

**TechniCom Tests Digital Prototyping Workflows
Comparing:**

**Autodesk Inventor Professional 2011
with
SolidWorks Premium 2011**

Final Report

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Executive Summary

Background

Last Summer (August 2010) TechniCom Group published a report comparing Autodesk Inventor and Dassault Systemes SolidWorks using our Delphi Expert Analysis methodology¹. The results of this report were somewhat controversial; Autodesk Inventor scored better in all fifteen categories considered than did SolidWorks. The scoring for the Delphi Expert report was the result of a very detailed survey of eight expert users of the two systems, four experts for each system. The experts had comparable familiarity with their systems and comparable backgrounds.

Readers of that report evidenced hunger for more detailed information, one that might be less sensitive to opinions and be more factual. As a result, TechniCom worked with Autodesk to develop a series of tests between the two systems that might expose the differences between the two systems and perhaps highlight advantages Inventor might have as compared to SolidWorks.

Deciding what to test

First we had to decide what to test and the scope of the testing.

Followers of the mechanical CAD market are no doubt aware of the term Product Lifecycle Management, often designated as PLM. Autodesk's mechanical philosophy is to eschew developing PLM software in favor of digital prototyping.

The term "Digital Prototyping" has led to some confusion in the industry. One clear definition comes from IDC in a paper entitled "Digital Prototyping: Autodesk Strengthens Competitiveness of Worldwide SMB Manufacturers", published October 2008. This whitepaper differentiates digital prototyping from PLM by noting that "PLM reaches from a product's cradle to its grave. On the other hand, digital prototyping stops at the completion of the digital product and its engineering bill of materials . . . The beauty of digital prototyping is that designs can be tested out before they go to manufacturing."

Thus, Autodesk's definition of digital prototyping includes the basic functions of PLM — industrial design, design and engineering, data vaulting, and collaboration, without the post-manufacturing baggage.

Autodesk has been carefully steering its Inventor software product development over the past few years to enable workflows that take maximum advantage of seamlessly passing data among its built-in application solutions. Thus, what we see in Inventor today is a careful melding of technologies that Autodesk has acquired or built. Many of these technologies are not available as extra cost add-ons to the base software, but fully included as part of the Inventor software. Some example, of which you will see more later, include mold analysis software, mold base design capabilities, built-in advanced simulation, inherent design automation options, an intelligent part library, built-in engineering calculations, and many others. Not only are these available as an integrated part of Autodesk Inventor, but they are often combined to form workflows that aid in developing the digital engineering models.

Thus, when deciding the scope of what to test, we settled on a series of tests that focus on the areas in our Delphi Expert analysis where Inventor rated the highest. These areas include the following:

1. Plastic Part Design
2. Plastic Injection Mold Design

¹ "Comparing the Capabilities of Autodesk Inventor Professional 2011 and SolidWorks Premium 2010 Using TechniCom's Delphi Expert Technique", 9 August 2010, a paper by TechniCom Group, available at <http://www.cad-portal.com>.

3. Assembly Design and Analysis
4. Exporting BIM-ready Models
5. Interoperability
6. Design Automation
7. Mechatronics

Even deciding on these seven areas leaves a great many options to be tested. Autodesk decided on the detailed functions to be tested. Autodesk has specified the seven tests in detail. They are aimed at comparing the two systems ability to perform common, real-world engineering workflows. These tests are not designed to be impartial; they are taken from standard demos used by Autodesk that were designed to represent a series of engineering workflows highlighting Inventor's digital prototyping capabilities. Most of them, as the users can see below, are aimed at performing a complete design sequence. We have included, in the following seven sections, the details of what we tested, images and videos of the results, what we observed comparing the two systems, and our summary of how well each system was able to perform the desired workflow.

Tests specified by Autodesk

Autodesk provided TechniCom with the test definitions including videos of Inventor performing the desired task, starter geometry, related dimensions, and other relevant data, all described below within each test section. *TechniCom's task was to perform the same tests using SolidWorks Premium 2011.* Because Autodesk provided much of the model data we were able to focus on the desired workflow details of each test, rather than building geometry.

Full disclosure

Autodesk paid TechniCom to perform these tests and to document the results in this report.

Our approach

TechniCom, in collaboration with a Certified SolidWorks Professional (CSWP) performed and analyzed these tests during November and December 2010 using Inventor Professional 2011 and SolidWorks Premium 2011. To make the scope reasonable, we limited each vendor's software strictly to what was included with the package or third party add-ins that we were able to find and download free of charge.

As input, we used the Inventor videos illustrating the work to be performed. We attempted to deliver the same results using SolidWorks Premium 2011, as did Inventor.

We are making available to the readers of this report, annotated videos of both Inventor and SolidWorks performing the tests on TechniCom's blog at www.raykurland.com. Readers wanting to understand how the two products compared have the unique ability to review these videos along with reading our test summaries in this report.

We remind the reader that we compared Inventor Professional 2011 versus SolidWorks Premium 2011 with the restriction that extra cost third party software was not to be considered. When we were able, we used no-charge third party add-ins for SolidWorks — none were needed for Inventor.

Summary of the test results

In the first two tests, plastic part design and injection mold design, Inventor clearly outclasses SolidWorks. Whereas Inventor completed all aspects of the test, SolidWorks was unable to complete major portions of the analysis of the part and the mold. Inventor was also able to design the mold significantly faster than SolidWorks due to the inclusion of automated tools for designing the various subsystems of the mold.

For the assembly design and analysis test, both systems were able to model the addition of a clevis pin. However, Inventor excelled in its ability to design the correct pin by coupling its engineering calculation library to the potential design. In other words, Inventor helped select the correct pin size because it was able to use its calculations concerning the required

stress that the pin would need to perform correctly. This is subtly different than SolidWorks, which used its library to size the pin, but without taking into account its stress requirements. The SolidWorks approach was to design the pin and then analyze it in an iterative fashion using its built-in FEA solution until the specifications were met. In this case SolidWorks was unable to verify that its built-in FEA solution was correct. A more advanced version of the FEA solver would have been required; concomitant with more advanced engineering skill.

The latest release of SolidWorks added some BIM exchange capabilities, but Inventor's BIM data transfer capabilities exceeded SolidWorks in key areas important to building designers. These included specifying connection points and component types that are carried over to the BIM-designer's software. In addition, the mechanical designer using SolidWorks had a more difficult time orientating the model and simplifying a non-native model for export.

Our test of CATIA interoperability and direct modeling on imported models reiterated the widely known issue that SolidWorks does not directly import a native CATIA V5 file, even though both products are part of the same company. Direct modeling was comparable for both Inventor and SolidWorks, with SolidWorks being a little easier to use for the simple direct model changes we made. The SolidWorks drawing output in DWG format produced an incorrect dimension in a scaled view.

For design automation, our tests revealed two weaknesses of SolidWorks. SolidWorks with DriveWorksXpress was not able to automatically scale drawing views to fit a part within the confines of a drawing after the size of the part was changed. Manual intervention was necessary. A second weakness was shown when scaling a copied assembly using 3D curves to define key points as the assembly was copied and scaled to other planes. Inventor was easily able to scale a copied assembly using drive curves; SolidWorks could, but required significant manual effort.

Both systems proved to be comparable in mechatronics where we tested the ability to build wire harnesses using schematic input from electrical software packages, albeit Inventor was able to do so with many fewer interactions.

Conclusions

Ray Kurland, President of TechniCom knew that the tests were meant to highlight Inventor strengths, but was surprised that SolidWorks Premium 2011 was, in many tests, not able to do the work without adding pricey third party software. Duplicating Inventor's capability on these tests with third party products will also make SolidWorks substantially more expensive than Inventor.

These seven tests underscore our contention from our previous Delphi Expert Analysis, that Inventor is a mature system that can more than effectively compete with SolidWorks and should definitely be considered for even the most complex situations.

The Inventor workflows illustrated in this series of tests are integrated and highly logical, enabling users to accomplish their design goals with minimal effort. Beyond that, we hope to have shown the value of Autodesk's digital prototyping emphasis, which we expect will continue to evolve even further.

