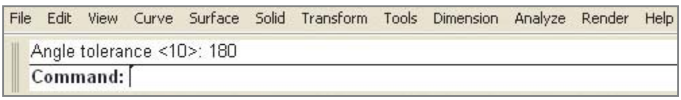


Figure 34. Angle tolerance entered in the command bar

With an angle tolerance of *180*, the *Weld* command **always** merges adjacent triangle points.

10. From the *Tools* menu, select **Polygon Mesh > Unify Normals**.

This setting unifies the normals of all triangles, so that they have the same definition for "up."

11. To validate that the object is watertight, type `SelNakedMeshEdgePt` in the command bar, and press Enter.

If the resulting object contains holes or gaps, the mesh point is highlighted in the display.

12. Repeat the *Save As* procedure (from step on page 18).

Phase 1: Saving a Model Design in STL Format

NX software from Siemens PLM (formerly USG), supports STL output at the core level, enabling you to save not only entire parts as STL files, but also selected surfaces of a part. This gives you great flexibility when preparing objects for 3-D printing. In addition, assembly output enables you to save several components as a single unit while maintaining each component as a *separate* volume (shell).

To save an NX part as an STL file:

1. From the *File* menu, select **Export > STL...**

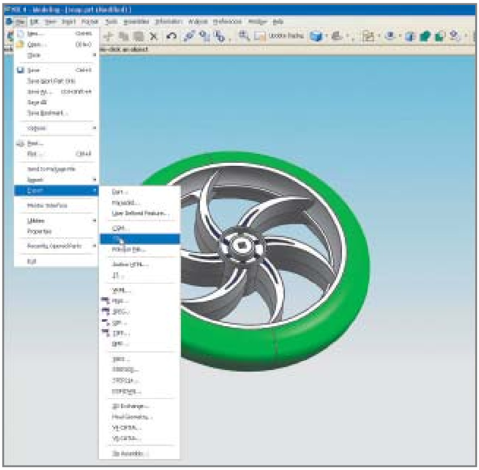


Figure 35. Exporting a design as an STL file

The *Rapid Prototyping* dialog box opens.

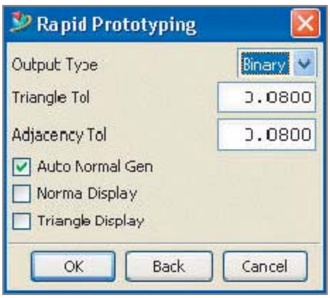


Figure 36. *Rapid Prototyping* dialog box

2. Set *Output Type* to **Binary**.

Binary STL files are much smaller than STL files saved in *ASCII* format.

3. Adjust the *Triangle Tol* setting to an appropriate value for your model.

This is the maximum distance allowed between the surface of the original design and the tessellated surface of the STL triangle, and affects the smoothness of the model surface.

4. Adjust the *Adjacency Tol* setting.

This determines if two adjacent surfaces "attach." If the distance between the two surfaces is less than this setting, they are considered attached. **This setting must be *less* than the printing resolution**. For example, when printing models at a resolution of 30 micrometers (microns), the setting must be no more than 0.03.

5. Click **OK**.

6. In the *Export Rapid Prototyping* dialog box, enter the file name and click **OK** (Figure 37).

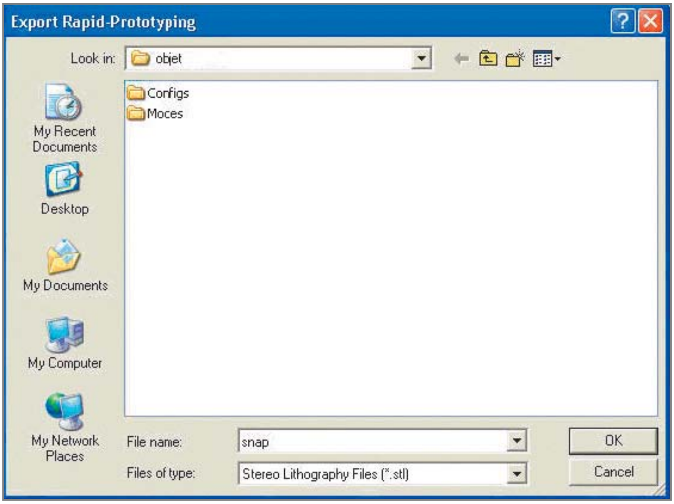


Figure 37. *Export Rapid-Prototyping* dialog box

For phases 2–3, see pages 7–8.

