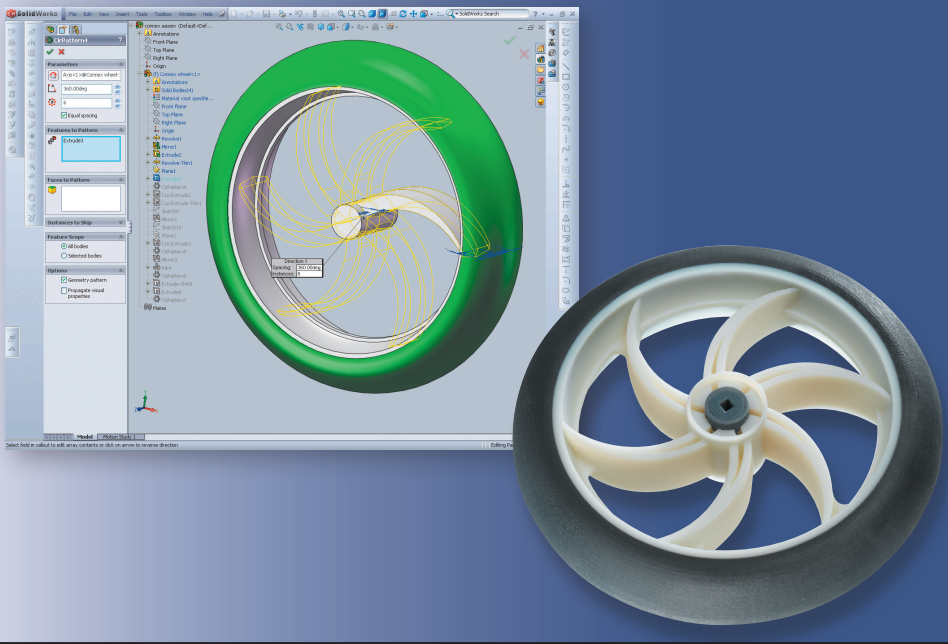

From CAD to Objet Studio Workflow for Connex Technology

Stratasys

DOC-13043 Rev. A

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Contents

CAD - Objet Studio Workflow

Understanding the interface between your CAD software and the Connex 3-D printer lets you get the most out of innovative Connex technology.

Overview

The basic workflow for printing parts from more than one model material (Mixed Part™) consists of five phases, as illustrated in the following figure.

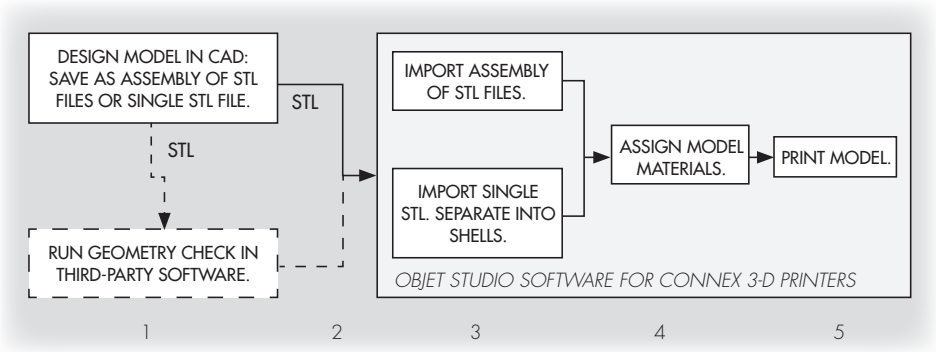


Figure 1: Phases of the CAD-Objet Studio workflow

The workflow begins with the design of an assembly of parts whose solid bodies are distinct objects.

This ensures that the model will be separable into its components (shells) when saved in STL format.

Models containing "holes" or "gaps" adversely affect quality when printed. Therefore, it is recommended that you perform a geometry check of the STL files before continuing. If there is a problem with a file, the software attempts to fix the geometry.

Objet Studio™ contains a full range of tools for printing Mixed Parts. You can print several individual models at once, using different model materials, and you can print the distinct components of one model, each with a different material. To make this possible, each component must be a separate STL file. This is achieved by either loading the model into Objet Studio as an assembly, or by separating the model into shells with Objet Studio software.

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

Smart Shell Design

It is recommended to save separate STLs (shells) for each color and then insert them as an assembly. The number of STLs can be the number of colors used, even if many parts use the same color. Color assignment is done in Objet Studio.

Opacity

Consider the surface thickness of parts or shells when preparing them for printing in different colors. This is especially important when printing a light color next to a dark one (for instance, yellow next to black).

To maintain an opaque color, the surface thickness of adjacent shells should be at least 2 mm.

Overlapping Shells

Avoid designing parts with overlapping shells. The color of overlapping parts cannot be properly controlled and the results are unpredictable. This is especially true when one of the parts is a light color and the other is a dark color.

It is recommended to use Boolean tools in your CAD software to eliminate overlapping areas.