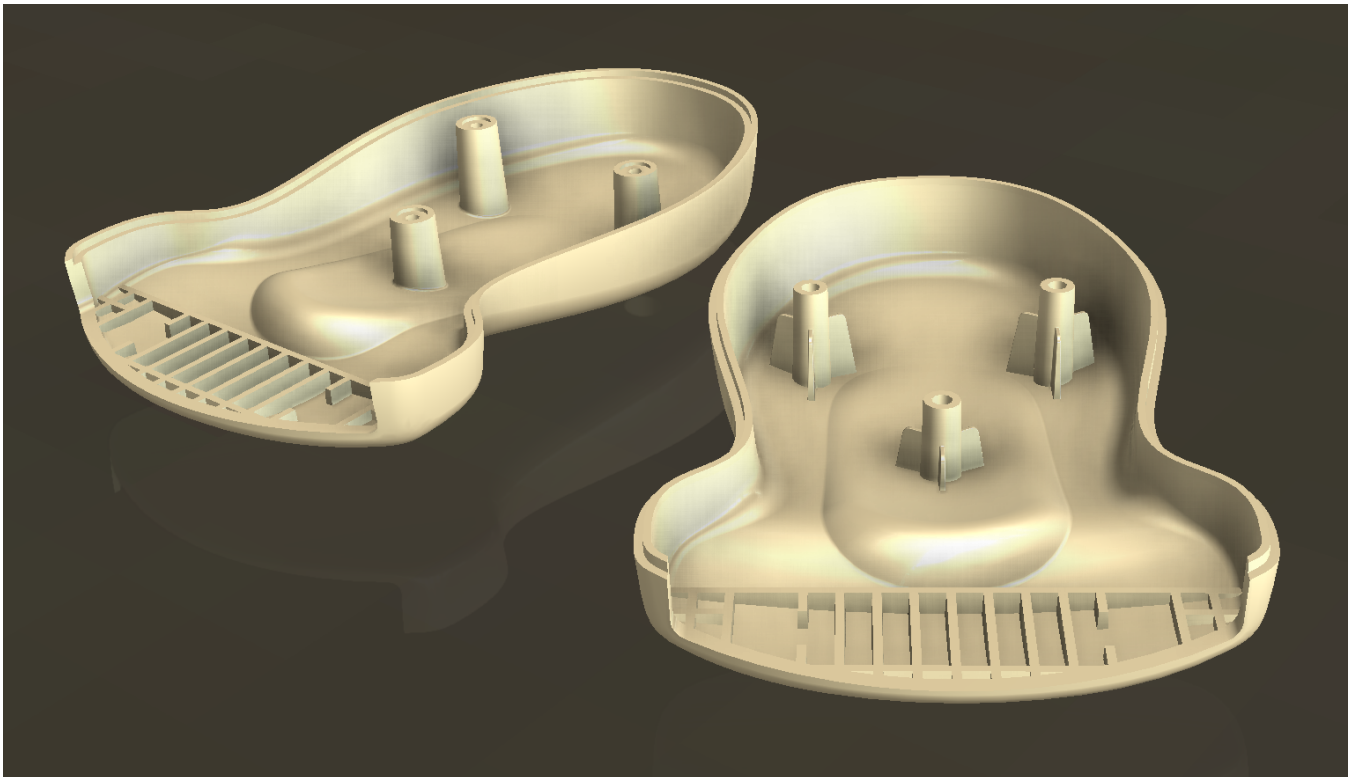
"I didn't know that Inventor had this much functionality," said TechniCom Group's associate performing the tests, a CSWP. "I know that they acquired a lot of technology over the past few years, but I am surprised to see it all integrated so well into Inventor."

Overall, TechniCom is most impressed with Inventor and the direction Autodesk is taking for the future. To keep abreast with our continued tracking of the industry and our reactions to Autodesk's direction we advise readers to follow our blog and twitter feeds.

Plastic Part Design

This tests the ability to import surface data from Alias, stitch the skins into a solid body, design the shell the part with a specified wall thickness, part using surfacing and plastic features, import new surfaces and update the model, and perform an injection molding simulation on the part.

Autodesk provided us with data files specifying the solid model, IGES and WIRE data files of the surfaces, and three movies depicting the workflow for surface import, engineering design, and simulation and validation.



A view of the final parts.

What's Important in Plastic Part Design

- Rapid design, ready for manufacturing
- Working with surfaces from industrial design software
- Ability to directly create mold ready parts, typically for injection molding
- Evaluating the moldability of the part

Observations

Surface Import

Inventor was able to import the Alias wire file natively without issue. SolidWorks was unable to import the Alias wire file. Users must first translate Alias data to an IGES file, which is susceptible to translation errors, albeit not in this case.

Build the Model

Creating a solid model from imported surfaces and being able to shell the resulting solid are typically the most error-prone steps in the process. Inventor was able to stitch and shell the part with zero errors. The shell was created in one step by defining the variable in the shell dialog box.

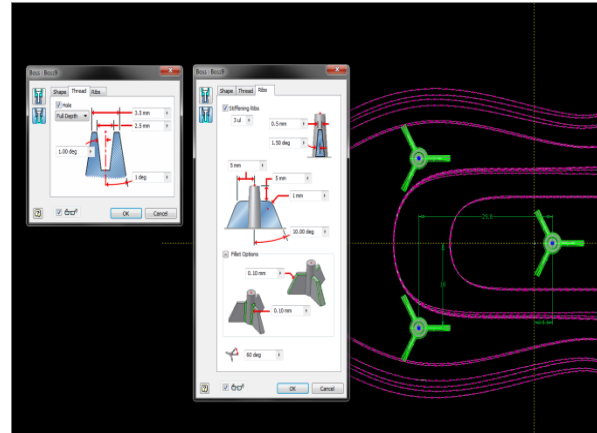
SolidWorks was also able to stitch and shell the part with zero errors. Shell creation involved several steps to create and define the variable.

Plastic parts are typically designed using a set of standard features such as ribs, bosses, grilles, snap-fits, and lips to name just a few. The MCAD software should assist the user in efficiently modeling these features. Inventor used its plastic features toolset to add the two different types of bosses, a lip feature, and ribs. SolidWorks used plastic features for a majority of the features, although the recessed bosses required for this test first needed to be built manually and then added from the library of custom user features.

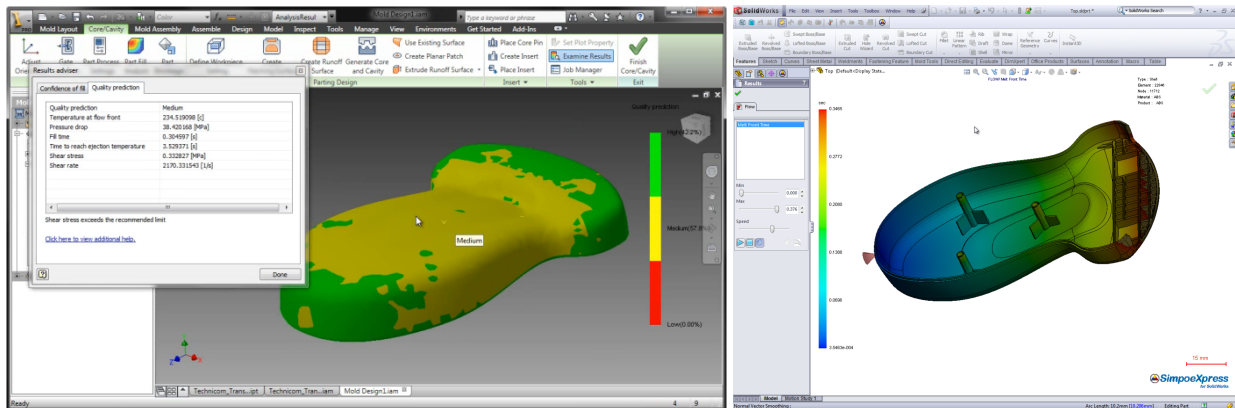
Simulate and Validate the Part Design

Plastic parts must be checked for potential quality defects prior to committing to the cost of designing and building the mold.

In the simulation to evaluate the manufactured quality of the product as-designed, Inventor simulated the injection molding process and uncovered high amounts of shear stress due to the part being too thin. If left uncorrected, this issue would lead to material degradation and molding defects or field failure. Inventor's built-in mold analysis software also provided more extensive capabilities in terms of material selection and multiple gate analysis.



Specifications of the bosses.



Inventor & SolidWorks feedback about potential quality issues.

There are no built-in simulation capabilities within SolidWorks for evaluating the manufactured quality of the product as designed. However, we were able to use a third party add-in module called SimpoeXpress. This has limited function, but allows for some material selection and a single gate. SolidWorks was able to simulate the molding process but the only result the user received was the filling pattern, which provided limited value. It was unable to identify any quality defects and the user was misled into thinking the design was acceptable. More comprehensive simulation packages are available at a cost of more than \$5000.

After modifying the 3D CAD model, we reran the simulation to validate the design change. After making the recommended change to the part, Inventor automatically updated the model in the simulation environment; all that was required was to re-run the analysis. SolidWorks automatically updated the geometry but the analysis had to be setup from scratch, including processing parameters, gate location, and material selection.

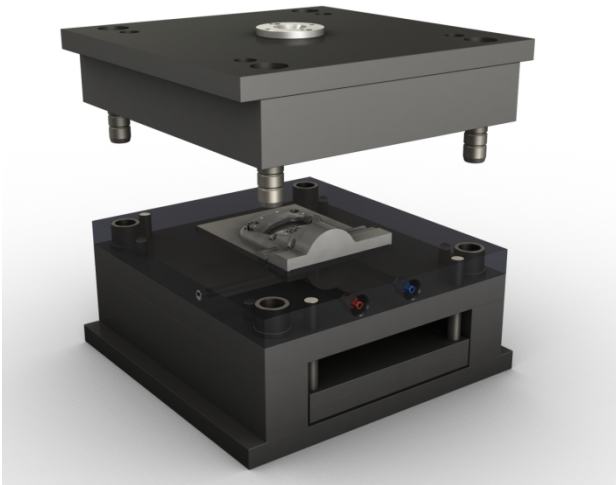
Summary

Importing the IGES files and creating the plastic part was comparable for both products. While SolidWorks was able to import IGES curves from industrial design software, Inventor was able to directly read Alias (a leading industrial design software package) surface data, an advantage. Both products had excellent capabilities for building specialized plastic features such as the mounting boss and the lip and groove on the connecting halves of the model.

Inventor's built-in analysis software powered by Moldflow provides impressive analysis capabilities, and is well integrated into part design. SolidWorks, on the other hand, required a no-charge third party product that was able to perform only a limited analysis of the part. The limited function mold analysis software that was free does not provide the engineer enough insight to be confident that parts can be manufactured at all or at acceptable quality. More complex mold analysis software with advanced analysis capabilities (which was not used) is available to SolidWorks users at considerable extra cost.