Phase 1: Saving a Model Design in STL Format

Solid Edge software from Siemens PLM (formerly USG) supports STL output at the core level, enabling you to save both parts and assemblies as STL files. Note, however, that when saving an assembly, all of its components are included in a single STL file.[1]

To save a Solid Edge part or assembly as an STL file:

1. From the *File* menu, click **Save As**.

The *Save As* dialog box opens.

2. From the *Save as type* drop-down menu, select **STL documents (*.stl)** and click **Options...**

The *STL Export Options* dialog box opens.

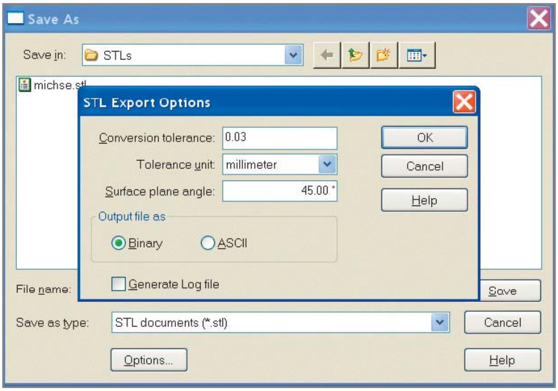


Figure 38. Saving a design as an STL file

3. Set *Conversation tolerance* and *Surface plane angle* to appropriate values for your model.

The lower the *Conversation tolerance*, the finer the tessellation.
The lower the *Surface plane angle*, the greater the accuracy (noticeable in small details).
As a rule, the finer the tessellation and the greater the accuracy, the larger the size of the STL file, and the longer it takes to generate it.

4. In the *Output file as* section, select **Binary**.

Binary STL files are much smaller than STL files saved in ASCII format.

5. Click **OK**.

6. In the *Save As* dialog box, click **Save**.

For phases 2–3, see pages 7–8.

[1] Solid Edge is technically capable of creating individual STL files from the components of an assembly, but this functionality is not built into the program. It is achieved through the application programming interface (API), using Visual Basic scripts. This solution does not enable a visual preview of the polygon mesh before saving the STL files.

Phase 1: Saving a Model Design in STL Format

Note: The *facetres* variable controls the resolution of the STL file, and can range from 0.01 to 10. (The default value is 0.5.) The higher the number, the smoother the model surface will be, but the file size will be larger.

AutoCAD Mechanical allows you to save model designs in STL format if they are single, threedimensional, solid objects.

To save an AutoCAD Mechanical solid object as an STL file:

1. In the command bar, type FACETRES, and press Enter.

2. In the command bar, type 7 for the new FACETRES value (see Figure 39), and press Enter.

```
Command: *Cancel*
Command: FACETRES
Enter new value for FACETRES <0.5000>: 7
```

Figure 39. New FACETERS value in the command bar

3. In the command bar, type STLOUT, and press Enter.

4. Select an object with the right mouse button, and press Enter to confirm that it is selected.

```
STLOUT Select a single solid for STL output: 1 found
Select a single solid for STL output:
```

Figure 40. Command-bar confirmation of selected solid object

The command bar prompts you to save the STL file in binary format (see Figure 41).

5. Press Enter to confirm that the STL file will be a binary file.

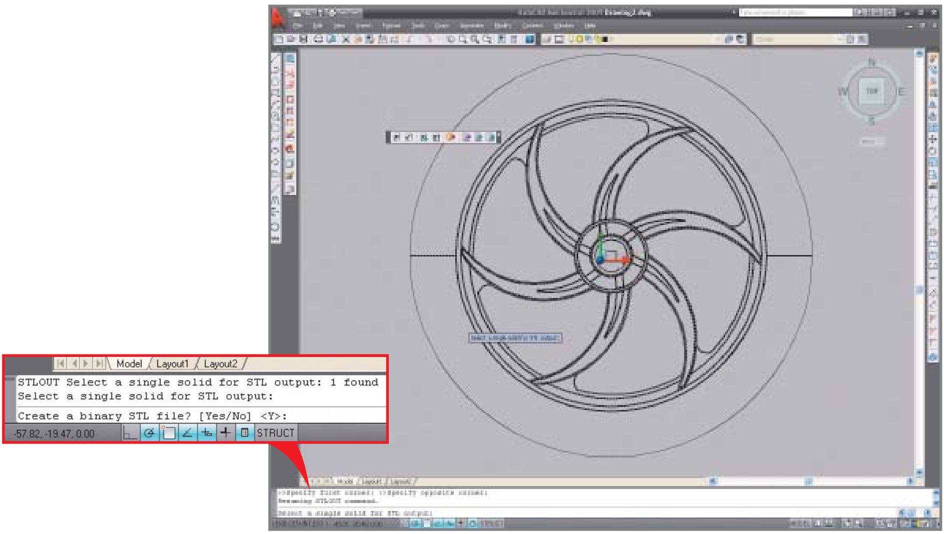


Figure 41. Binary file confirmation

The *Create STL File* dialog box opens.

