autodesk

Mechanical Desktop Translator

Technical Comparison

Inventor R3 vs. SolidWorks 2000

SolidWorks Corporation has recently released a Mechanical Desktop™ 4 to SolidWorks™ 2000 feature based translator. The goal of this document is to educate and show the main differences between the SolidWorks 2000 and Autodesk Inventor® R3 translators.

Quick Comparison Chart

Features of Mechanical Desktop	Autodesk Inventor	SolidWorks 2000
Maintain Assembly constraints	**YES,** all mating conditions are preserved	**NO** assembly constraints are maintained
Transfer open sketches	**YES**	**NO.** Feature will fail without any notification to user
Translate Parametric Booleans	**YES**	**NO**
Warning or error messages during translation	**YES**	**NO**
Stability	**YES.** Extremely Stable	**NO.** Crashes often
Windows 2000 support	**YES**	**NO**

	Autodesk Inventor	SolidWorks 2000
Features:		
Extrudes	✓	✓
Linear	✓	✓
From-To	✓	
To Face/Plane	✓	Limitations
Sweep	✓	Limitations
Loft	✓	✓
Revolve	✓	✓
Hole Feature	✓	✓
Sketch Features:		
Project Constraints	✓	
Relational Constraints	✓	
Sketch Constraints	✓	
Open Sketches	✓	
Assemblies:		
Assembly Constraints	✓	
Operating Systems:		
Windows NT 4 sp3+	✓	✓
Windows 2000	✓	

As you will see by these comparisons SolidWorks provides less than 60% of the functionality contained in the Mechanical Desktop to Autodesk Inventor translator.

Three different benchmarks were conducted during the evaluation of both translators in order to emulate a Mechanical Desktop user's environment:

1. Translate part models ranging from simple to complex.

2. Translate small hybrid (local and non-local) assemblies.

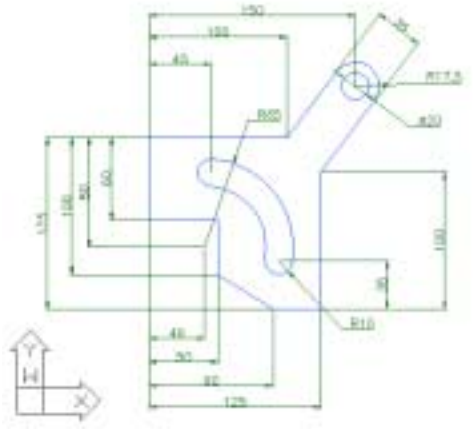
3. Translate large assemblies starting with 10 parts to 500 parts.

Feature Topology/Single Parts

SolidWorks is able to translate simple features, but it is unable to translate the geometric or dimensional constraint information contained within the feature. This is a problem because it forces the user to go back to the geometry and constrain the sketch for predictability of updates.

Autodesk Inventor automatically translates all of the feature data from Mechanical Desktop including the geometric and dimensional constraint information.

Here is an example of a simple multi-loop profile in Mechanical Desktop that has been fully constrained.



In todays solids modeling environments it is not necessary to fully constrain sketches. But if the user wants to drive the part with spreadsheets or apply design rules, the sketch must have the geometric and dimensional constraints applied to what will change.

Both translators brought in the part model with no errors as shown.

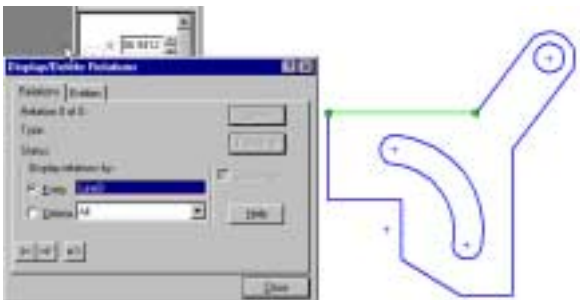
Autodesk Inventor

SolidWorks 2000

The main concern is how the sketch geometry is translated. Autodesk Inventor translated all geometric and dimensional constraints while maintaining the original design intent.

Autodesk Inventor

SolidWorks translated the features but did not translate any geometric or dimensional constraints as shown in the Display/Delete Relations dialogue box. In order to have this design intent applied the user must recreate all constraint data within SolidWorks.