Videos of users of both systems performing these tests can be seen on [Ray Kurland's blog, Part 2](#), at <http://wp.me/pvn8U-4H>.

Plastic Injection Mold Design



Simple mold design used in this test.

In this section, we test the ability to use the 3D model of the plastic part to create the core and cavity of the mold, design and engineer the multiple components and systems of the mold, and validate the design to ensure it can manufacture high-quality plastic parts.

Autodesk provided us with a model of the handle to be molded, detailed specifications for the mold, and three movies of Inventor performing the desired tests showing the workflow for splitting core and cavity, engineering of the mold, and a simulation and validation of the mold.

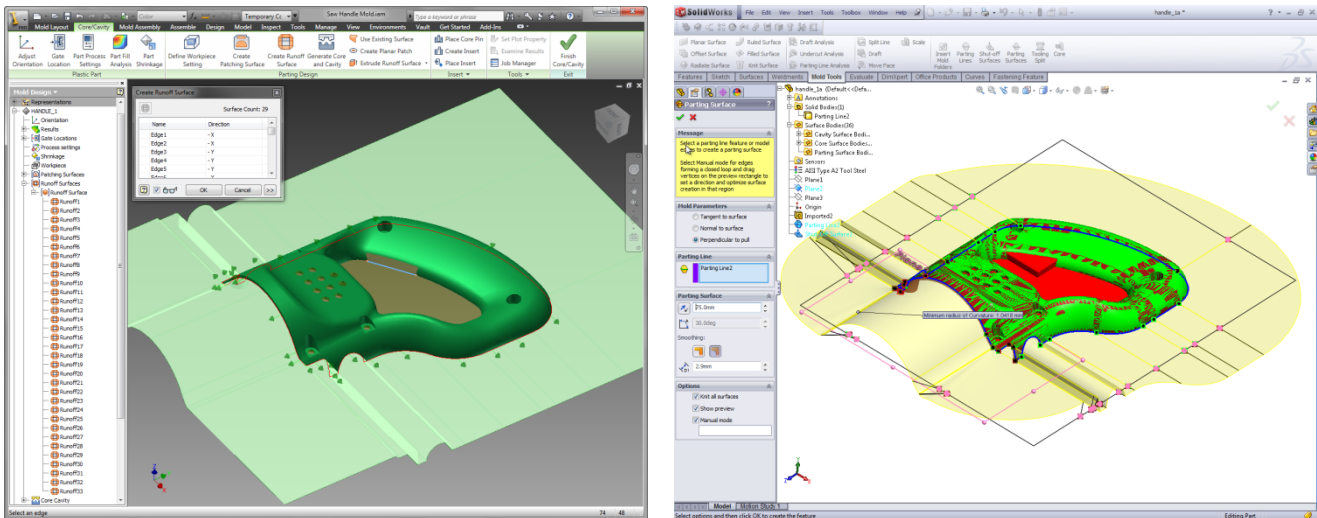
What's Important in Plastic Injection Mold Design

- Balance of speed in designing the mold while ensuring high quality.
- Accurate design of mold components including runners for injecting the plastic materials, cooling of the mold, and ejecting the finished part.
- Iteration of the mold design with simulation to arrive at an optimal design.

Observations

Splitting the Core and Cavity

The desired result was to generate parting surfaces and complete the core and cavity operations.



Inventor & SolidWorks parting surface generation.

Inventor used a mixture of automated and manual patching and runoff surface creation tools. Surfaces for simple holes and profiles were created automatically which increases productivity. Complex patching and runoffs were created using Inventor's surfacing tools.

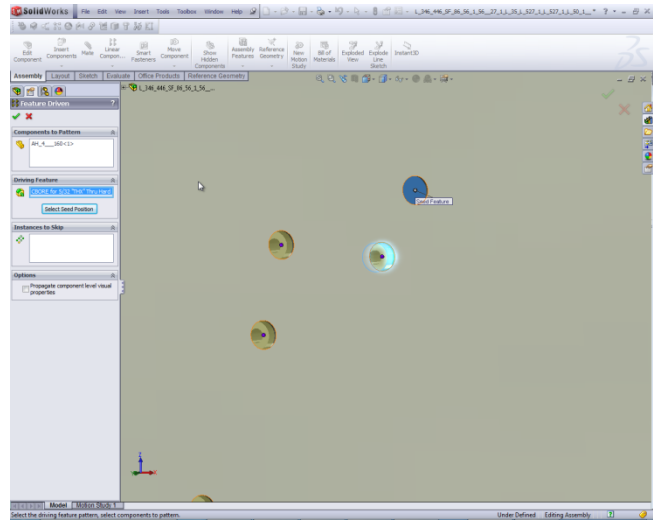
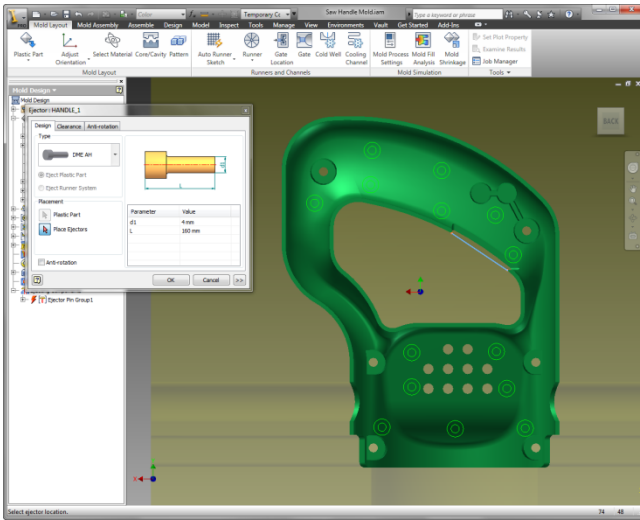
SolidWorks also assisted the user in splitting the core and cavity with automated and manual tools for defining the parting line and creating patching and runoff surfaces.

The two systems are comparable in capability. SolidWorks required a few more menu picks and interactions, but both came up with an acceptable mold core and cavity. SolidWorks generated an odd triangular shape in the area to be removed, but it was temporary and did not affect the final part.

Engineering the Mold

The tasks completed included: designing the runners, adding a submarine gate, inserting a properly sized mold base, inserting a sprue bushing, designing cooling channels, attaching pipe fittings for cooling channels, and adding ejector pins as specified.

Inventor completed this task using a built-in workflow for designing injection molds that includes libraries of mold bases and standard components as well as automated design tools for runners, gates, cooling channels, slides, lifters, and ejectors.



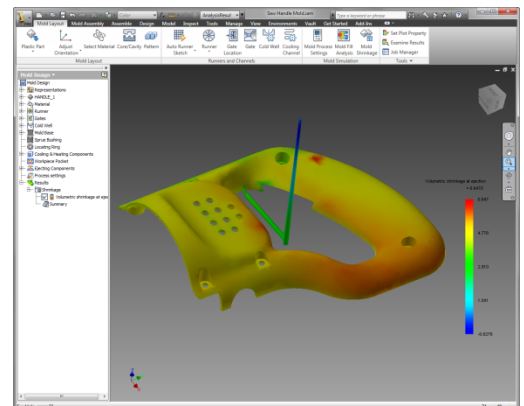
Automated ejector placement in Inventor (left) vs. manual placement in SolidWorks (right).

SolidWorks had no built-in functionality for designing injection molds. All standard components needed to be searched for and brought in from external content centers or supplier websites, a time-consuming process. All modeling was done manually as there are no automated design tools for the various systems of the mold. This made mold design in SolidWorks a tedious and labor-intensive process with low user productivity. SolidWorks was able to build the geometry required for the moldbase design, but it was a laborious process.

Validating the Mold Design

To validate the mold design for manufacturability we needed to first determine the optimal molding conditions for the entire system as designed. Next, we performed a filling analysis to determine if the mold, as designed, could completely fill the cavity at acceptable quality. Then, we assessed the location of air traps and weld lines. Lastly, we performed a shrinkage analysis so exact figures could be input for core and cavity sizing rather than manually inputting generic percentages.

Inventor includes Autodesk Moldflow simulation built-in to the mold design workflow, which was used to simulate the filling phase of multi-cavity molds and their respective runner systems to validate manufacturability. A shrinkage analysis



Shrinkage result in Inventor.

was also used to ensure cavities were sized based on the specific design, rather than relying on generic shrinkage factors from the material supplier.

SolidWorks has no built-in simulation capabilities and therefore had no ability to validate the mold design for optimum molding conditions or for a filling analysis that would help the user avoid manufacturing problems that can potentially result in huge expenses in both time and cost. SolidWorks was unable to perform any portion of this validation using its software or no-charge software, of which there was none available.