Translucency

For translucent parts, in Objet Studio select a combination of clear material (VeroClear) and a color. Thinner parts are more translucent than thicker parts.

Phase 1: Designing Models in SolidWorks

During the model-design phase, you decide how to create the model, either as an assembly of parts, or as a multi-bodied model. Connex 3-D printers support both methods.

Model Assembly

With a model assembly, each part of the model is saved as a separate STL file. Taken together, the model appears as one unit. However, because each part is an independent element, special treatment can be applied to them in Objet Studio.

Multi-Bodied Model

With a multi-bodied model, each part of the model is designed as an individual, distinct component. Seen as a whole, the model is one unit. However, because of the multi-bodied design, the distinct parts can be isolated, in Objet Studio, for special treatment.

Creating Multi-Bodied Models

To create multi-bodied models, use the following commands:

- Extrude Boss; Cut (applies also to thin features)
- Revolve Boss; Cut (applies also to thin features)
- Sweep Boss; Cut (applies also to thin features)
- Surface Cut
- Boss; Cut thicken
- Cavity

Make sure that the *Merge result* check box is not selected.

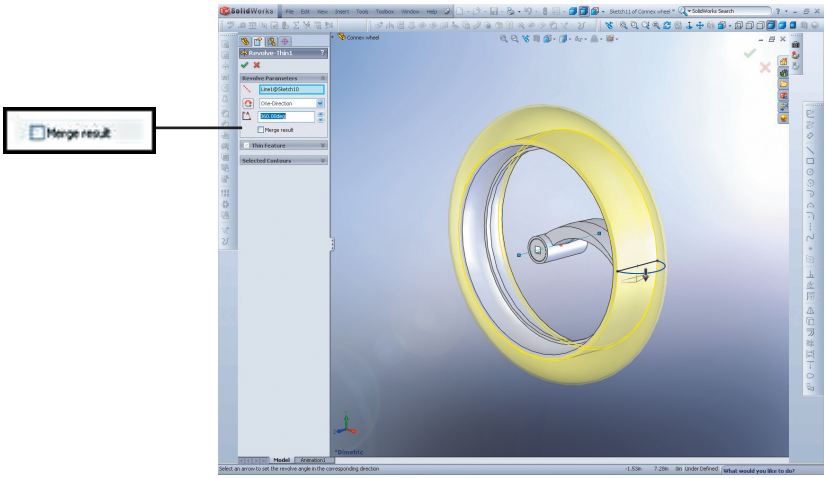


Figure 2: Creating a multi-bodied model in SolidWorks

Phase 2: Saving a Model Design in STL Format

SolidWorks supports the saving of 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 multi-bodied model or a model assembly in STL Format:

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

   The *Save As* dialog box opens.

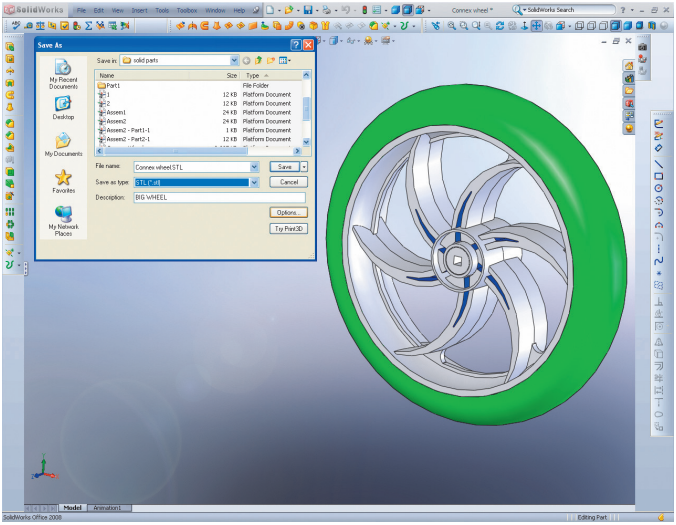


Figure 3: *Save As* dialog box in SolidWorks

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 selected as the *File Format* (see Figure 4).