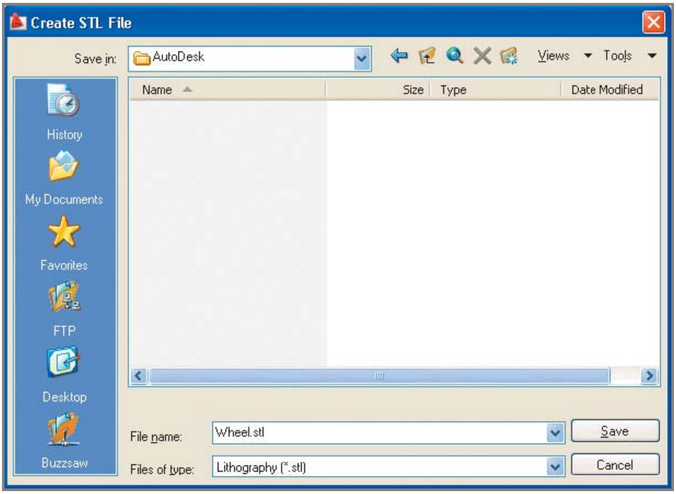


Figure 42. *Create STL file* STL dialog box.

6. Select a folder, enter a file name, and make sure that *Lithography [*.stl]* is selected in the "Files of type" field.

7. Click **Save**.

AutoCAD Mechanical saves the selected object as an STL file.

An STL file cannot be created if part (or all) of the selected solid geometry is in a negative XYZ plane. If this is the case, the following warning message is displayed in the command bar.

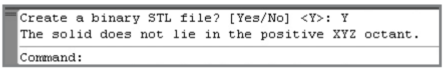


Figure 43. Negative space warning message

Translate the geometry to positive XYZ co-ordinates (for example, by using the *Move* command), and repeat steps 3–7.

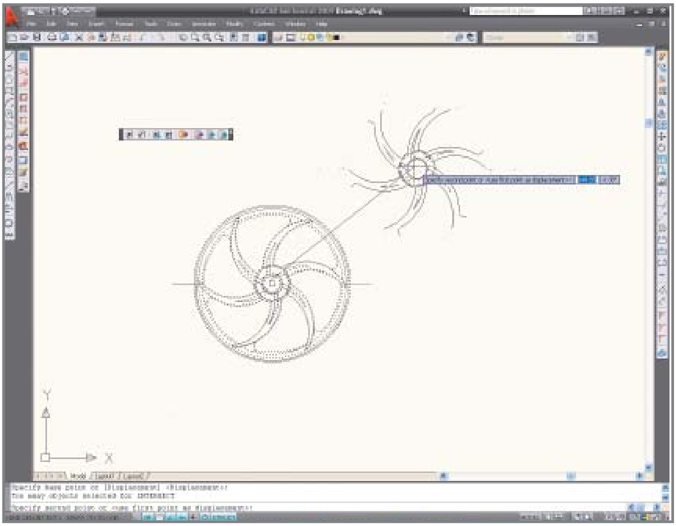


Figure 44. Translating to positive XYZ co-ordinates

For phases 2–3, see pages 7–8.

Phase 1: Saving a Model Design in STL Format

Autodesk Inventor allows you to save both individual parts and assemblies in STL format, at all design levels.

To save an Autodesk Inventor model design as an STL file:

1. From the *Tools* menu, select **Rebuild All**.

This ensures that the design data contains recent changes, and that it is not corrupt.