Users could use Autodesk Moldflow stand-alone software that would have the capabilities required, at extra cost. *Our analysis did not include examining extra cost third party products. We only compared Inventor Professional to SolidWorks Premium.*

Summary

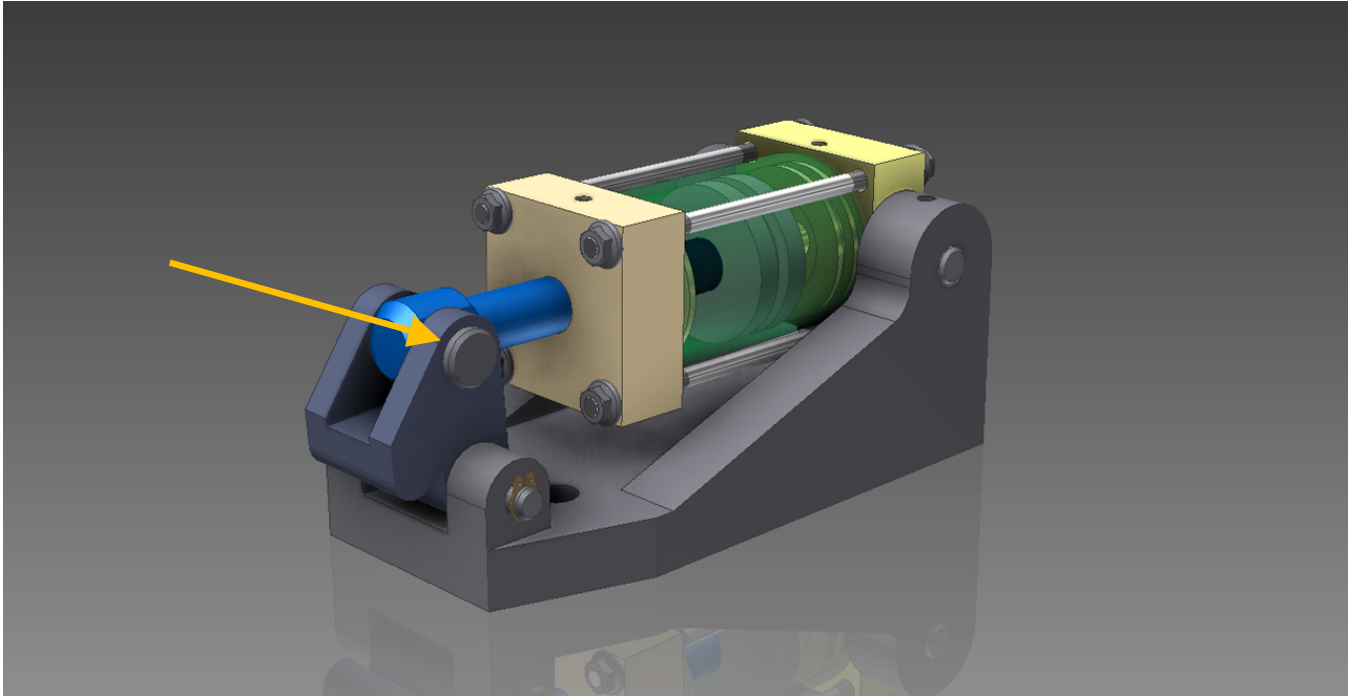
Autodesk Inventor provides standard libraries of mold bases and components along with automated tools for splitting the core and cavity and for designing the runners, gates, and cooling and ejection systems. The inclusion of Autodesk Moldflow simulation software directly in the design workflow allows designs to be validated and improved upon until they will optimally manufacture products of the highest quality.

SolidWorks includes dedicated functionality for splitting the core and cavity, but that is where the mold design capabilities end. With no automated design tools and no libraries of components, the design of injection molds is entirely manual and inefficient. Without any built-in plastics simulation capabilities, mold designers must purchase 3rd party software, such as Autodesk Moldflow, to validate and optimize their designs to ensure quality.

Videos of users of both systems performing these tests can be seen on [Ray Kurland's blog, Part 3](http://wp.me/pvn8U-5f) at <http://wp.me/pvn8U-5f>.

Assembly Design and Analysis

This test focuses on the design of a clevis shear pin and its related holes in this hydraulic clamping assembly. The pin needs to be optimized so that if a failure occurs, the pin fails and not the other components. We'll seek a factor of safety of 2 for the clevis pin, and 4 for the rest of the assembly. The Clevis Pin shear pin must withstand 250 N force with a factor of safety of 2. The bending force on the pin is important. If it exceeds maximum allowance the pin cannot be removed. The pin should be sized to meet the forces at the designated safety factor and fit within the support structure. The pin should further be selected from a standard library of components and created with the size required. The design calculations should be stored for documentation.



Final assembly, arrow points to clevis pin to be designed.

Autodesk provided a Parasolid model of the above assembly and a movie of Inventor performing the design and analysis to select the proper pin to fit within the clevis opening.

What's Important in Assembly Design and Analysis

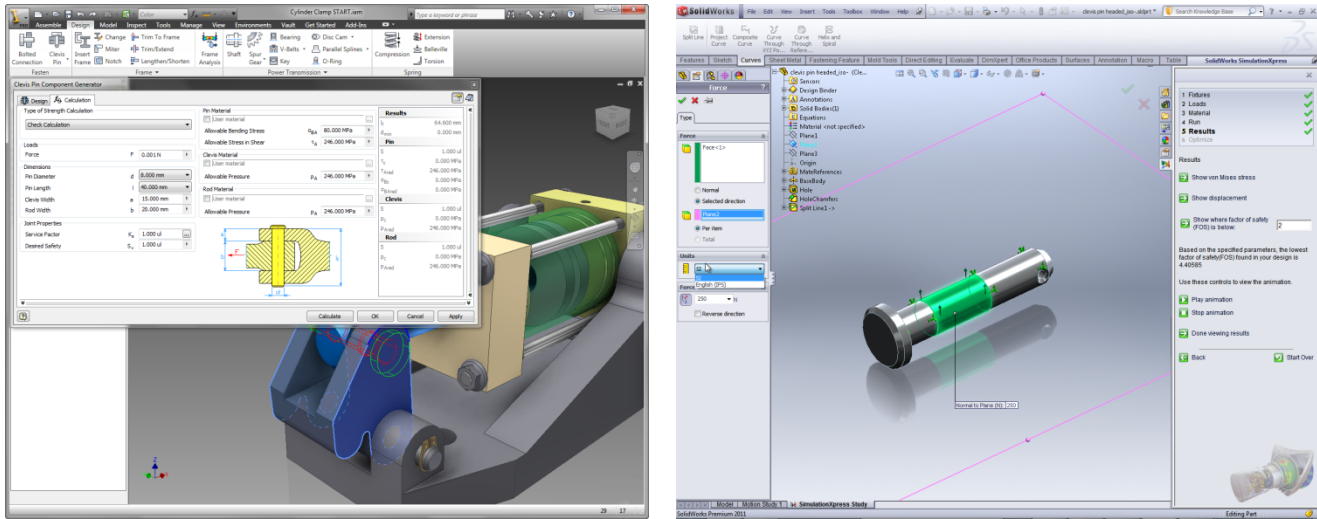
- Select the most appropriate purchasable clevis pin that meets the specifications
- Weigh design decisions that affect cost, product reliability, and weight
- Evaluate the performance of the final design

Observations

Inventor executes all of the steps (make the hole, insert the pin, perform the engineering calculation) required within one command (feature). SolidWorks uses separate commands for each of the three key steps. Both Inventor and SolidWorks seem to have an equally robust clevis pin library, and automatic sizing capability.

The engineering calculation is the large differentiator in the example. Both Inventor and SolidWorks offer integrated finite-element analysis (FEA). However, using FEA methods for this kind of problem is a questionable strategy. Autodesk prompted the user to determine the correct pin size by using its engineering calculations. SolidWorks does not provide this kind of functionality. SolidWorks' concept was to have the designer select the pin size and then perform an FEA analysis in an

iterative fashion to arrive at the correct sizing. SolidWorks was able to use its own no-charge Simulation Xpress to perform the FEA analysis. SolidWorks Simulation Xpress is a first-pass basic stress analysis tool that comes with every SolidWorks Standard and Professional software package, offering limited FEA functionality.



Design Accelerator in Inventor (left) and SimulationXpress environment in SolidWorks (right).

Standard engineering calculations (analytical methods) exist for this situation, making FEA methods unnecessary/overkill. The FEA boundary conditions necessary for this case present a convergence issue (singularity) for most solvers. This increases the expertise required to verify the accuracy of this FEA study with any level of certainty. Some FEA programs will simply never reach a reliable result for this case. This is typically referred to as a divergent case. SolidWorks Simulation Xpress did not allow us to individually manipulate the mesh to test for convergence.

Summary

Both Autodesk Inventor and SolidWorks can solve this problem; however there are important differences in the steps required to complete the exercise and the confidence in the engineering optimization.

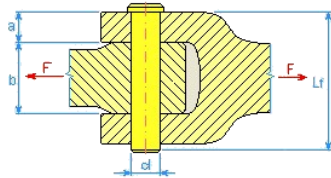
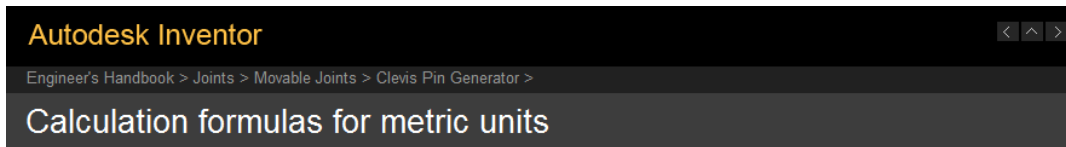
Inventor executed all of the steps within one command sequence. SolidWorks uses separate commands for each of the three key steps. Both Inventor and SolidWorks seem to have an equally robust clevis pin library, and automatic sizing capability.

The engineering calculation is the differentiator in the example. For this problem Inventor designed a solution using its engineering calculations library for clevis pins. The only solution offered by SolidWorks was to use FEA, which in this case, proved to be difficult to verify and is a questionable strategy.

