



CAD Formats

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Formats of files

A **file format** is a [standard](#) way that information is encoded for storage in a [computer file](#). It specifies how [bits](#) are used to encode information in a digital storage medium. File formats may be either [proprietary](#) or [free](#) and may be either unpublished or open.

CAD file is a digital **file format** of an object generated and used by **CAD** software. A **CAD file** contains a technical **drawing**, blueprint, schematic, or 3D rendering of an object.

There may be other **CAD** tools that can be used to create, open, edit and export these .



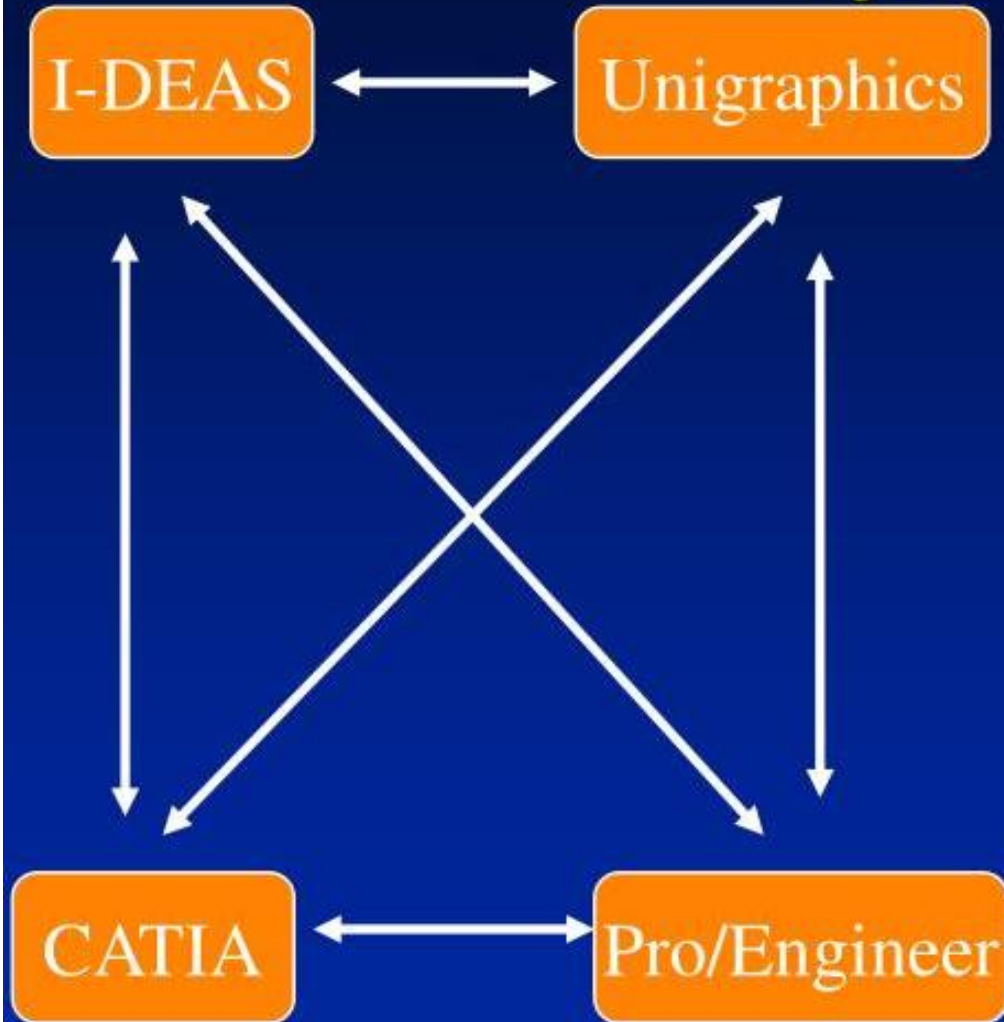
Why do we need Data Exchange?

Design projects require data to be shared between suppliers

- Different companies often used different CAD systems
- All CAD systems have their own database formats that are mostly proprietary and often confidential
- Data is stored in different ways e.g. 1.0,2.0,3.0 or X1.0,Y2.0,Z3.0, etc.
- Data conversion between systems becomes necessary



Direct Translation between CAD Systems



- Need a translator from every CAD package to every other
- For 4 CAD packages, need 6 translators
- For 6 CAD packages, need 16 translators!