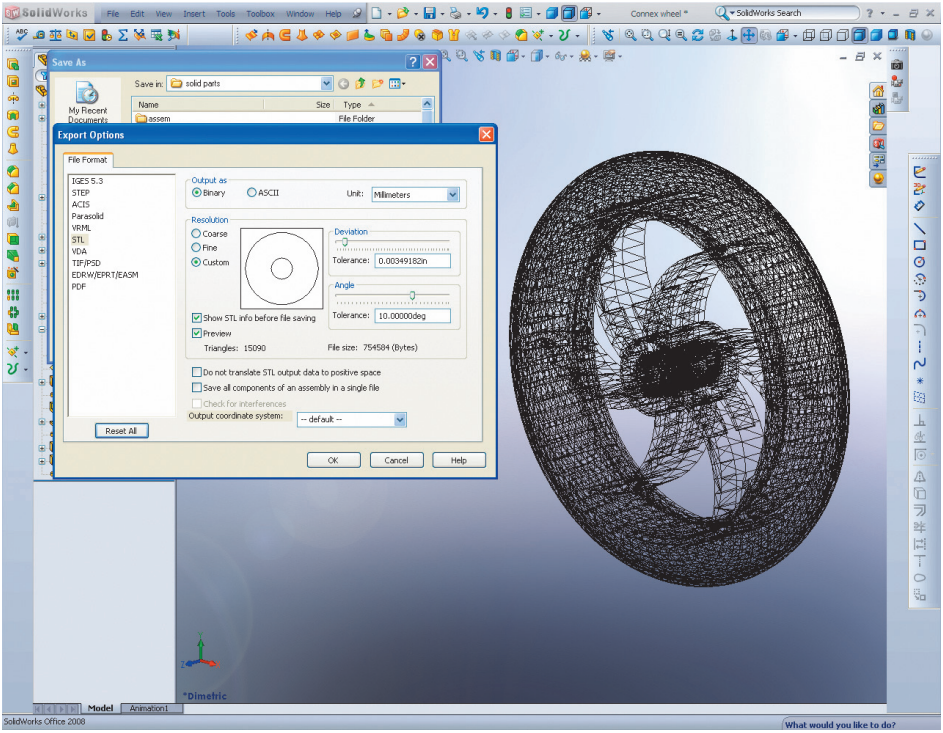


Figure 4: Export Options dialog box and SolidWorks screen

4. In the *Export Options* dialog box (see Figure 5), in the *Output as* section, select **Binary**. (The resulting file size will be smaller than if saved as "ASCII.")

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 it takes to generate.

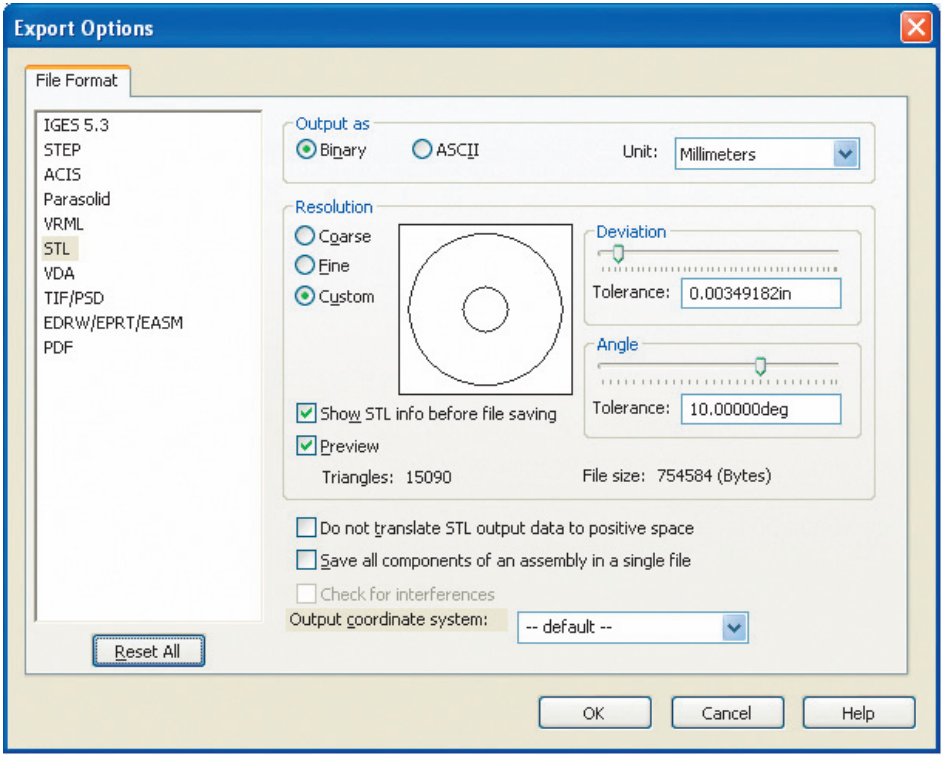


Figure 5: Export Options dialog box

6. Make sure that these check boxes are not selected:

- **Do not translate STL output data to positive space**
  This ensures that the parts maintain their original position in global space, relative to the origin.
- **Save all components of an assembly in a single file**
  This ensures that each component is saved as a separate STL file.
  Note: This check box is not relevant when saving multi-bodied models, since the model is saved as single STL file.

7. Click **OK**.
8. In the *Save As* dialog box, click **Save**.
9. In the confirmation message, click **Yes** (see figures 6 and 7).