Inventor users have the ability to leverage standard engineering calculations shown below which are included in design accelerator tools. In this test we examine the clevis pin generator. The same concept applies to bolts, bolts, frames, shafts, gears, bearings, belts, chains, keyways, cams, splines, o-rings, and springs. Inventor does not require the user to know or learn the engineering equations; the software does it for you. In the case of using FEA, the engineer must be concerned with the accuracy (error) inherent to FEA methods, necessitating a higher skill set, and certainly more time. Inventor’s automation of standard engineering calculations provided a better solution and reduced design time.

Videos of users of both systems performing these tests can be seen on [Ray Kurland’s blog, Part 4](http://wp.me/pvn8U-5P) at <http://wp.me/pvn8U-5P>.

Below are the engineering equations Inventor uses for its clevis pin calculation.



Bending stress

$$\sigma_o = \frac{M_o}{W_o} = \frac{F(b+2a)}{\frac{\pi \cdot d^3}{32}} = \frac{4F(b+2a)}{\pi \cdot d^3} \leq \sigma_{oD} \quad [MPa]$$

Pressure in rod

$$p_1 = \frac{F}{d \cdot b} \leq p_{1D} \quad [MPa]$$

Pressure in clevis

$$p_2 = \frac{F}{2d \cdot a} \leq p_{2D} \quad [MPa]$$

Shear stress

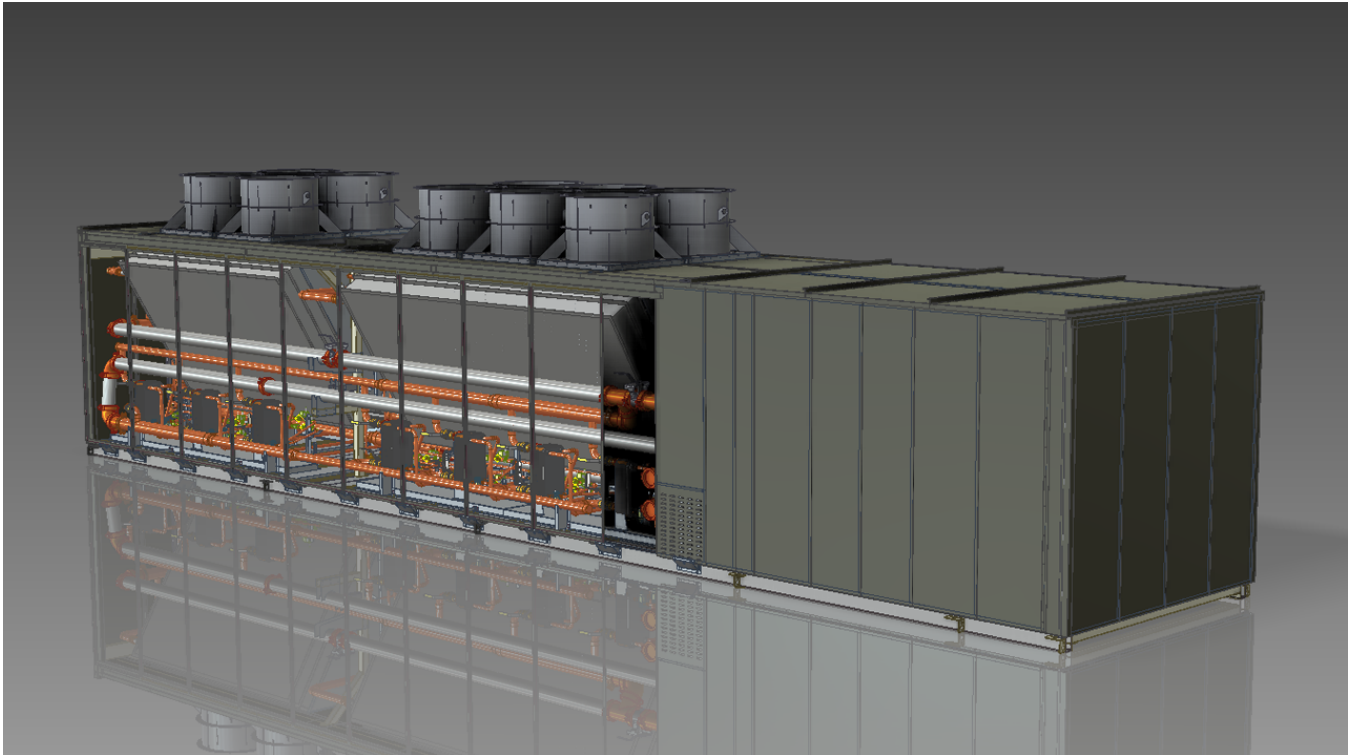
$$\tau = \frac{2F}{\pi \cdot d^2} \leq \tau_D \quad [MPa]$$

where:

- F force [N]
- σ_o bending stress [MPa]
- σ_{oD} allowable bending stress [MPa]
- M_o bending moment [N mm]
- W_o bending section modulus [mm³]
- τ shear stress [MPa]
- τ_D allowable shear stress [MPa]
- d pin diameter [mm]
- p_1 pressure in rod [MPa]
- p_2 pressure in clevis [MPa]
- p_{1D} allowable pressure in rod [MPa]
- p_{2D} allowable pressure in clevis [MPa]
- b rod width [mm]
- a clevis arm width [mm]

Exporting BIM-Ready Models

In this test, we started with a complex assembly that had already been designed in the mechanical system. The goal was to export a simplified version of the assembly for inclusion in Autodesk Revit software (BIM software). The software should allow the user to reduce the level of detail in the exported file to protect proprietary design information, to indicate to the BIM software the category (window, door, HVAC, etc.) of the exported data, to provide types and locations for plumbing and electrical connections, and to be able to control the orientation of the exported part or assembly.



View of the chiller unit used in this exercise.

Autodesk provided a video of the chiller and the workflow performing the desired steps. Also provided was an IGES file of the chiller with all the model details.

What's Important in Exporting BIM-Ready Models

- Provide lightweight BIM-ready models which can be directly incorporated into the building design process
- Mechanical models should include unique architectural and construction information

Observations

Model Preparation

The first step was to simplify the model by suppressing proprietary and unnecessary parts from the assembly and removing details such as small holes, grilles, etc. This eliminates exposing intellectual property and also reduces the level of detail, avoiding large file sizes.

Inventor was able to suppress all unnecessary components easily. A shrinkwrap feature simplified the assembly with native and non-native data by removing small features and patching small holes, reducing the file size.

SolidWorks was able to suppress unnecessary components without issue. Because this was imported data, SolidWorks was unable to further simplify the designs, although had it been native data it may have been possible. Readers should note that the Inventor data, however, was native and thus a somewhat unfair comparison.

Orientation for Import

The second step was to orient the part properly for use in BIM software. Ensuring that models come in with the correct orientation removes the frustrating process of reorienting every time the product is inserted into a design.

Inventor allows the user to create and assign a custom local coordinate system that can be specified on export thus eliminating this issue.

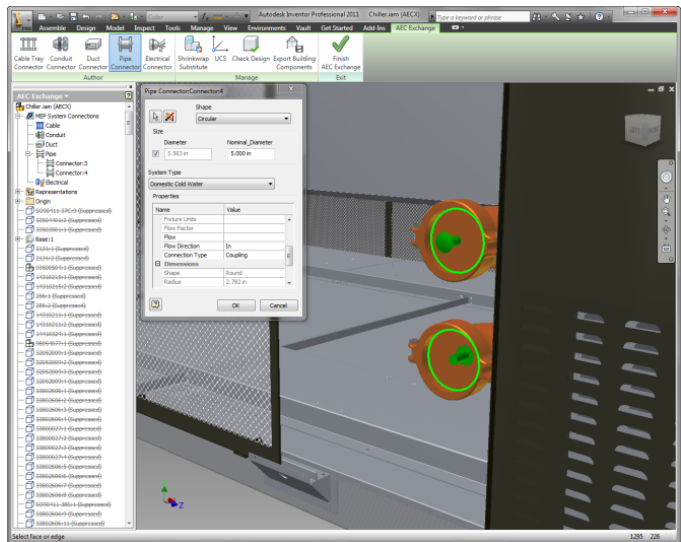
SolidWorks can create a custom user coordinate system (UCS), but cannot use it when exporting IFC files (a BIM standard file format). A new assembly needs to be created with the product placed in the correct orientation. This workaround requires additional time to create the new assembly and creates additional data to manage.

Define Connection Points

The third step was to define connection points to the assembly with sizing and connection attributes. Mechanical, electrical, and plumbing (MEP) engineers need to know the location, size, and type of connections required for the product when designing piping or wiring for the building.

Inventor allows the user to specify the location of connections along with information about pipe size, wiring, connection method, system type (i.e. hot water/cold water/120v/240v/etc.), and flow direction.