Data Exchange Standards

- To address the problem, many standards for CAD data exchange have been developed
- CAD systems can import and export to many of these standard formats

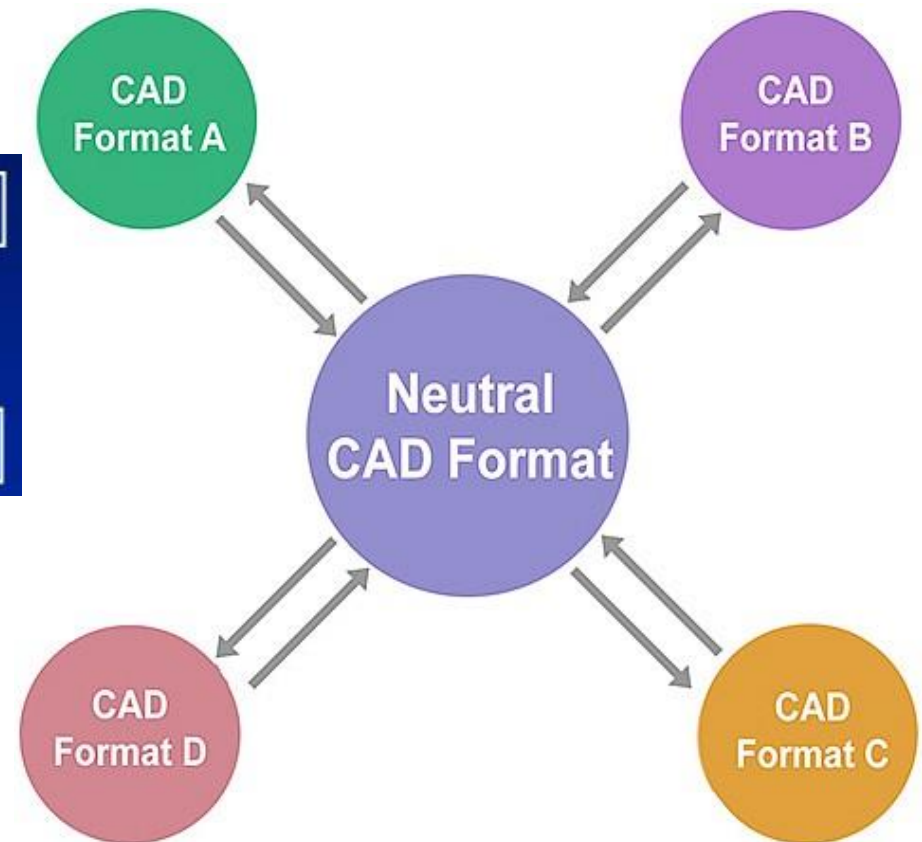
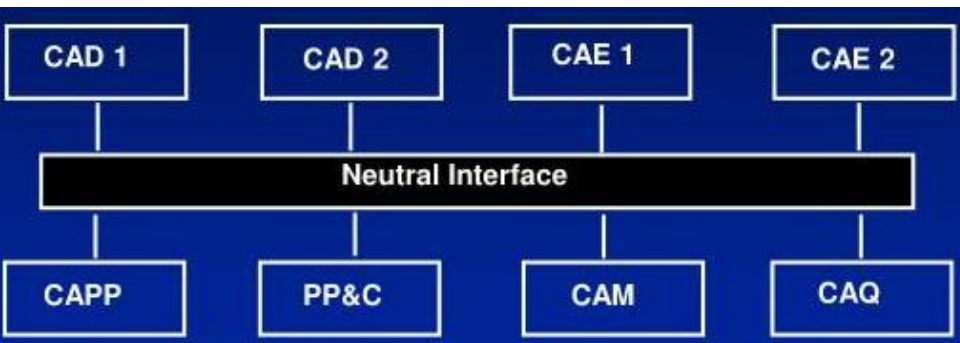


Evolution of Standards





Neutral file format





STEP

STEP stands for Standard for the Exchange of Product Data. This ISO standard exchange format is used to distribute 3D data **files**.