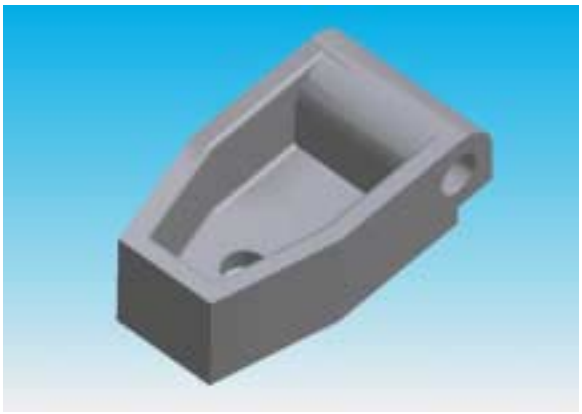


SolidWorks 2000

A customer's Mechanical Desktop part model was used for the next test.

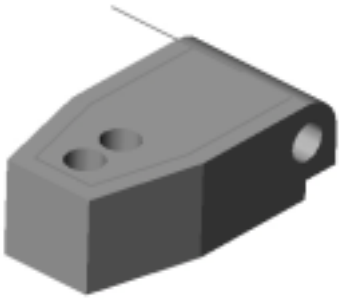
Mechanical Desktop

Autodesk Inventor translated the Mechanical Desktop part 100% with no failures.



Autodesk Inventor

SolidWorks is unable to translate Mechanical Desktop project constraints; therefore the sketch that creates the pocket fails to translate as shown.



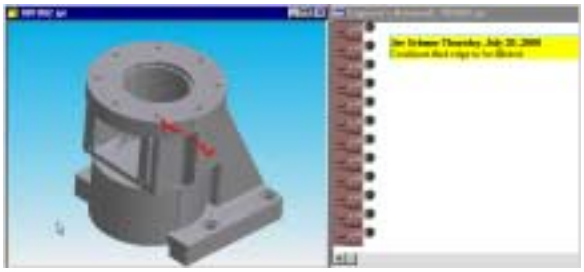
SolidWorks 2000

Any and all subsequent features dependant upon failed features also fail to translate in SolidWorks.

What happens if a part fails?
SolidWorks shows a dialog box during translation that effectively notes some feature(s) failed. SolidWorks does not explicitly state what feature(s) failed or why there was a failure.

Autodesk Inventor documents errors with the Engineers Notebook, an innovative documentation tool used to transfer design intent, assembly documentation, and team collaboration.

Autodesk Inventor will add a note on the feature (or failed feature location) noting what feature failed and why it failed.



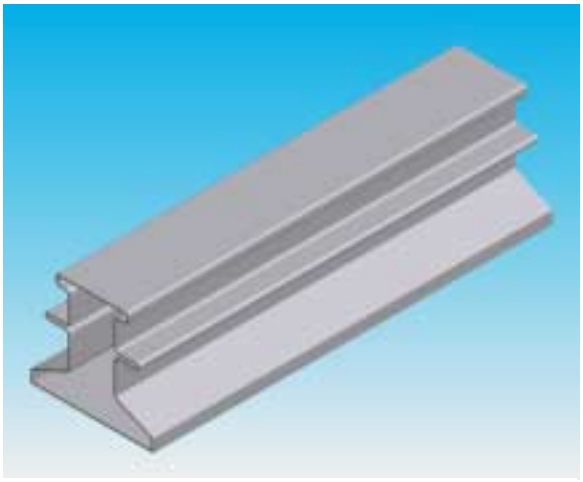
Autodesk Inventor's Engineers Notebook

Direct comparisons between the two translators reveal that Autodesk Inventor's translations are far more accurate and intelligent.

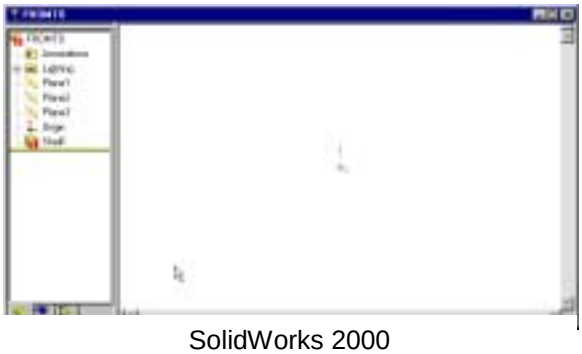
Focusing on the features listed in the chart on page 2, let's consider the following examples:

From/To Extrusion – an extrusion that is required to transition from one face/plane to another face/plane.