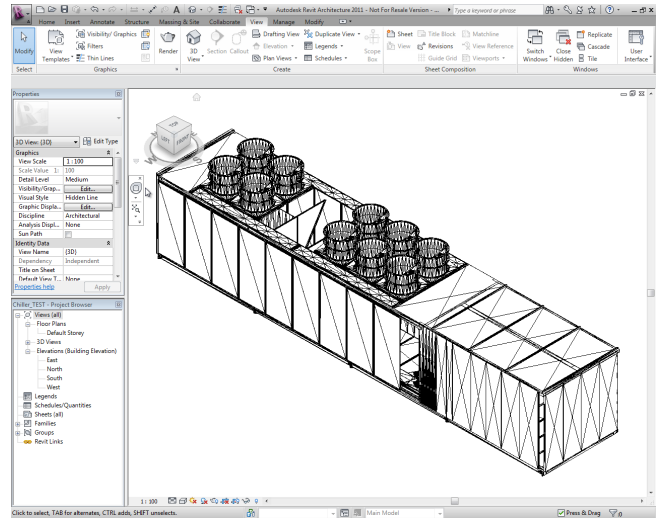
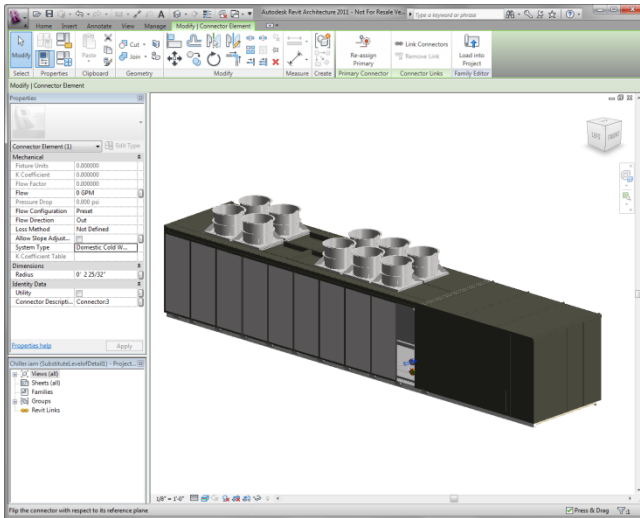
SolidWorks is unable to assign connection properties, instead requiring the data to be manually communicated, a minimally acceptable alternative.



MEP connection definition in Inventor.

Data Export

The fourth step was to export the assembly to a file format that can be read into BIM software with the necessary attributes assigned. Inventor exports file formats that can be directly read into Revit or AutoCAD and can be included in Building Information Models. Component types assigned in Inventor were carried over so no additional categorization was required. Additional properties are carried over, such as weight, size, and appearance.



Chiller inside Autodesk Revit (Inventor export on left, SolidWorks export on right)

SolidWorks exports IFC 2x3 neutral format files that can also be directly read into Revit or AutoCAD. Component type can be assigned on export and are carried over into Revit. BIM designers are required to manually assign additional properties.

Summary

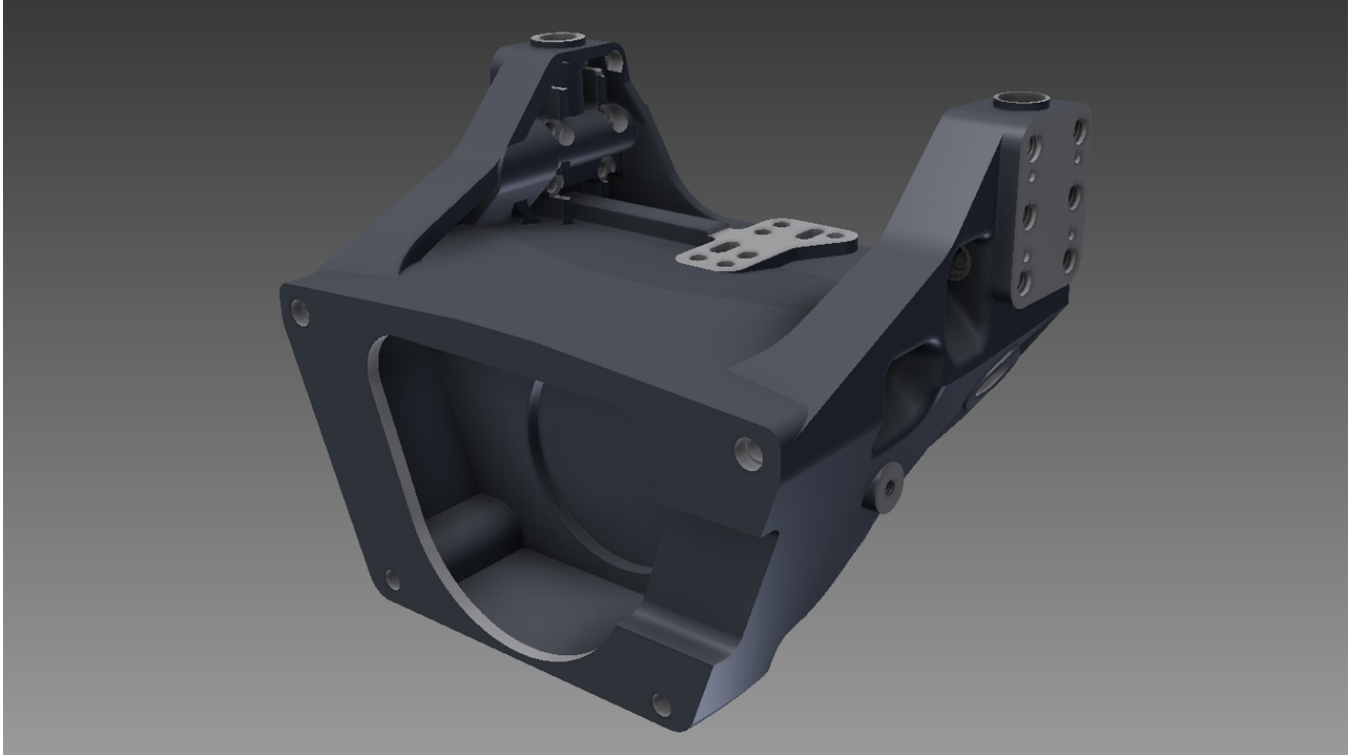
Autodesk Inventor has a dedicated workflow that communicates a lightweight version of the geometry with critical information like connection points and component types while maintaining all physical and visual aspects of the design in a file format that can be easily handled by the most common BIM applications.

SolidWorks does not have a dedicated workflow. SolidWorks 2011 added the ability to exchange simplified geometry data, but with limited amounts of component information due to the use of a neutral file format. It was not able to provide connection points or carry physical and visual properties over.

Videos of users of both systems performing these tests can be seen on [Ray Kurland's blog, Part 5](http://wp.me/pvn8U-61) at <http://wp.me/pvn8U-61>.

Interoperability and Direct Modeling

To examine interoperability, we tested the capabilities of the software by importing a CATIA part, modifying the imported part, and creating and validating the accuracy of a DWG drawing of the part for communication with vendors.



View of the bell housing used in this exercise.

Autodesk provided a video of Inventor being put through this exercise, the bell housing in CATIA format and the bell housing in IGES format.

What's Important in Interoperability

- Directly reading the other systems data directly - in this case CATIA V5- rather than performing a multi-step and potentially error prone process of intermediate data conversion.
- Easily share design data with customers, vendors, suppliers, and other departments using different CAD systems.
- Reading and writing DWG files for production, and publishing designs in formats that customers can use in their own applications.

Observations

Importing CATIA Part

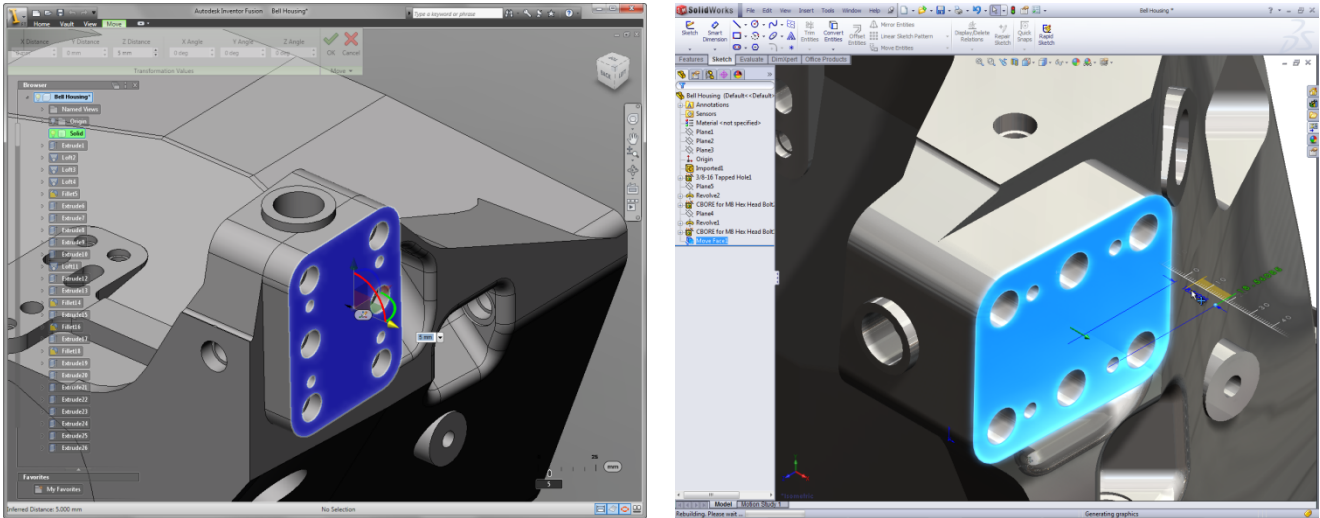
The desired result of this test was to import a CATIA V5 model into the software.

Autodesk Inventor read the CATIA data directly and was able to open the model with no issue.