2. From the *File* menu, select **Save Copy As**…

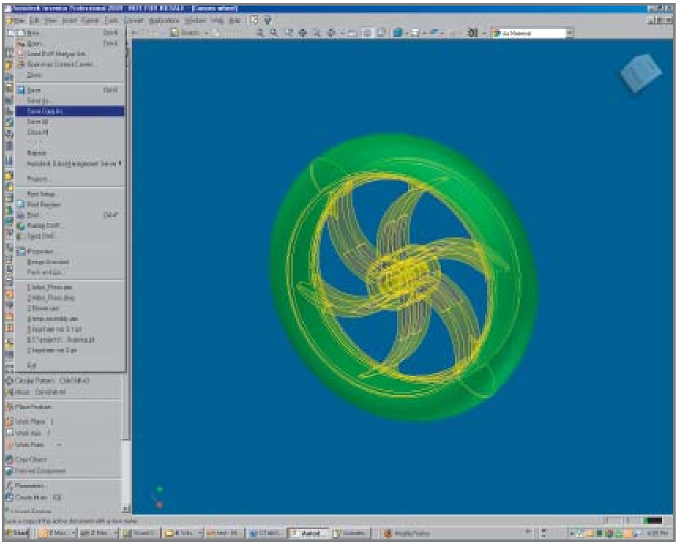


Figure 45. Saving a copy of the model design

The *Save Copy As* dialog box opens.

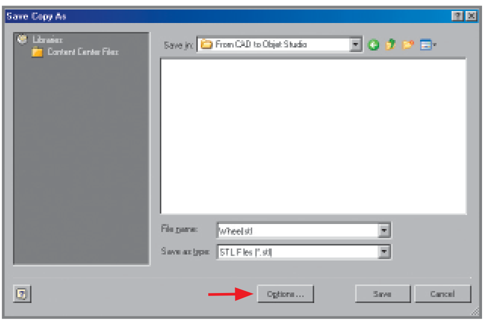


Figure 46. *Save Copy As* dialog box

3. In the *Save as type* field, select **STL Files (*.stl)**, and click **Options**.

The *STLOut Save Options* dialog box opens.

Figure 47. *STLOut Save Options* dialog box

4. Select *High* and click **OK**.

5. In the *Save Copy As* dialog box, click **Save**.

Note: To change the values associated with each of the resolution settings (*High/Medium/Low*) you need to edit the Windows registry.

For phases 2–3, see pages 7–8.

When designing models for printing with Objet 3-D printers, they must be saved with properties that ensure satisfactory results. Listed below are some common mistakes that can lead to unsatisfactory models, and tips for avoiding them

Saving the CAD Design as a Low-Resolution STL File

Because STL files describe a series of polygon meshes, they only approximate the high-level CAD design. The resolution used when creating the mesh determines how closely it matches the original. Reducing the size of the triangles (where applicable) increases the resolution of the polygon mesh.

Low resolution is noticeable on *non-planar* surfaces, which are faceted, rather than smooth – so much so, that you can see this on the display screen (see Figure 49). Printing an STL file saved at low resolution may result in a printed model whose surface is rough. If this is inadequate for your needs, increase the resolution before saving the STL file and printing the model.

Figure 48. CAD design

Figure 49. Design as low-resolution polygon mesh

A mesh resolution of 0.01 to 0.03 millimeters generally produces a good-quality STL file. Reducing mesh resolution below this range does not necessarily mean that model accuracy is improved. As a rule of thumb, designs that have many contours or curved surfaces need a higher resolution than flat, geometric surfaces.

Saving a CAD Design with Very Thin Features

For satisfactory results, it is recommended that the features of a design have a minimum thickness of 0.6 mm. (Although Objet printers are capable of printing thinner features, this requires that the printer is calibrated to factory specifications.)

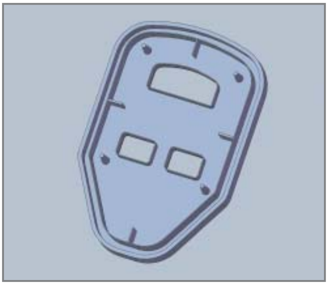


Figure 50. Example of a model design with thin features

Saving Non-continuous Designs

If a model design contains holes or gaps, it is not suitable for 3-D printing. You should make it "watertight" *before* saving it as an STL file. (It is much harder to fix STL files than native CAD files.)

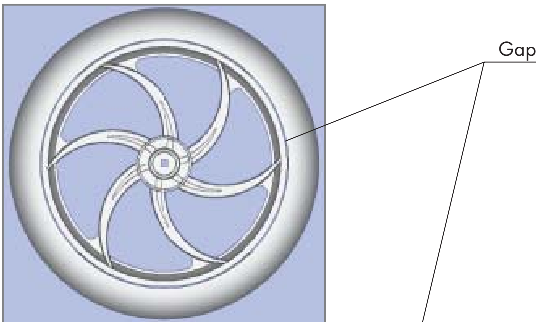


Figure 51. Design with gap

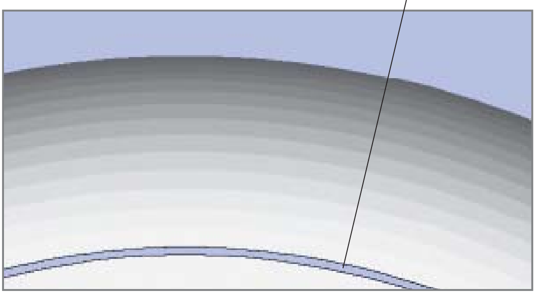


Figure 52. Design with gap (enlarged)

