

SolidWorks Data Exchange FAQ

What file formats can be imported in SolidWorks?

The table below contains the file formats that can be imported in SolidWorks

Application	File Format	Version
ACIS	.sat (parts & assys)	Up to R16
Adobe Illustrator	.ai	
Adobe PhotoShop	.psd	
Autodesk Inventor	Inventor Part (.ipt) & Assembly (.iam)	11.0 or above
CADKEY	.prt (part or assy), .ckd (multi-body part)	19 (.prt) or 21(ckd)
CATIA Graphics	.cgr	V5.0 (R1-17)
DXF/DWG	.dxf, .dwg	13 or higher
DXF 3D	.dxf	R14 or higher
IDF	.emn, .brd, .bdf, .idb	
I-DEAS	.x_t or .step	
IGES	.iges or .igs	
Mechanical Desktop	.mdt (part or assembly), Mechanical Desktop must be installed on same system with SolidWorks for translator to function	MDT 4.0 or higher
Parasolid	.x_t, .x_b, .xmt_txt, .xmt_bin	All versions up to v18.0
Pro/ENGINEER	ProE Part (.prt, .prt, .xpr) ProE Assembly (.asm, .asm, .xas)	v17 - 2001 & Wildfire 1, 2 & 3
Rhino	.3dm	
ScanTo3D	Mesh Files (.nzip;.nxm;.scn;.3ds;.obj;.stl;.wrl;.ply;.ply2) PointCloud Files (.xyz;.txt;.asc;.vda;.igs;.ibl)	
Solid Edge	Solid Edge Part (.par, .psm) Solid Edge Assembly (.asm)	Current version
STEP	.step	
STL	.stl	
Unigraphics	UGII & NX parts & assemblies (.prt)	10 or higher and NX up to NX 4
VDAFS	.vda	
VRML	.wrl	1.0 & 2.0 (97)

Can I export SolidWorks documents to be used in other applications?

Yes, you can export SolidWorks documents to a number of formats that can be used in other applications. The table below contains the different file formats that are available for export.

Application	File Format	Parts	Assemblies	Drawings
3D XML	.3dxml	X	X	
ACIS	.sat	X	X	
Adobe PDF	.pdf	X	X	X
CATIA Graphics	.cgr	X	X	
DXF/DWG	.dxf/.dwg			X
eDrawings	.eprt - Part doc. .easm - Assembly docs. .edrw - Drawing docs	X	X	X
Highly Compressed Graphics	.hcg	X	X	
HOOPS	.hsf	X	X	
IGES	.iges	X	X	
JPEG	.jpeg	X	X	X
Parasolid	.x_t or .x_b	X	X	
PDF	.pdf	X	X	X
Pro/ENGINEER	ProE Part (.prt) ProE Assembly (.asm)	X	X	
STEP	.step	X	X	
STL	.stl	X	X	
TIFF	.tif	X	X	X
U3D	.u3d	X	X	
VDAFS	.vda	X		
Viewpoint	.mts	X	X	
VRML	.wrl	X	X	
XPS	.eprt; .easm; .edrw	X	X	X

When I import a neutral file (Parasolid, ACIS, Step or IGES), there are no features in the model.

When importing a neutral file, the feature data is not included in the file, only the geometric data is supported. As a result, only the shape and size of the object is transferred and not the history of how it was made. If the data imports and forms a solid, subsequent features can be added to the solid; such as cuts, bosses, fillets, etc; but the shape of the original solid cannot be changed. A data translator is required if you would like to 'recognize' the features of the solid.

There are no features in the imported model, is there a way to recognize the features?

A translator is required to recognize the features of an imported model. FeatureWorks is a SolidWorks Add-in product that recognizes features on an imported solid body in a SolidWorks part document. Recognized features are the same as features that you create using the SolidWorks software. You can edit the definition of recognized features to change their parameters. For features that are based on sketches, after you recognize the features, you can edit the sketches from the SolidWorks FeatureManager design tree to change the geometry of the features.

What features would FeatureWorks recognize?

FeatureWorks recognizes the following features:

- Extruded or revolved features
- Chamfers on linear or circular edges
- Constant or variable radius fillets on linear or circular edges
- *Ribs*: extruded parallel to sketch, extruded normal to sketch, and ribs with negative draft.
- Draft features
- *Holes*: With automatic or interactive feature recognition, you can recognize these types of holes: simple, simple drilled, taper, taper drilled, countersunk, countersunk drilled, counter bored, counter bored drilled, counter drilled, and counter drilled drilled.
- You can also recognize Hole Wizard holes.
- Shells
- *Sweeps*: Interactively recognize base-sweeps.
- Volume features
- *Feature patterns*: linear, circular, and rectangular.
- *Sheet metal features*: base flanges, edge flanges, sketched bends, hems, and miter flanges.
- *Sketch Patterns*: Using interactive recognition, you can create a sketch pattern from similar features that were created randomly. Partial imprints of features cannot be recognized. Creating a pattern of a pattern feature is not supported.
- *Multibody parts*: Recognize multibody parts one body at a time.



What can I do with my DXF/DWG files after they are brought into?

You can leverage your 2D legacy data by reading in and exporting out DXF/DWG data for reuse in a SolidWorks environment. SolidWorks eases data reuse by allowing you to create 3D parametric solid models from the imported 2D data by turning it into sketch geometry. SolidWorks can also import 3D DXF/DWG data as curves in the part environment that can be used to create solid models. 3D DXF files are also supported for import within SolidWorks.

Does SolidWorks have any way to read in data from Mechanical Desktop?

Yes. A translator for Mechanical Desktop files (XchangeWorks) is available for SolidWorks. This is a feature-based converter for files created using Mechanical Desktop 4.0 and higher. The conversion utility provides direct translation of files created with Mechanical Desktop into the SolidWorks environment, complete with feature information. In the past, users migrating from Mechanical Desktop had to export data to a neutral file format and import the data. Mechanical Desktop must already be present on the system during the SolidWorks installation process in order for this translator to be installed.

Does SolidWorks have any way to read in data from Pro/Engineer?

SolidWorks includes a feature-based converter for Pro/ENGINEER files. The SolidWorks Pro/E Converter recognizes most features in parts and assemblies in files created from Versions 17 through 2001 of Pro/ENGINEER and Wildfire versions 1, 2 & 3.

Can I read-in native Unigraphics part files?

Yes. SolidWorks includes a translator which allows for the import of native Unigraphics part and assembly files. SolidWorks and Unigraphics use the same modeling kernel – Parasolid®. Instead of sending the data through translators such as IGES and STEP, the direct UG .prt files can be opened directly by SolidWorks. This capability allows you to bypass IGES and STEP, providing you with a cleaner data exchange between SolidWorks and Unigraphics products. Parts and assemblies from UGII versions 10 or higher, and up to NX 4, can be imported.

How does SolidWorks handle Catia data?

SolidWorks and Catia are both owned by Dassault Systemes. SolidWorks exchanges data with Catia by IGES, STEP and directly by the third party products. We have developed a document that will assist customers in properly setting up their system parameters for dealing with Catia data. The third-party products we recommend is FormatWorks (SolidWorks add-in) or 3DTransvidia (stand-alone app), both are products from CapVidia.

See www.javelin-tech.com and www.capvidia.com for further information.