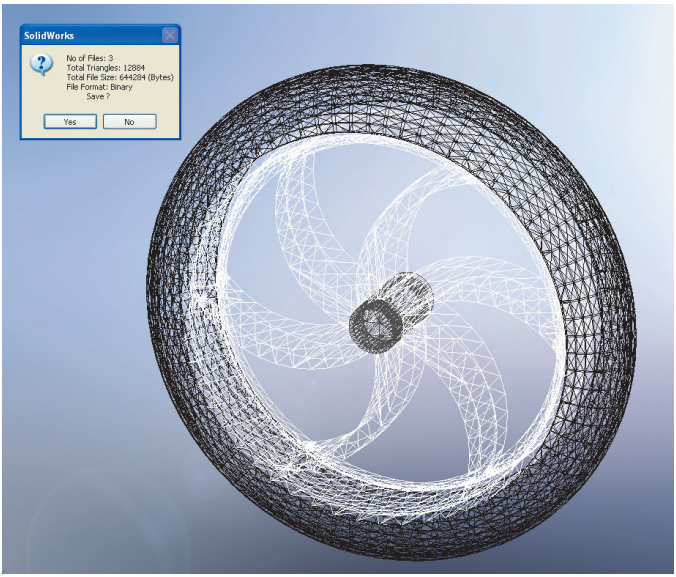


Figure 6: Save confirmation message—for saving a model assembly

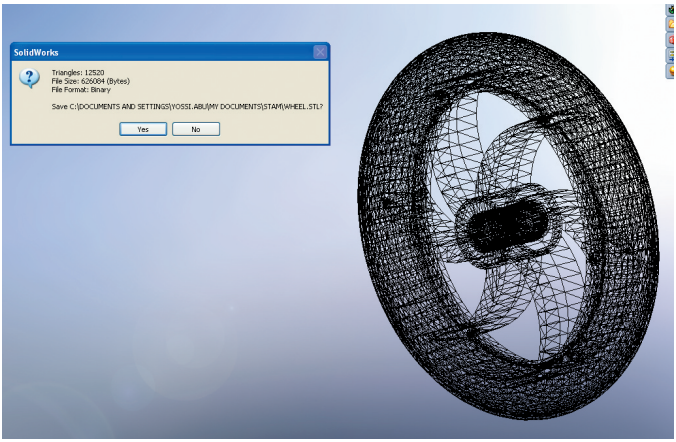


Figure 7: Save confirmation message—for saving a multi-bodied model

Phase 3: Opening Model Files in Objet Studio Software

There are two procedures for opening model files in Objet Studio, one for opening an assembly of STL files, the other for opening a single, multi-bodied STL file.

A. Placing a model assembly on the Objet Studio build tray

1. Open the Insert dialog box:

- Right-click in the build tray and select **Insert**.
- From the Object menu, select **Insert**.
- Click the *Insert Model* icon 

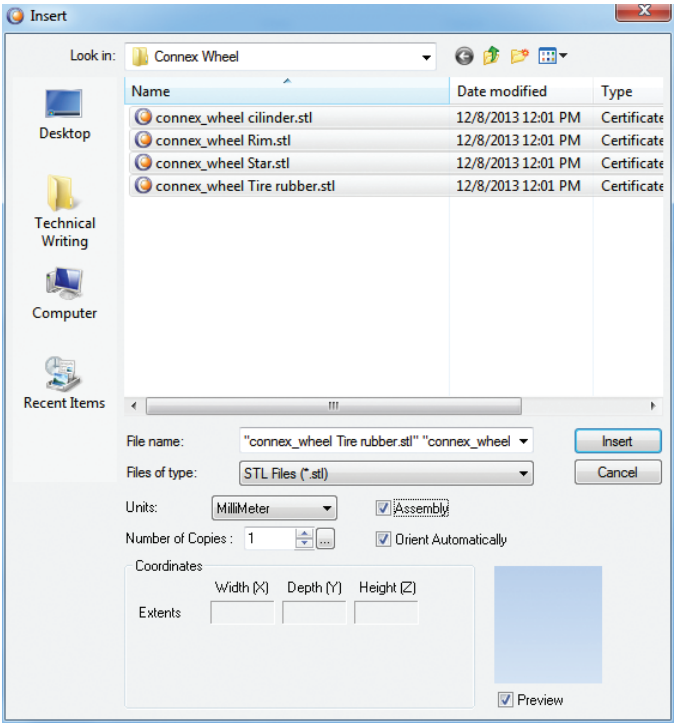


Figure 8: Selecting an assembly in the Insert dialog box

2. Select *all* of the assembly's component STL files.
3. Select the *Assembly* check box and click **Open**.
   The selected components are inserted as a complete, integrated model, and they are listed in the hierarchy pane, on the left (see Figure 9).