It is primarily used to transfer real-world mechanical and architecture designs between different CAD software tools. Common CAD software tools that support STEP files include AutoCAD, Fusion, SolidWorks, OnShape, PTC Creo, CATIA and ArchiCAD.



IGES

The **Initial Graphics Exchange Specification**

(IGES) (pronounced *eye-jess*) is a vendor-neutral [file format](#) that allows the [digital exchange](#) of information among [computer-aided design](#) (CAD) systems.

IGES file is composed of 80-character [ASCII](#) records.



Parasolid

Parasolid file format is native for **Parasolid** geometrical kernel. Today it is widely used in a range of applications, including SolidWorks, SolidEdge, NX, Microstation, Ansys, Abaqus and others. **Parasolid** has two primary **file** extensions: . x_t. Another format is . x_b, which is in binary format so it is more machine independent and not subject to **binary-to-text** conversion errors



DXF

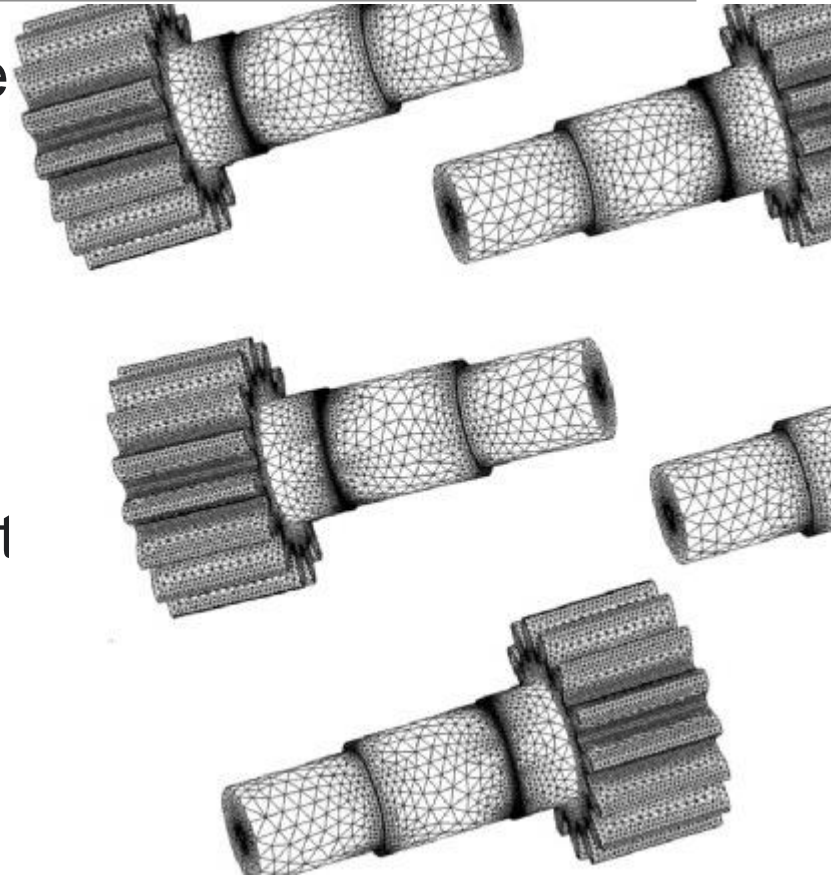
DXF is an exchange **format** for content of AutoCAD Drawing **Files** (DWG). A DXF_ASCII **file** is an ASCII text **file**; a binary version of **DXF** has also been defined, but is less widely used and not described separately in this resource. The **DXF format** specification is maintained and has been openly published by AutoDesk



STL (Standard Tessellation Language)

