



Preferred File Types for Import into Solidworks

Each product has its own considerations that must be taken into account in order to make the most of imported files from other CAD systems. The following is a listing of the preferred file types for common CAD systems in order of probability of success. If your CAD system is not listed please contact us.

Unigraphics: Parasolid (.xmt_txt, .xmt_bin, .x_t, .x_b)
UGII (Unigraphics Native .prt)

Although we have the ability to read Native Unigraphics files, we prefer Parasolid over UGII when given a choice. UG native files also contain all the construction geometry which is unnecessary when importing a part of assembly. The Parasolid file strips all the unnecessary data from the file and makes the import much faster.

Furthermore, UG Native assemblies (.prt) externally reference the part files which will also need to be included in any dataset. A Parasolid file of an assembly contains the assembly and ALL the part files in a single file.

SDRC Ideas: Parasolid (.xmt_txt, .xmt_bin, .x_t, .x_b) starting with V9 of IDEAS
STEP (.stp, .step)
IGES (.igs, .iges)

Catia: Catia Native (.model, .exp, .session) using 3rd party translator for SolidWorks.
STEP (.stp, .step)
IGES (.igs, .iges) setting Catia to 14 decimal precision ([See Sheet 2](#))

Pro Engineer: Pro/E native (.prt, .xpr, .asm, .xas)
STEP (.stp, .step)
IGES (.igs, .iges)

AutoCAD 2D: AutoCAD Native (.dwg) R12, 13, 14, 2000, 2002, 2004, 2005 and 2006
Includes Layers, Line Types, Fonts, Colors, Blocks, and Attributes.

AutoCAD MDT: ACIS (.sat)
STEP (.stp, .step)
IGES (.igs, .iges)

Inventor: Inventor Native (.ipt)

SolidEdge: Parasolid (.xmt_txt, .xmt_bin, .x_t, .x_b)
SolidEdge Native Part (.par)
SolidEdge Native Sheet Metal Part (.psm)
SolidEdge Native Assembly (.asm)

CADKEY: Cadkey Native (.prt, .ckd)

File types also accepted by SolidWorks:

Adobe Illustrator (.ai)
Circuit Board Layouts IDF (.emn, .brd, .bdf, .idb)
Stereolithography (.stl)
VDAFS (.vda)
VRML (.wrl)
Catia Graphics (.cgr) graphics only



Setting Catia to 14 Decimal Place Precision:

The Catia IGES translator is based on IBM IGES Format (IIF). When you save a Catia model as an IGES file, it undergoes two translations: Catia=>IIF=>IGES.

The IIF=>IGES step is done by the igesp program. This program has two parameters which affect the accuracy of the model and therefore increase the potential that SolidWorks will be able to form a solid out of the data. You can modify these values by editing the file igesinp.data. The 2 parameters which should be changed are:

- SIGFIG COEF **n**
- SIGFIG CORD **n**

Where **n** is the number of significant digits. The default values are 8 and 6 respectively. The range of values are from 5 to 14. The recommended values are 14 and 14.

- COEF indicates the maximum number of significant digits in the IGES file for a real number that is a coefficient of an equation.
- CORD indicates the maximum number of significant digits in the IGES file for a real number that is a coordinate.

By using the maximum values for these parameters, the size of the IGES file can become large, but the potential that SolidWorks will be able to form a solid out of the data is much greater.