

Support Sheet: Conversion Settings for SDRC

The following is a list of recommended settings for SDRC that have been determined to give the best results when passing data from SDRC to SolidWorks.

When converting files out of SDRC use of the following procedure usually results in successful translation of SDRC files.

Importing an IGES Part file from I-DEAS to SolidWorks

In I-DEAS:

- 1) Go to File/Export/IGES and select OK on the Export form.
- 2) The Prompt region will ask you to "Pick Entity". Pick a part - a white bounding box appears around it.
- 3) On the IGES Export form enter the Filename and Pick Flavors.
- 4) On the IGES Flavors form, toggle on "Use Standard Flavors". Select the ? and pick Number 14 - Unigraphics.
- 5) Select OK in the IGES Flavor forms.
- 6) When you reach the IGES Export Form select the Export button and then OK the I-DEAS Warning message which tells you that a new IGES file will be created.
- 7) Send the IGES file to the system which will import it into SolidWorks.

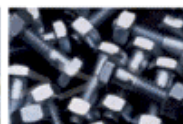
In SolidWorks:

- 1) Go to File/Open.
- 2) Change "Files of type" to Iges Files (*.igs) in the File Open dialog.
- 3) Browse to find the directory with the Iges file and move it to the "file name" box by clicking on it.
- 4) Finally Click on the "open" button and the Iges file will be imported.

Importing an IGES Assembly file from I-DEAS to SolidWorks

In I-DEAS:

- 1) Go to File/Export/IGES and select OK on the Export Form.
- 2) The Prompt region will ask you to "Pick Entity". Grab all the assembly instances by holding down mouse button one and drawing a box around them. Alternatively when it says "Pick Entity" use mouse button 3, Get From Bin and select the assembly.
- 3) On the IGES Export Form enter the Filename and Pick Flavors.



- 4) On the IGES Flavors form toggle on "Use Standard Flavors". Select the ? and pick Number 14 - Unigraphics.
- 5) OK the Flavor and IGES Flavor forms.
- 6) When you reach the IGES Export Form select the Export button and then OK the I-DEAS Warning which tells you that a new IGES file will be created.
- 7) Send the IGES file to the system, which will import it into SolidWorks

In SolidWorks:

- 1) Go to File/Open.
- 2) Change "Files of type" to Iges Files (*.igs).
- 3) Browse to find the directory with the Iges file and move it to the "file name" box by clicking on it.
- 4) Click on the "open" button and the Iges file will be imported. The file imported will include the assembly as well as all the parts.