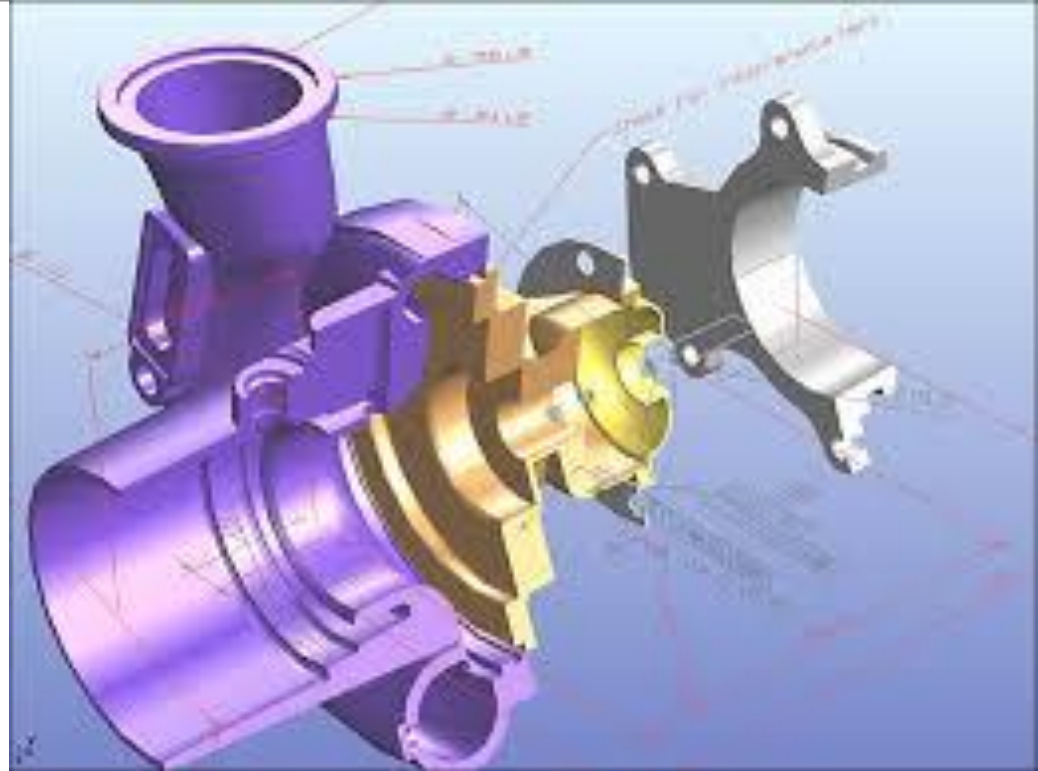
STL is a **file format** native to the stereolithography CAD software created by 3D Systems. ...

This **file format** is supported by many other software packages; it is widely used for rapid prototyping, 3D printing and computer-aided manufacturing



JT File format

The Jupiter Tessellation (**JT**) is a 3D data **format** which corresponds to an ISO standard. It is used for product visualization, collaboration, CAD data exchange, and for long-term data retention.