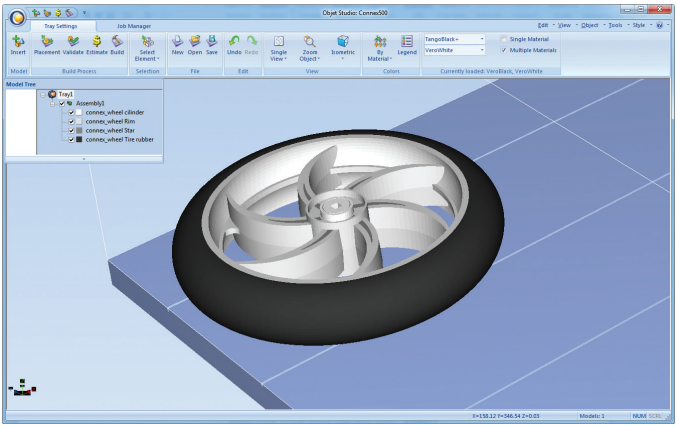


Figure 9: Assembly inserted in Objet Studio

B. Placing a multi-bodied model on the build tray and separating it into shells

1. Open the *Insert* dialog box:

- Right-click in the build tray and select **Insert**.
- From the *Object* menu, select **Insert**.
- Click the *Insert Model* icon  .

2. Select the STL file, and click **Open**.

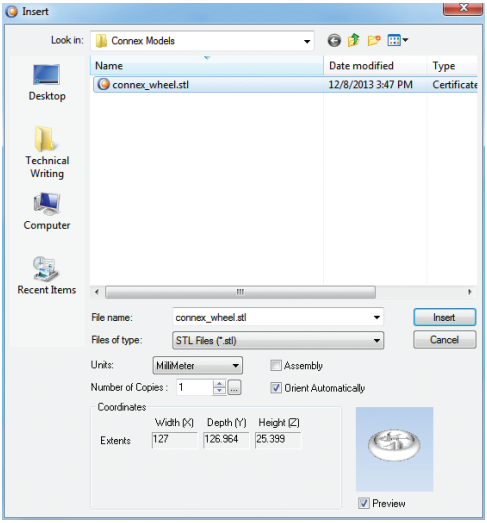


Figure 10: Selecting a single STL file in the Insert dialog box

The model is placed on the build tray and a single element appears in the hierarchy pane, on the left (see Figure 11).

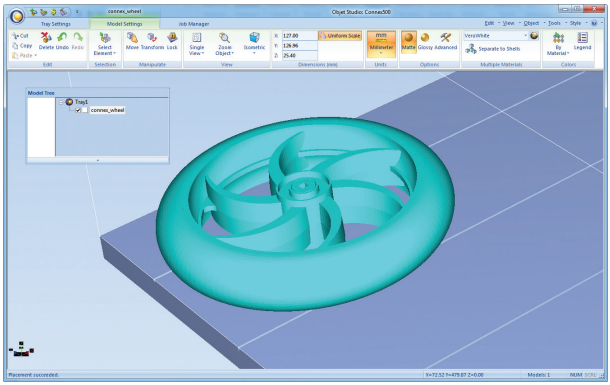


Figure 11: Single object inserted in Objet Studio

3. Select the object, and separate it into shells:

- From the *Object* menu, click **Separate into Shells**.
- In the *Model Settings* tab, *Multiple Materials* section, click **Separate to Shells.**

A dialog box opens in which you can specify where to save the component STL files (shells).

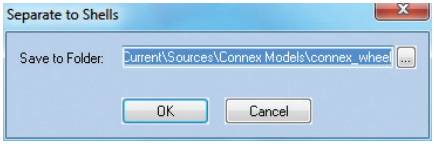


Figure 12: Folder selection

4. Click **OK**.

Objet Studio splits the object into an assembly of component parts (shells), each a separate STL file. The object now appears on the build tray, and in the hierarchy pane, as an assembly of shells (see Figure 13).

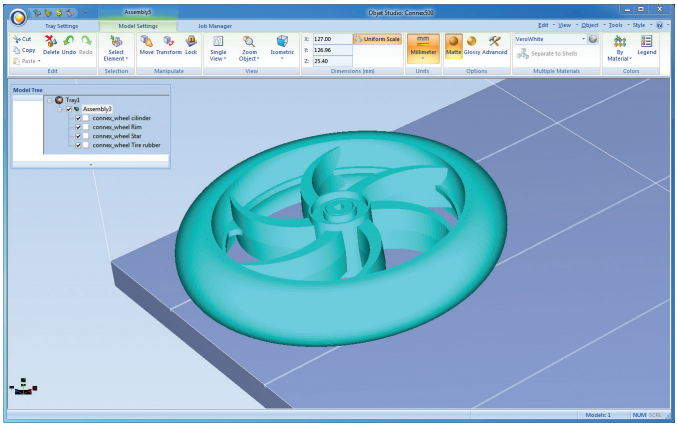
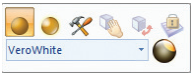


Figure 13: Object after being converted to an assembly

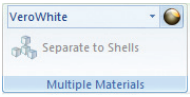
Phase 4: Assigning Model Materials to Shells

When different model materials appear in the *Tray Settings* ribbon, you can print objects (shells) with either material or with a combination of them – Digital Materials.