Saving Designs with Confined Hollows

If a model design contains confined hollows, there is no way to remove the support material. Design the model so that support material can be removed. This is especially important when printing moveable parts and parts made from clear material.

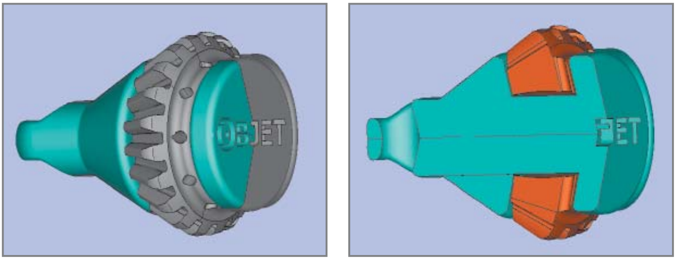


Figure 53. Model design with holes for removing support material

Saving Designs with Inadequate Clearances

To ensure the intended functionality of the design – for example, relative movement of parts, fine-detail separation, etc. – clearance between parts of a model should be between 0.25–0.3mm.

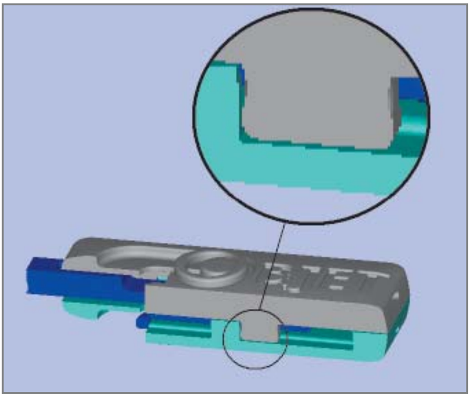


Figure 54. Clearance between moving parts

USG-04010

About Objet Geometries

Objet Geometries, the photopolymer jetting pioneer, develops, manufactures and globally markets ultra-thin-layer, high-resolution 3-dimensional printing solutions for rapid prototyping and rapid manufacturing.

The market-proven Eden™ line of systems is based on Objet's patented office-friendly PolyJet™ technology. Objet's FullCure® materials create accurate, clean, smooth and highly detailed 3-dimensional models, enabling even the most complex 3-D models to be printed with exceptionally high quality, accuracy and speed.

Connex500™, Objet's latest innovation, is based on Objet's PolyJet Matrix™ technology, which offers jetting multiple model materials simultaneously. PolyJet Matrix jets Digital Materials™ creating composite materials which are fabricated on the fly.

Objet's solutions enable manufactures and industrial designers to reduce cost of product development cycles and dramatically shorten time-to –market of new products. Objet systems are in use by world leaders in many industries, such as automotive, electronics, toy, consumer goods, and footwear industries in North America, Europe, Asia, Australia and Japan.

Founded in 1998, Objet serves its growing worldwide customer base through offices in USA, Europe and Hong Kong, and a global network of distribution partners. Objet owns more than 50 patents and patent pending inventions.

Objet Geometries Ltd.
Headquarters
2 Holtzman st.,
Science Park,
P.O. Box 2496,
Rehovot 76124, Israel
T: +972-8-931-4314
F: +972-8-931-4315

Objet Geometries Inc.
North America
5 Fortune Drive
Billerica, MA
01821
USA
T: +1-877-489-9449
F: +1-866-676-1533

Objet Geometries
GMBH
Im Leuschnerpark 4,
64347 Griesheim
Germany
T: +49-6155-605-346
F: +49-6155-605-344

Objet Geometries AP
Asia Pacific
13th floor, Unit 52A,
HITEC
1 Trademart Drive
Kowloon Bay, Hong Kong
T: +852-217-40111
F: +852-217-40555

Objet Geometries AP
Limited China Rep Office
Rm1220, CIMIC Tower,
1090 Century Blvd,
Pudong Shanghai
2000120 P. R. China
T: +86-21-5836-2468
F: +86-21-5836-2469

info@objet.com www.objet.com

OBJET