SolidWorks was unable to read the native CATIA V5 model. Third party products are available that offer direct CATIA – SolidWorks translation. As a workaround for this workflow test, the IGES file was used in later tests and successfully imported into SolidWorks. The need to convert each CATIA V5 file to the IGES format may be a major issue for automotive and aerospace suppliers and OEMs since there are many companies involved that require data in native CATIA format!

Modifying Imported Geometry

Next, the ability of each software to make small modifications to the “dumb” (history free) solid created from the imported file was examined.



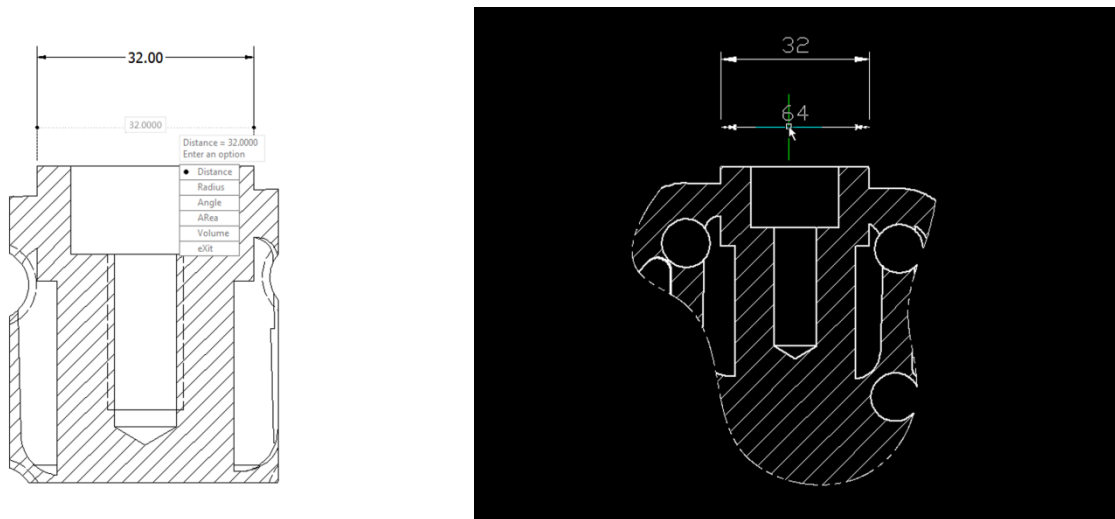
Modifying the geometry in Inventor Fusion (left) and SolidWorks (right).

In the video, Inventor made the necessary modifications using the free Inventor Fusion Technology Preview labs application. The changes were made successfully and then Change Manager was used to update the dumb solid in Inventor.

SolidWorks had no problem with the direct modification of the imported part. Feature recognition capabilities were used to modify the plates and the holes as required. In this case it was easier than Inventor, which required back and forth interaction with Inventor Fusion.

Creating DWG Drawing

Next, each software system was used to create a drawing in DWG format, make a change to the 3D model and update the DWG.



Measuring the resultant DWG’s created by Inventor (left) and SolidWorks (right). Note the incorrectly scaled dimension in the SolidWorks created DWG.

Inventor created the drawing in DWG format - no translation was required. The file was opened in AutoCAD and presented exactly as it had been in Inventor. After making a change to the 3D model in Inventor, the DWG version of the drawing updated automatically.

SolidWorks is able to create DWG files for export to vendors. However, these DWG files lack the ability to be fully associative with the SolidWorks 3D model. This means an additional change to the SolidWorks 3D model requires that a new DWG file be created. The exported SolidWorks DWG was opened in both AutoCAD and Dassault DraftSight. In both AutoCAD and Dassault DraftSight, the SolidWorks DWG produced a dimension that showed as 64mm instead of the correct 32mm.

Summary

Autodesk Inventor easily imported the CATIA V5 file. Working with imported data requires the direct modeling tools found in Inventor Fusion Technology Preview to make changes. Creating a fully associative drawing in DWG format using Inventor requires no additional effort since Inventor uses native DWG as the file type for drawings created from the 3D model.

SolidWorks is also able to import IGES files and export a DWG drawing format but has no support for CATIA V5 files, which must be translated into a neutral file format like IGES before use. File translations introduce complexity and the potential for errors. Once called up in SolidWorks, the program has tools for modifying non-native 3D geometry with several functions like feature recognition and move face. Last, in SolidWorks, DWG drawings are not associative to the 3D model and may require a significant amount of time and effort to create new DWG for each change of the 3D SolidWorks model. More time may also be needed to clean up any DWG drawing export errors that occur prior to sending them to customers and vendors. Though the SolidWorks DWG associativity did not work in this test on TechniCom's version of SolidWorks 2011, SolidWorks has supported this capability in past releases and it may work in other installations.

Videos of users of both systems performing these tests can be seen on [Ray Kurland's blog, Part 6](http://wp.me/pvn8U-6l) at <http://wp.me/pvn8U-6l>.

Design Automation

This test looks at a simplified automation examples, yet in the time allotted provides a glimpse of this capability in both products.

- Create a simple piece of stock lumber (2x4 board) and examine how a user can make that same part file represent several variations of lumber that could be used in a project.
- Automate the variation of individual drawing views, scales, and annotations.
- Automate assembly variations that vary by size and position.

Autodesk provided us with three movies showing Inventor completing the tasks. They also provided three STEP files of the frame, the assembly, and the curves to follow for the frame assembly resizing.

What's Important for Design Automation

- Engineers can capture design intelligence by using rules to embed design intelligence into parts, assemblies, and even drawings
- Such design intelligence, in the case where repetitive designs or portions of repetitive designs are used, can radically reduce design time and produce more repeatable results.
- What techniques are used to build the design intelligence (often programmatic)
- How easy is it used to create new designs once the rules have been built
- How such design intelligence is accessible and how it can be maintained in the future

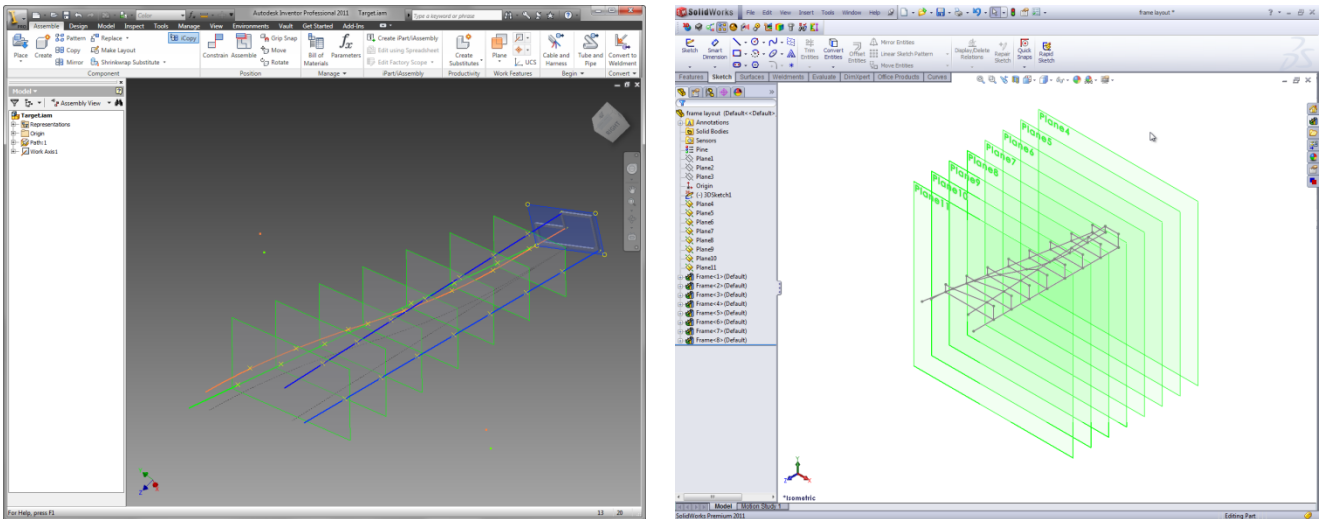
Observations

Autodesk Inventor includes iLogic and iCopy technologies that use rules to control the parameters of the part, assembly, or drawing. Inventor controls the parameters of a part through a single dialog box that updates the model on the fly. It is also able to automate the process of creating unique assembly configurations by modifying the parts and sub-assemblies automatically for the user. iLogic also can use rules to drive drawings - from view placement to scales and annotations - for a family of parts or assemblies, which can save significant amounts of time in large scale projects.

SolidWorks has some of the same capabilities built into the modeler. It can handle configuring a part when placed into an assembly, but updating it in the part model on the fly is not possible. It was also able to create the multiple configurations of an assembly – although it required more steps. Without using extra cost third party software, SolidWorks is unable to automate the creation of drawings for part families, which requires users to create a drawing for each instance of the family.

For the stock lumber workflow, both Inventor and SolidWorks were able to capture all the variations within a single part file.

A family of parts or assemblies also requires a family of drawings to document their design intent. Recreating essentially the same drawing, which only varies by a few critical dimensions wastes time and effort. Inventor allows the user to easily automate drawings using iLogic functionality. Inventor drawings can be set up to automatically vary view placement, scale, and annotations for a family of parts or assemblies. SolidWorks, without extra cost third party software, is unable to automate the creation of drawings for part families.



Creating copies of a frame along a path using Inventor (left) and SolidWorks (right).

For the frame variation example, Inventor allows the user to automate complete assemblies that vary along specified paths. SolidWorks was able to manually model the frames along a path, but took substantially longer.