STL

solid CREO STL

facet normal 0.000000e+00 0.000000e+00 -1.000000e+00

outer loop

vertex 5.000000e+01 -5.000000e+01 0.000000e+00

vertex -5.000000e+01 5.000000e+01 0.000000e+00

vertex 5.000000e+01 5.000000e+01 0.000000e+00

endloop

endfacet

facet normal 0.000000e+00 1.000000e+00 0.000000e+00

outer loop

vertex 5.000000e+01 5.000000e+01 6.000000e+00

vertex 5.000000e+01 5.000000e+01 0.000000e+00

vertex -5.000000e+01 5.000000e+01 0.000000e+00

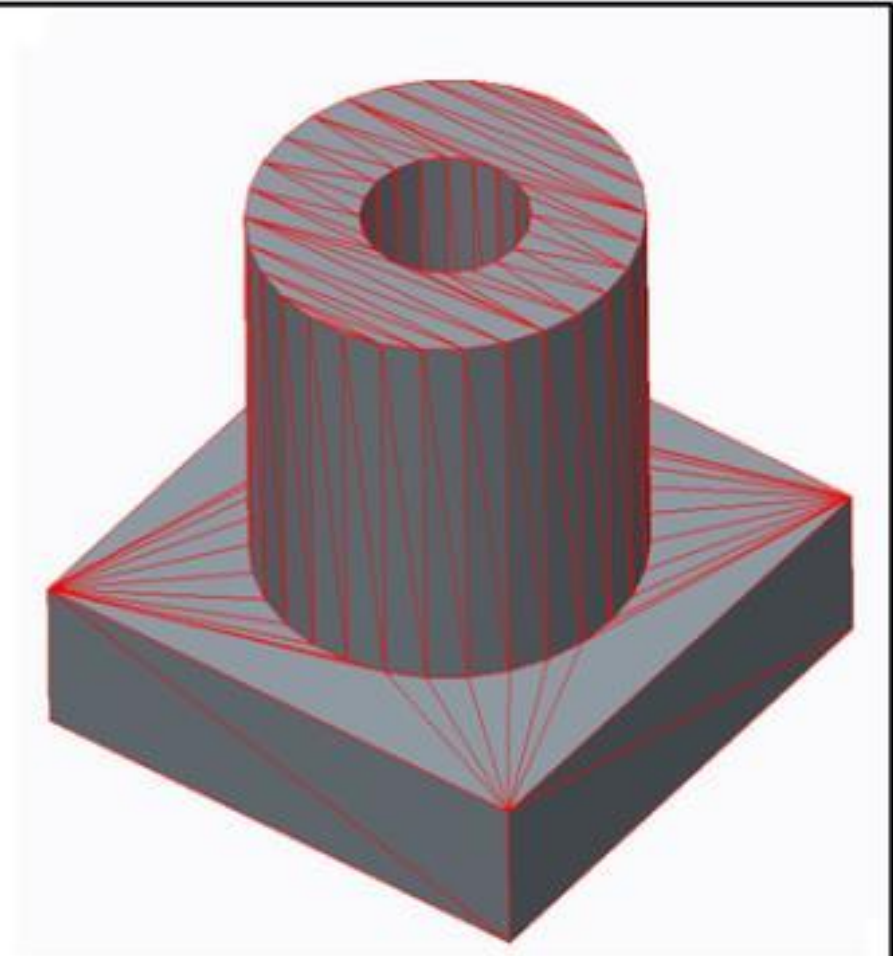
endloop

endfacet

.....

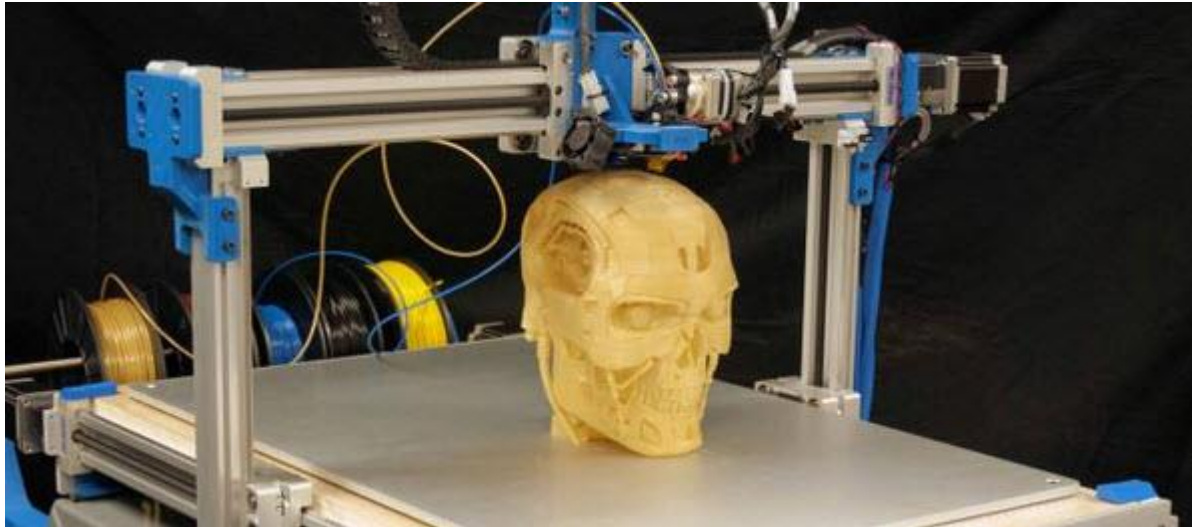
.....

endsolid CREO STL





Formats for 3D printing



There are a lot of 3D file formats that are used by [3D printing designers](#) today.