1. Select an object on the build tray.
2. Select the desired model material from: the pop-up toolbar.



- the *Multiple Materials* section on the *Materials Settings* ribbon



On color printers, you can assign materials to models using a color palette if certain combinations of materials are available.

Other color combinations can be used with duotone palettes (two-color digital materials).

If more colors are needed, consider printing separate parts to assemble as a post-printing process.

Notes:

- Colors selected in Objet Studio might not accurately reflect the printed color due to shading in 3D views and different screen calibrations.
- In Objet Studio, transparent colors appear opaque.

Printing Transparent Parts

Transparency

UV light causes a yellow tint on transparent parts. For better results, limit exposure of transparent parts to UV light. For this reason, the surface finish you choose affects the clarity of parts printed with VeroClear.

Matte Surface Finish – It is preferable to print transparent parts with a matte surface finish. The extra support used to cover matte surfaces provides protection from UV light and therefore improves clarity.

Glossy Surface Finish – Arrange parts with a glossy surface finish so that they have similar heights. This reduces the number of passes the UV light needs to make over the models during printing.

There are various post-processing techniques you can use to improve a part's transparency. For example, polishing parts printed with VeroClear improves clarity. Applying a coat of clear lacquer gives parts a shine and protects their surfaces. For details and more suggestions, refer to the "Guide to Post Printing Applications"

Phase 5: Printing the Model

When the models on the build tray are ready to be printed, place them in the print queue:

- From the *Tray Settings* ribbon, in the *Build Process* section, click Build

Notes:

- Color printing is always performed in DM mode.
- Gloss and matte surfaces might have slightly different hues.
- Pausing a print job for more than 30 minutes might cause a color variation in the Z direction. To avoid unnecessary pauses, before printing sure that the material cartridges are sufficiently full and that the waste container is empty.

The Connex 3D printer builds the model, using the materials assigned to the model's components.

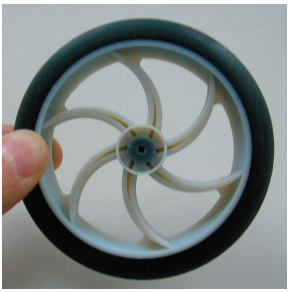


Figure 16: Finished model

Phase 1: Designing a Model in Pro/ENGINEER

Since Pro/ENGINEER does not enable designing multiple-bodied models (as is possible with SolidWorks), non-continuous models must be handled as Pro/E assemblies, not as parts. This does not present a problem when beginning the design in Pro/E, as will be explained below.

However, when importing files, especially those saved in a neutral format (STEP or IGES, for example), multiple-bodied models must be imported as assemblies to take advantage of the multiple model-material features in Connex 3-D printer. For this reason, it is recommended to import all neutral files as assemblies.

Phase 2: Saving a Model Design in STL Format

Pro/E allows you to save both parts and assemblies in the STL file format. Handling the model design as an assembly enables you to print the model's components with different model materials on Connex 3-D printers. Models designed and exported as parts can only be printed with a single model material. However, using the Mixed Tray™ feature, you can print several parts, each with a different model material, at the same time.

This section describes saving both parts and assemblies in STL format, for printing with Connex 3-D printers.

Saving a Pro/E Part in STL Format

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 17).

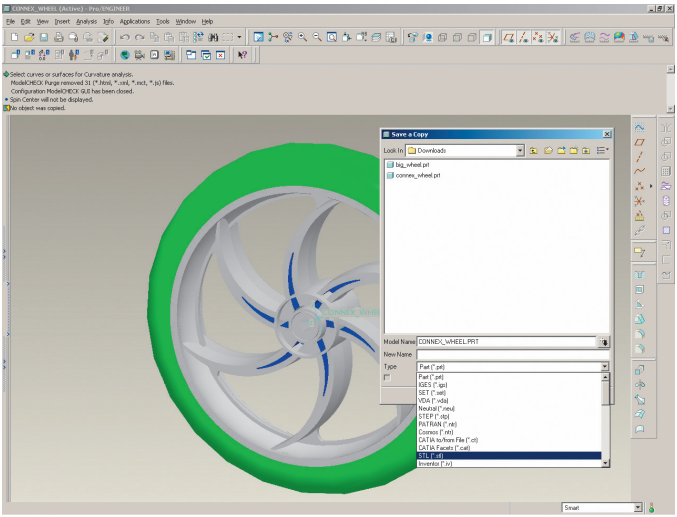


Figure 17: Saving a part as an STL file

The *Export STL* dialog box appears.

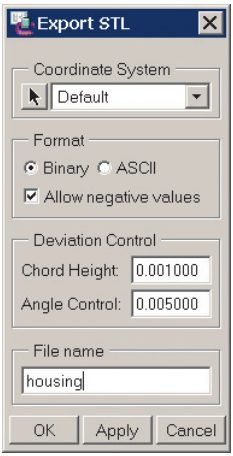


Figure 18: *Export STL* dialog box

Deviation Control

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

- **Chord Height** – specifies the maximum distance between a chord and the part's surface (see Figure 19). This 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.

