Exporting 3D files: STL vs. OBJ vs. IGES vs. STEP - Jaycon Systems - Medium

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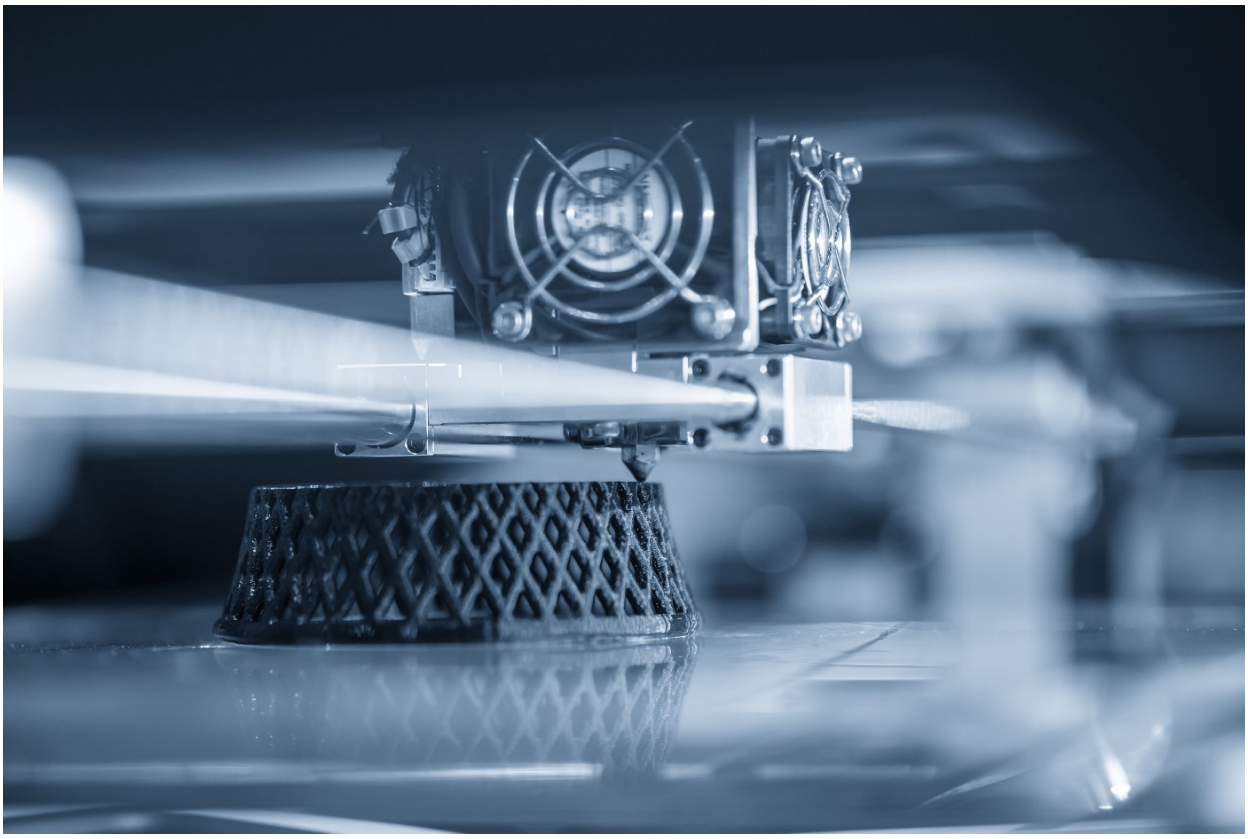
Regardless of your CAD preference, whether it be SolidWorks, Creo, Catia, or Siemens NX, your files will need to be exported sooner or later. But the question is, in what format? This article will dive into the most common file types used for exporting 3D models, when you should use each of them, and why. These file types include STL, OBJ, IGES, and STEP.

STL (Stereolithography)

The STL filetype has been around since 1987 but has exploded in popularity in recent years, as 3D printers have become more commonly used. It is the go-to filetype when it comes to 3D printing. STL files store objects as sets of vertices joined by edges to make triangular faces, which is where the misconception of STL standing for *Standard Triangle Language* or *Standard Tessellation Language* comes from. These triangles result in the inaccurate representation of previously curved surfaces as tessellated surfaces. Generally speaking, these tessellations do not cause a problem for 3D printing, as long as the mesh and the intended model have discrepancies smaller than the resolution of the 3D printer.



3D Printer Printing STL File

OBJ

The OBJ file format was developed by Wavefront Technologies, specifically for animations, but has since been adopted by the 3D community. Similar to an STL file, OBJ files are stored as a list of vertices joined by edges. However, unlike STLs, they support polygonal faces. Polygons allow for a closer representation of the original geometry than STLs do but still do not accurately portray the original geometry. With that being said, OBJ files can represent an object more accurately while using the same number of faces as an STL file.