But STL, OBJ, AMF, and 3MF are the most important among all of them.



AMF

Additive manufacturing file format () is an **open standard** for describing objects for **additive manufacturing** processes such as **3D printing**.

The official **standard** is an **XML**-based format designed to allow any **computer-aided design** software to describe the shape and composition of any **3D** object to be fabricated on any **3D printer**. Unlike its predecessor **STL format**,

AMF has native support for color, materials, lattices, and constellations.



File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render T-Splines Help

Perspective

Ardiano Vi...
Main1
Main2

Layers - All Layers

Name	Material L...	Lir
Curves		Co
Images		Co
Surfaces		Co
Layer 03		Co
Layer 04		Co
Layer 05		Co

Properties

Viewport

Title	Perspective
Width	1401
Height	805
Projection	Perspective

Camera

Lens Length	50.0
X Location	150.797
Y Location	281.791
Z Location	151.704
Location	Place...

Target

X Target	-18.457
Y Target	53.897
Z Target	-24.304
Location	Place...

Wallpaper

Filename	(none)
Show	<input checked="" type="checkbox"/>
Gray	<input checked="" type="checkbox"/>

File successfully written as C:\Users\Mark Landsaat\Desktop\NURBS_Hand.igs

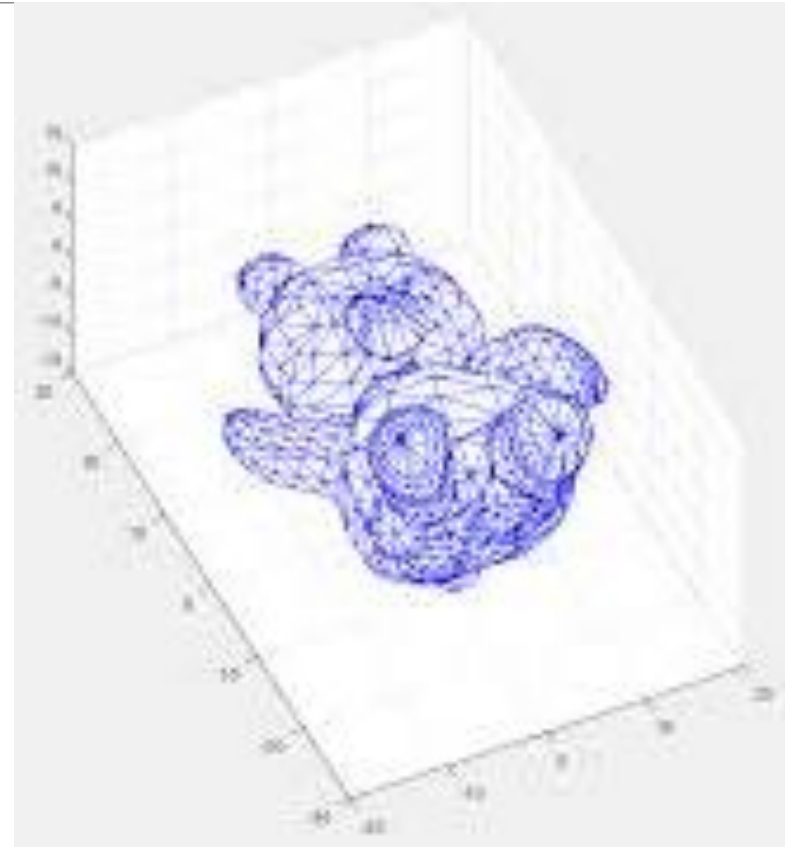
Command: |

CPlane x -46.178 y 185.720 z 0.000 Curves Snap Ortho Planar Osnap Record History



OBJ

The **OBJ file format** is a simple data-format that represents 3D geometry alone — namely, the position of each vertex, the UV position of each texture coordinate vertex, vertex normals, and the faces that make each polygon defined as a list of vertices, and texture vertices





3MF

A **3MF file** is an archive saved in the 3D Manufacturing **Format (3MF)**, which is used by various 3D design programs to save 3D models. It contains a 3D model, material, and property information compressed with Zip compression. **3MF files** may also store a print ticket, thumbnail image, and one or more digital signatures.