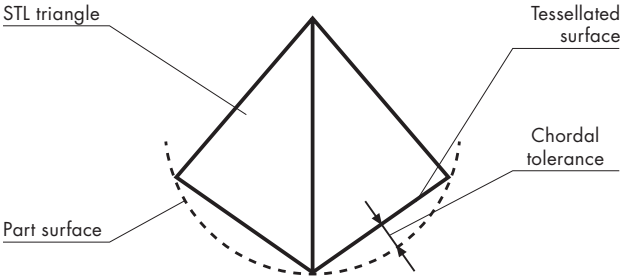


Figure 19: Chord Height

- **Angle Control** – 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.

Saving a Pro/E Assembly in STL Format

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 20, 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 model resulting from these settings (when you click **OK**) is shown on the left.

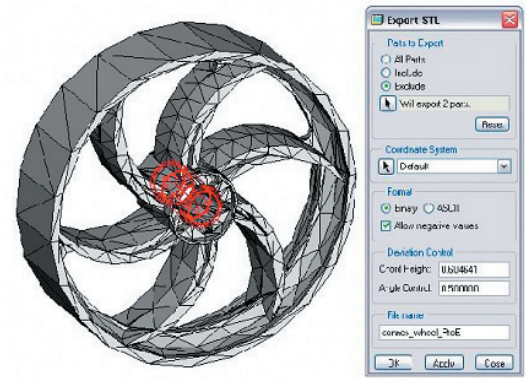


Figure 20: Export STL settings for assemblies and the resulting model saved

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

For phases 3–5, see pages 7–10.

Phase 1: Designing and Handling Models in Catia

With a special add-on module, you can save designs in Catia as STL files. Catia V5 is capable of exporting 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 exported as parts. If the source design was saved as an assembly, it is imported as a product. To export it as an STL file, you must convert it to a multi-bodied part. The procedure described below is one of several methods for doing this. Since Catia 5 supports non-continuous model designs, importing geometry into a part by copying and pasting is not problematic.

Importing a model design in Catia for export as an STL file

Catia can import almost any design-file format, but only those that include solid data (IGES, STEP, Parasolid, etc.) can be exported from Catia as STL files.

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 21).

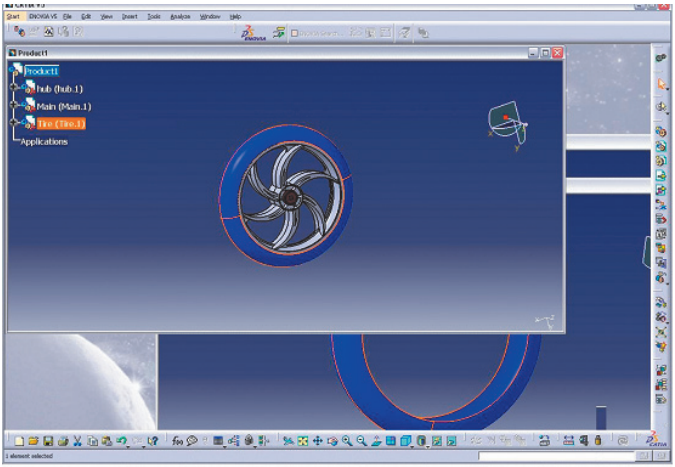


Figure 21: An assembly imported into Catia

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 21), 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 22 shows the result of pasting the components of the model from Figure 21 as individual parts.

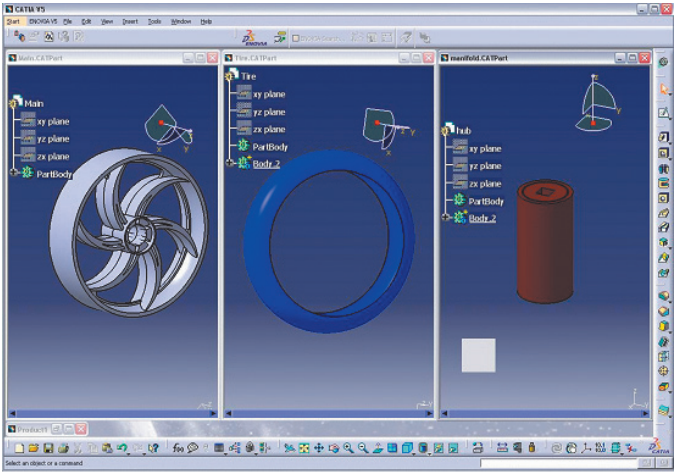


Figure 22: Assembly components saved as individual parts

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 22) and paste them into the new (combined) model file (Figure 23).