IGES (Initial Graphics Exchange Specification)

The standards for IGES were first published in 1980 by the National Bureau of Standards. IGES differs from both STL and OBJ because it allows for the encoding of curves/curved surfaces instead of approximations using smaller, flat surfaces. The ability to encode curved surfaces allows IGES to maintain high accuracy while exporting the model. Although the export geometry is highly accurate, it is not perfect. Features such as splines will be converted to polylines during this process.IGES files also have the ability to store units within the file header.

STEP (Standard for Exchange of Product Data)

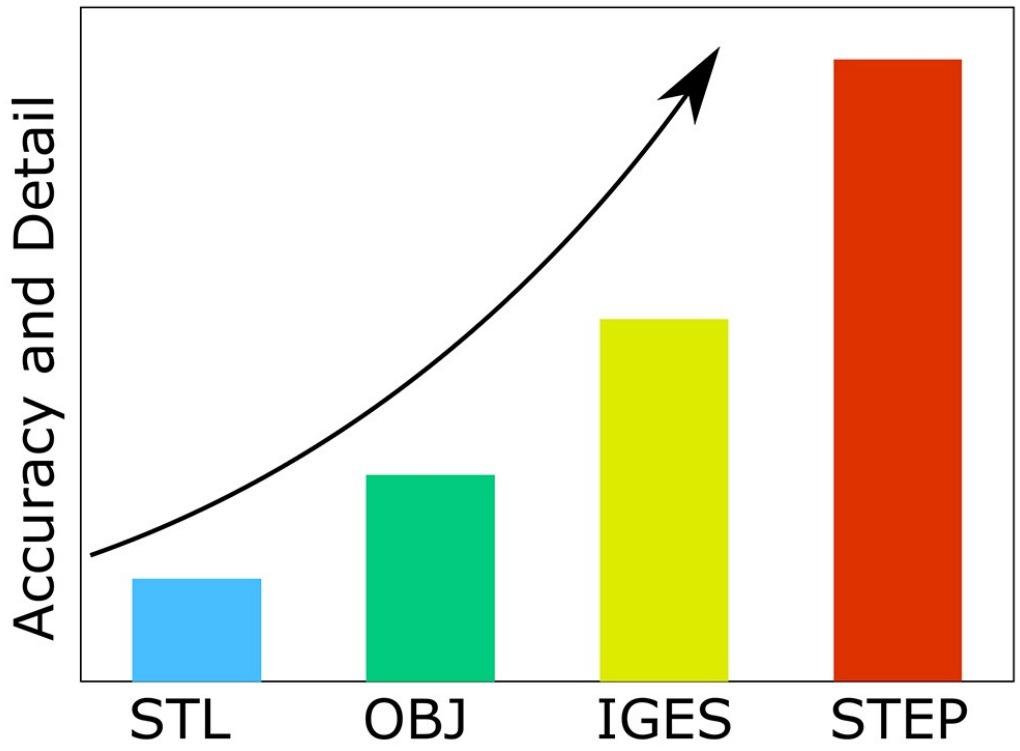
In 1994, the International Standards organization developed the STEP file format to facilitate the move to Model-Based Definition. Like IGES, STEP allows for the more accurate storage of models by encoding curves and curved surfaces; however, it still suffers from small losses while encoding splines.

Unlike the other file types here, STEP files allow for incorporating engineering data such as materials, geometric dimensions & tolerancing, and model intent within the model itself.

The ability to store additional data is attractive because it can eliminate the need to accompany 2D drawings and allow for annotations to be embedded within the model. STEP files are becoming more and more popular and are likely to dissolve the need for IGES.

Conclusion

The overall difference between these files comes down to the accuracy and detail at which they can store 3D objects.



All in all, the differences in how each file type records the geometries and details of an object can mean that each file type is better suited to a particular application. The approximation provided by STL and OBJ files is suitable for use in 3D printing and graphics. In contrast, the higher precision and more accurate features supported by IGES and STEP files are much better suited to CNC milling and CNC lathe applications.

Filetype	Overview	Level of Detail	Application
STL	STL files store objects as sets of vertices joined by edges to make triangular faces.	Medium-Low	3D Printing
OBJ	OBJ files store 3D information as a list of vertices joined by edges, and unlike STLs, they support polygonal faces that allow for a closer representation of the original geometry.	Medium	3D Printing (it includes object color and texture information)
IGES	IGES is similar to OBJ, however, it is able to encode curves and curved surfaces rather than an approximation using smaller, flat surfaces.	Medium-High	CNC
STEP	STEP is similar to IGES, but it includes engineering data such as materials, geometric dimensions & tolerancing, and model intent within the model itself.	High	CNC & Injection Molding

Lastly, it is important to consider that once an object is exported to a file type containing less accurate information about the model, it can become challenging to recreate the original geometry from that object.