



Importing Files with XchangeWorks

This manual describes the following XchangeWorks topics:

- ❑ Importing the following part file types:
 - ACIS®
 - Autodesk Inventor®
 - CADKEY®
 - IGES
 - Parasolid®
 - Pro/ENGINEER®
 - Solid Edge®
 - SolidWorks®
 - STEP
 - Unigraphics™
 - VDAFS
- ❑ Setting import options
- ❑ Finding more information about XchangeWorks

Importing Part Files

You can use the XchangeWorks software to import part files from other applications. The following file formats are supported:

ACIS (*.sat)	Solid Edge (*.par)
Autodesk Inventor (*.ipt)	SolidWorks (*.sldprt)
CADKEY (*.prt)	STEP (*.step)
IGES (*.igs)	Unigraphics (*.prt)
Parasolid (*.x_t, *.x_b)	VDAFS (*.vda)
Pro/ENGINEER (*.prt, *.*)	



You cannot import assembly files with XchangeWorks.

To import a part file from another application:

- 1 For ACIS, Autodesk Inventor, IGES, STEP, or VDAFS files, set the necessary import options. See the next section, **Setting Import Options**, for more information. No options are necessary for the other file types.
- 2 Click **Import**  on the XchangeWorks toolbar.
The **Open** dialog box appears.
- 3 Select the desired format from the **Files of type** list.
- 4 Browse to the desired file, and click **Open**.
- 5 Click **OK** to open the file.

If the attempt to form a solid succeeds, a solid body appears in the graphics area. You can add features (bosses, cuts, and so on) to this base feature, but you cannot edit the base feature itself.

If the attempt fails, an AutoCAD® body object appears in the display window. The body consists of trimmed or bounded surfaces.

Error and report files (.err and .rpt) are written to the same folder as the file you imported. These files list any errors encountered or unsupported entities. Error files are written for IGES, STEP, ACIS or VDAFS files; report files are only written for IGES files.

Setting Import Options

You can specify the XchangeWorks application's default response to importing files of other formats. The import options affect only the surface-based translators (IGES, ACIS, STEP, and VDAFS) plus Autodesk Inventor.

To set options for importing files:

- 1 Click **Part Modeler**  on the XchangeWorks toolbar.

The Part Modeler appears.

- 2 Click **File, Open**.
- 3 In the **Files of type** list, click the desired format.
- 4 Click **Options**.

The **Import Options** dialog box appears.

- 5 Select from the following options:
 - **Surface/solid entities**. Imports surface and solid entities. You must also select one of the following options:
 - ☐ **Try forming solid(s)**. Tries to form solids. Additionally, you can select the following option:
 - B-REP mapping**. Attempts to import the model by directly mapping topologies using boundary representation (BREP) data. In general, this mode is faster than knitting, especially for complex models. If you select the **Try forming solid(s)** option, but do not select the **B-REP mapping** option, the SolidWorks application attempts to knit the surfaces into solid(s).
 - ☐ **Knit surface(s)**. Imports as surfaces and attempts to knit the imported surfaces.
 - ☐ **Do not knit**. Imports as surfaces and prevents surfaces from knitting.
 - **Free point/curve entities**. Imports free points and free curve entities. You must also select one of the following options:
 - ☐ **Import as sketch(es)**. Imports data as 2D and 3D sketch data. If you want to import free curves, SolidWorks strongly recommends you select this option, for performance reasons. Free points and 2D sketches import as 2D sketches. 2D curves, 3D curves, and 3D sketches import as 3D sketches.
 - ☐ **Import as 3D curves**. Imports data as 3D curves data. 2D and 3D curves import as curves. Free points and 2D sketches import as 2D sketches.
 - **Import multiple bodies as parts**. This option has no impact on importing multibody parts because XchangeWorks always imports a multibody part as a part document that contains multiple imported bodies. You cannot import assemblies with XchangeWorks.

- **Perform full entity check and repair errors.** With this option selected, import performance is slower because the software spends more time checking and repairing the model entities wherever possible. If the quality of the imported data is good, you may not have to select this option.
 - **Customize curve tolerance.** Customizes the tolerance when you import models with very small entities (smallest values on the order of 1.0e-6 to 1.0e-7 meters). With this option cleared, SolidWorks uses internal tolerance settings which are too large to properly import and display these small models. Enter the desired tolerance in the box.
 - **Unit.** Set the units of measure for the imported file. You must select one of the following options:
 - ☐ **File specified unit.** Use the units of measure from the imported file.
 - ☐ **Document template specified unit.** Use the units specified in the SolidWorks template files under **Tools, Options, System Options, Default Templates.**
 - **IGES - Show IGES levels.** Displays the **IGES-In Surfaces, Curves, and Levels** dialog box if there are curves or different levels (or layers) in the IGES file.
 - **STEP - Map configuration data.** Applies to STEP files only. Imports STEP file configuration data plus geometric data. Leave this option cleared to import only geometric data.
- 6 Click **OK** to return to the **Open** dialog box.
- 7 Click **Alt+Tab** to return to the Autodesk® software application.

Finding More Information about XchangeWorks

For further information about XchangeWorks, please visit:
<http://www.xchangeworks.com>

For a list of frequently asked questions, please visit:
<http://www.xchangeworks.com/faq.htm>