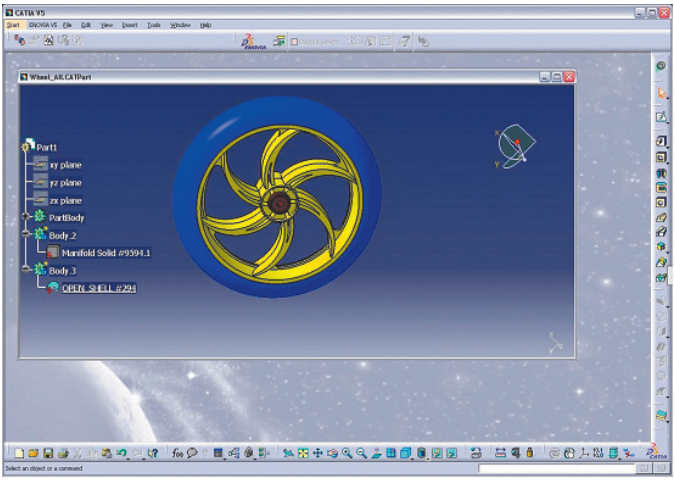


Figure 23: Multi-bodied part assembled from individual parts

Since the geometries of all of the parts are retained, they should be aligned correctly in the combined part. The new, multi-bodied 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 24).

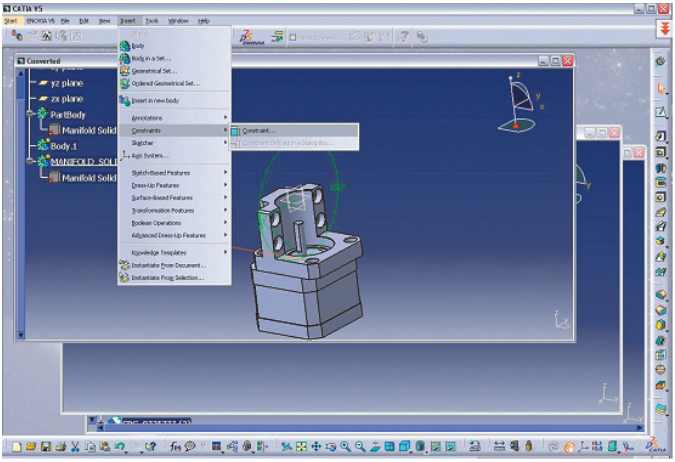


Figure 24: Re-aligning parts

Phase 2: Saving a Model Design in STL Format

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 25).

- **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).

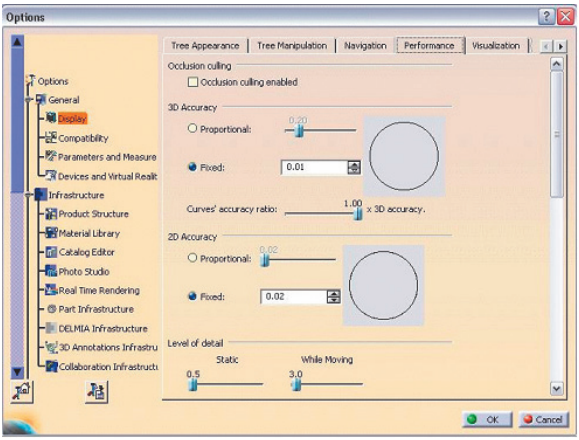


Figure 25: Setting the 3D Accuracy settings in the Options dialog box

When the Catia part is ready to be saved as an STL file, 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.

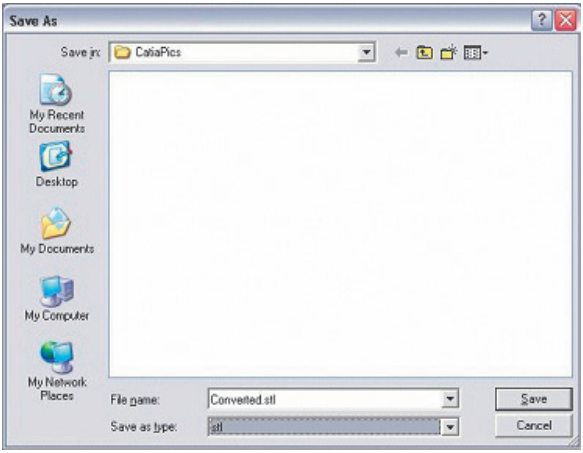


Figure 26: Saving the Catia files as an STL file

3. Click **Save**.

Please note:

- Hidden contour lines are not exported in the STL file.
- The Catia part does not need to be continuous.

For phases 3–5, see pages 8–12.

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Stratasys FOR A 3D WORLD™ | www.stratasys.com | info@stratasys.com

7665 Commerce Way
Eden Prairie, MN 55344
+1 888 480-3548 (US Toll Free)
+1 952 937-3000 (Int'l)
+1 952 937-0070 (Fax)

2 Holtzman St.,
Science Park, PO Box 2496
Rehovot 76124, Israel
+972 74 745-4000
+972 74 745-5000 (Fax)

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