Mechanical Desktop



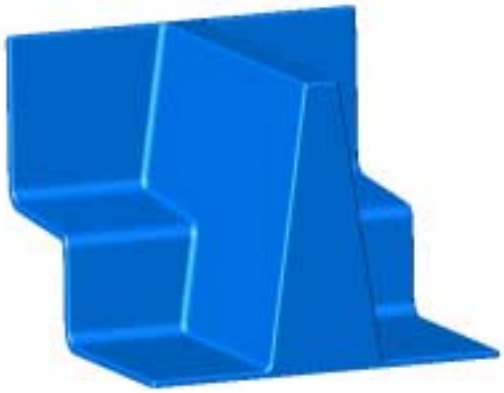
Autodesk Inventor



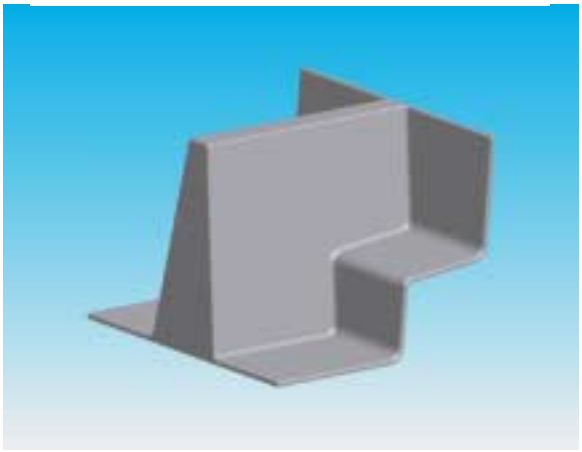
SolidWorks 2000

SolidWorks failed on the translation of this extrusion because they do not support a From/To extrusion type.

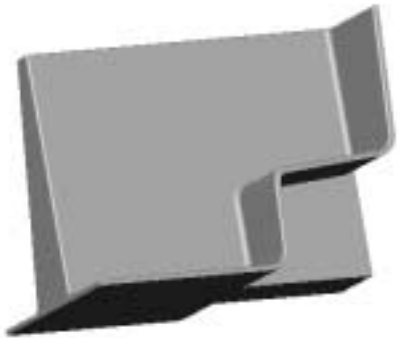
To/Face Extrusion – is used to go from a sketch plane to some generally odd shaped face.



Mechanical Desktop



Autodesk Inventor

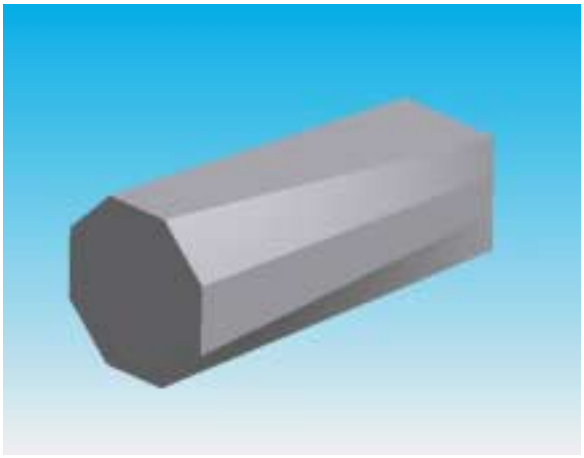


SolidWorks 2000

SolidWorks is unable extract the correct termination face causing the extrusion to terminate to the back face.

Loft Extrusion - used to create topology transitions from one sketch/face to another.

Mechanical Desktop



Autodesk Inventor

SolidWorks 2000

SolidWorks completes the loft but misses the point mapping. This creates an inaccurate twist effect on the loft feature.

Key Points:

- Autodesk Inventor translates dimensional constraints. SolidWorks does not.
- Autodesk Inventor translates geometric constraints. SolidWorks does not.
- Autodesk Inventor translates open sketches. SolidWorks does not.
- Autodesk Inventor translates 100% of Mechanical Desktops extrusion features. SolidWorks does not.

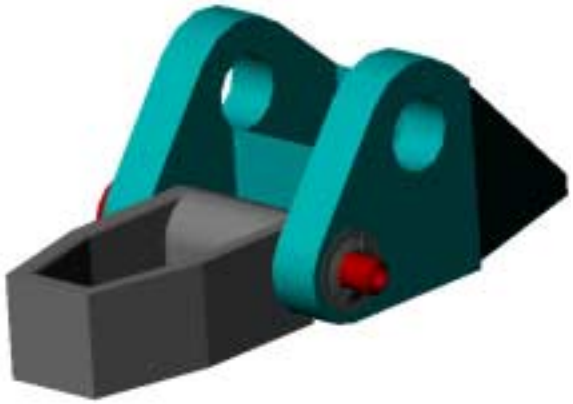
Translating of Assemblies:

Assemblies that fail will cause many problems for the SolidWorks user. SolidWorks will show random dialog boxes during the assembly translation. At the end of the translation there are no documented failures. This forces the user to manually sort through the entire assembly to find the failures. This manual method is prone to errors and may result in newer problems being introduced into the design.

