

Export/3D Printing

CX 7.5 - 3D printing - Can I use Design 3D to model an object for 3D printing?

Design 3D and 3D Printing

3D printing - also called additive manufacturing - is the process of making a physical object from a digital model. 3D printing is achieved using an additive process, where successive layers of material are laid down in different shapes.

3D printing is used for both prototyping and manufacturing in jewelry, art, industrial design, architecture, engineering and construction, automotive, aerospace, dental and medical industries, and many other fields. Materials available for 3D printing include many types different types of plastic; also ceramic, glass and metals.

Different printers, Different Requirements

The file format and building techniques will depend on the model, the printer or printing service and the material the model is intended for. There are three primary types of additive 3D printers, Fused Deposition Modeling (FDM), Polyjet or Jetted Photopolymer (JP), and Stereolithography (SL or SLA). There are many different 3D printers and 3D printing services, and each has its own requirements. It's important to find out these requirements; preferably before you begin modeling.

The resolution of a printed 3D model can vary widely, depending on the 3D printing technique. It's important to find out exactly how much detail to include, before modeling and printing. You will not get good results printing a highly detailed model with a technique that will not support that detail. To create functional pieces you need to know what technology will be used for printing, and request the proper material(s). Understanding the manufacturing requirements will help you create an appropriate model.

3D Printing Services

There are a number of companies that specialize in 3D printing, such as Shapeways, Sculpteo, iMaterialize, etc. Some have associated online stores where you can market and sell your designs. Some offer model checking and repair services, but some do not and will charge for printing mistakes caused by faulty files.

File Formats

Common file types accepted by 3D printers and print services include DAE, OBJ, STL, X3D, X3DB, X3DV, WRL. Strata Design 3D CX supports exporting to Collada (DAE), OBJ and WRL (VRML), and if the object(s) are created properly these formats can work with many 3D printers and services.

STL Format

STL files are preferred by some 3D printers and print services. **You can now import and export STL files in Design 3D CX 7.5.** STL files export with Strata model units. When you import an STL file into Design 3D CX 7.5, you can choose the model units in a dialog.

Export/3D Printing

If you have a **pre-7.5** version of Strata, you can still get a Strata model to an STL file but not directly. You will need to take the file through a third party program which can read one of Strata's export formats and in turn write the STL file for you.

-The most common method is to export a Collada or OBJ file from Strata 3D, then open that in a program like Silo or ViaCAD (both paid programs) or Wings 3D (free) and export to an .stl file. You can also use Netfabb to convert files from OBJ to STL. NetFabb is an STL file viewer and repair tool. The basic version of Netfabb is free.

-When converting a Strata file to an .stl file, you need to understand the scaling process. There is a helpful discussion on the StrataCafe about this:

http://www.stratacafe.com/forum/topic.asp?TOPIC_ID=13913&whichpage=1&SearchTerms=stl

-There is a lot of good information about 3D printing and scale conversion on the StrataCafe Forums. Simply search the Strata Design 3D CX forum, and include the archives. The search page is located here: <http://www.stratacafe.com/forum/search.asp>

File Repair

There are companies and software for checking and repairing a 3D model in preparation for 3D printing. We've heard good things about Netfabb, an STL file viewer and repair tool. The basic version of Netfabb is free. Many 3D printing services have their own repair software and/or services.

General Guidelines for Modeling in Design 3D

Here are some general guidelines for modeling in Design 3D for output to 3D printing. Remember, it's critical to contact your printing service for specific requirements.

1. Objects must be closed, and structurally sound. Your geometry must be watertight - no openings in the geometry. It can sometimes be difficult to identify openings in your model's geometry. Some printing services can have tools to help you identify these problems. Your model may need support material during the printing process to keep it from collapsing. Support material is generally break away or soluble for easy removal after the printing process.

2. Objects must be manifold - no shared edges. A mesh is non-manifold if it has edges that are shared between more than two faces. In Design 3D, you can identify non-manifold edges by choosing Select Non-manifold from the Context menu while in Polygon Edit mode.

If you find there are open edges in your object, you will need to correct them. Make sure the surface of the mesh is solid by filling in areas where edges connect to only one face. If there are more than two edges, choose a face to delete or Unweld the edges. Then Weld only two edges back together.

3. Pay careful attention to the maximum size, scale and wall thickness. Incorrectly constructed or scaled models can cause delays and extra costs.

The optimum size of your object and the minimum wall thickness depend on the 3D printer, the

Export/3D Printing

production method and the material that you are planning to use. Wall thickness is important - it needs to be thick enough to support the the object once printed, but minimized to reduce cost. Thickness and size must conform to the 3D printer's requirements.

Design 3D's basic unit of measure is the point - so you will need to work out the scale factor between Design 3D and the 3D printer. There is a very informative discussion about this on the StrataCafe.com: http://www.stratacafe.com/forum/topic.asp?TOPIC_ID=13913

4. Correct normals. All surfaces of your model should have their normals pointing in the correct direction. Inverted normals can cause problems with 3D printing (as well as rendering anomalies). Inverted surface normals can occur when importing files from other applications, during polygon editing, etc.

To find and correct an inverted normal in Design 3D, set the object to One-sided in the Object Properties palette. Next, turn on Backface Culling from the Scripting menu or Preferences dialog. Any inverted normals will be invisible - you will be able to see through them in the modeling window.

To correct an inverted normal, go to Polygon Edit mode. Select the inverted normals, and then use the Flip command from the Context menu.

Unique solution ID: #1457

Author: Julie Applegate

Last update: 2013-11-04 16:40