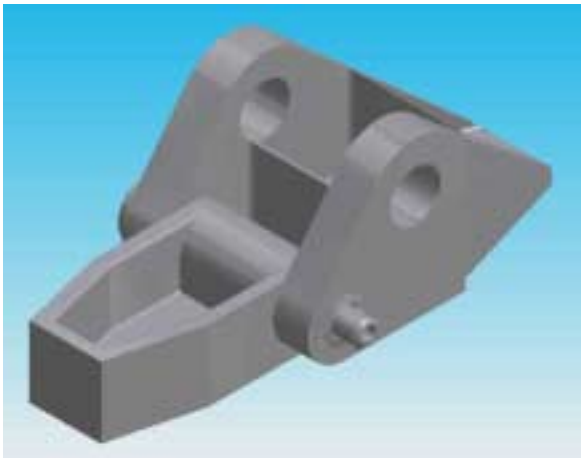
Autodesk Inventor creates the Engineers Notebook file(s) with small sticky notes on the failure(s) as visual clues. This communicates to the user exactly where the failure occurred and why. This allows the user to invoke the Design Doctor to diagnose and repair the reported problem.

Test Case 1 - Hinge Assembly

SolidWorks failed to translate 25% of the assembly. Autodesk Inventor translated 100% of the assembly.



Mechanical Desktop



Autodesk Inventor

SolidWorks 2000

Assembly			Autodesk Inventor		SolidWorks	
Assembly/Part	Number of Parts	Number of Unique Parts	Time to Translate	Number of Errors	Time to Translate	Number of Errors
Hinge	4	4	157	0	156	2

Test Case 2 – Valve Assembly

Mechanical Desktop

Autodesk Inventor

SolidWorks 2000

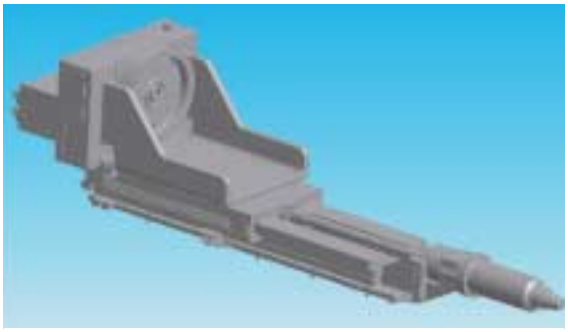
SolidWorks failed to translate over 74% of the assembly. Autodesk Inventor translated over 93.5% of the assembly.

Where Inventor failed to translate the assembly, Engineers Notebook notified the user of what failed and how to resolve the failure.

Assembly			Autodesk Inventor		SolidWorks	
Assembly/Part	Number of Parts	Number of Unique Parts	Time to Translate	Number of Errors	Time to Translate	Number of Errors
Valve	37	31	543	2	501	23

Test Case 3 – Spindle Assembly

Mechanical Desktop



Autodesk Inventor

SolidWorks 2000

SolidWorks failed to translate over 58% of the assembly. Autodesk Inventor translated over 99% of the assembly correct.

Assembly			Autodesk Inventor		SolidWorks	
Assembly/Part	Number of Parts	Number of Unique Parts	Time to Translate	Number of Errors	Time to Translate	Number of Errors
Spindle	554	106	3221	1	3201	44

Key Points:

- Autodesk Inventor translates assembly constraints. SolidWorks does not.
- Of the three test cases completed Autodesk Inventor had a success factor of over 97.5% while SolidWorks had a success factor of less than 48%.

Conclusion:

- SolidWorks does not translate dimensional constraints.
- SolidWorks does not translate geometric constraints.
- SolidWorks does not translate assembly constraints.
- SolidWorks failed on extrusions that were beyond simple extrudes or revolves.

The SolidWorks Mechanical Desktop translator is lacking too many features to perform file migration with confidence and provide results you can trust. Designers will be forced to constrain the part sketches, add design intent, add assembly relationships, and then repair the failures, all manually. The requirement for all this manual labor also introduces a high probability of introducing new errors into your entire design process. Clearly, the SolidWorks translator does not ensure that the end result is of high quality.

The solution is clear; Autodesk Inventor is the best decision for Mechanical Desktop users wishing to move to the next step in 3D design and collaboration.

More Information

For More Information go to the Autodesk, Inc. web site at www.autodesk.com
Or call your local Mechanical Reseller, for live demonstrations of the full suite of Mechanical Solutions from Autodesk, Inc.