Summary

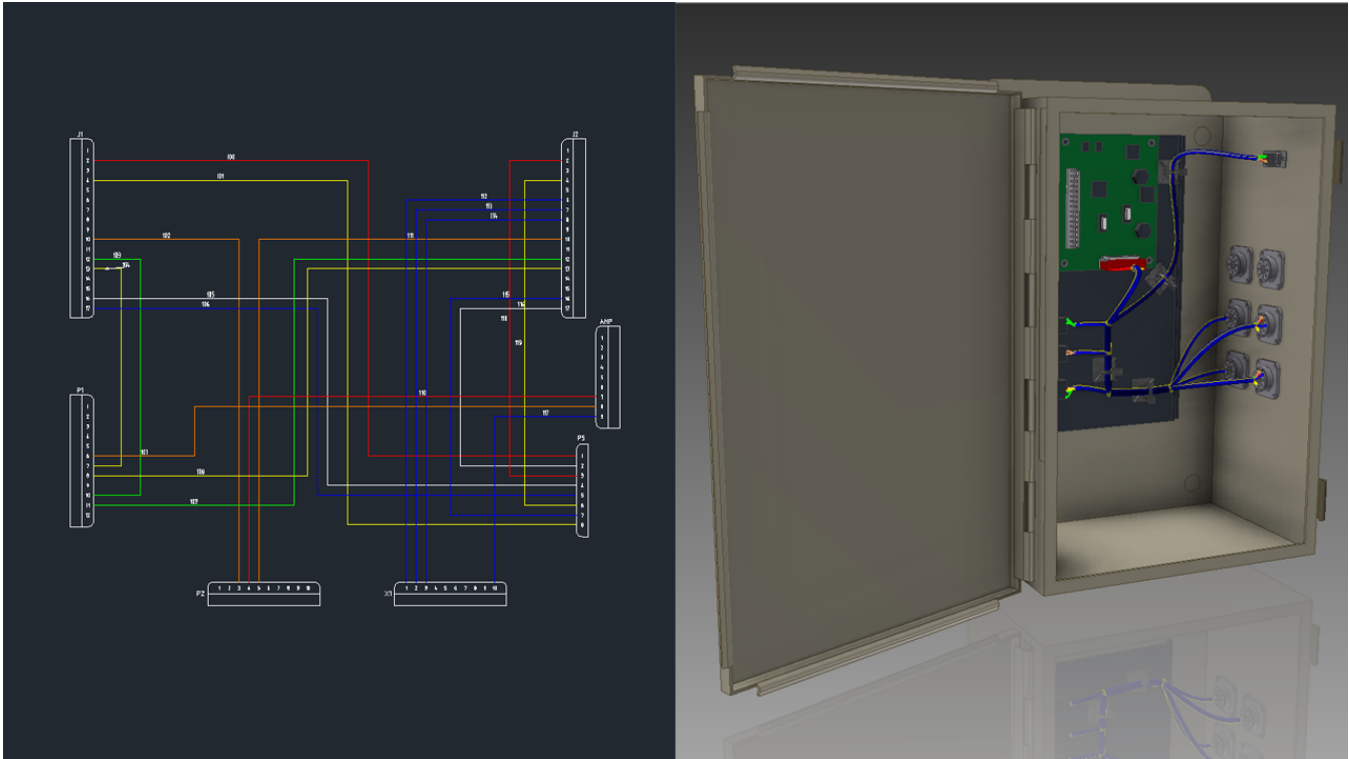
SolidWorks has some of the same capabilities built into the core modeling system and by using DriveWorksXpress, a 3rd party add-in delivered with SolidWorks. It can handle configuring a part when placed into an assembly, but updating it in the part model on the fly is not possible. It was able to create the multiple configurations of an assembly – although it required more steps than Inventor to achieve the same solution. SolidWorks Premium was unable to automate the creation of drawings for part families, which requires users to go through the manual process of creating a drawing for each instance of the family. SolidWorks' third party partner, DriveWorks, offers software that can perform this process, although at additional cost. The no-charge version was not able to control the final drawings, as desired. We did not evaluate DriveWorks, although the extra cost versions of DriveWorks Solo and DriveWorks Pro appear able to perform this task, again, at added cost.

Autodesk Inventor includes iLogic and iCopy technologies that use rules to control the parameters of the part, assembly, or drawing and these capabilities were used to complete this test. Inventor created the lumber workflow, the frame resizing and the drawing scaling without flaws.

Videos of users of both systems performing these tests can be seen on [Ray Kurland's blog, Part 7](http://wp.me/pvn8U-6D) at <http://wp.me/pvn8U-6D>.

Mechatronics

This tests the ability of the mechanical CAD system (MCAD) to leverage data from an electrical CAD system (ECAD). The ECAD system specifies the appropriate connectors, wires, and their connection points while the MCAD system specifies the physical location of those wires and connectors within a product.



Electrical schematic and assembly used in this example.

Autodesk supplied us with an Inventor video of their solution, a net list in Excel format, a STEP file of the enclosure assembly, and a schematic drawing (.dwg) of the connections.

What's Important in Mechatronics Design

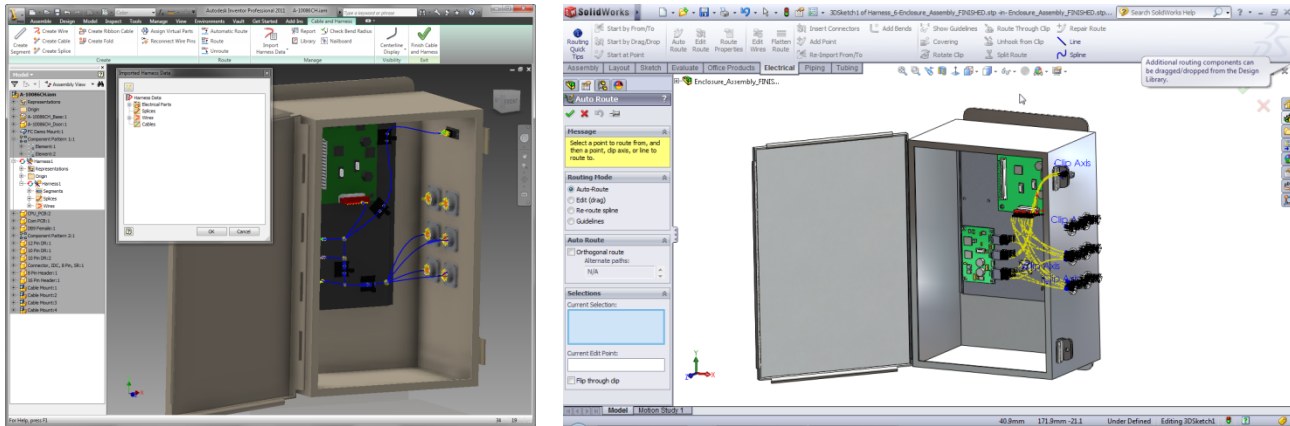
- Leverage the data stored in schematic drawing files to design wire harnesses in the mechanical system. Such data can be stored exported from an electrical design file using various techniques. At its most basic, the electrical design software sends a net list to the mechanical package containing connector information for each wire, wire types, and a list of pin-to-pin connections.
- Generate correct wire lengths
- Generate output to enable manufacturing of the wire harness
- Not tested were two-way associativity between the electrical and mechanical software, nor were any tests designed to simulate electromechanical interconnections such as activating switches or sensors based on mechanical actions.

Observations

For this test, on the AutoCAD side, AutoCAD Electrical exports an XML file to Inventor. Inventor reads this file and generates the 3D wiring and, under user control, assigns wires to cables. It can then generate wire lengths, a flat wire harness diagram and a pin board for manufacturing.

Inventor opens the 3D model and then the xml file of the net-list from AutoCAD Electrical. This designates the pin-to-pin connections where the wires are to be placed. Different than SolidWorks,

the Inventor video shows placing harnesses in anticipation of the wiring to be imported. The wire import could also have been done first, as seen in the SolidWorks video. The names of the connectors and the number of pins on each connector are stored in coordinated libraries in both the electrical and mechanical systems.



Importing the wires in Inventor (left) and SolidWorks (right).

After the import, the imported wires appear as direct point-to-point connections between the pins without using any harnesses. 19 wires were imported and identified as un-routed. Then Inventor asks for an auto-route of all un-routed wires. It then places all 19 wires into the predesigned harness, we guess by using closest entry and exit points. Then Inventor builds (and reports) a pin board payout of the harness showing the 3D derived wire lengths.

SolidWorks takes a slightly different, albeit very similar approach. After importing the net-list, the operator builds a 3D representation of the harness and then places the wires into the harness, with the software computing the wire lengths. This took more manual interaction than the Inventor solution, but yielded the same end result.

Summary

The two software packages (Inventor and SolidWorks) are comparable. Inventor has a tight connection to AutoCAD Electrical with the xml file transfer. SolidWorks has similar tight coupling with some third party software such as Zuken's E3. Both systems use added cost electrical software to generate the net-list. SolidWorks was not able to read the AutoCAD Electrical generated xml list, and instead used an Excel file with similar data that needed manual cleanup in Excel.

It appears that there are a few more interactions with SolidWorks, but this may be due to the operator-preferred method. Both systems effectively produced the required output. There appears to be no real operational advantage to either package when used with tightly integrated electrical schematics software. Since AutoCAD Electrical is one of the most widely used electrical schematic packages, the advantage goes to Inventor.

Videos of users of both systems performing these tests can be seen on [Ray Kurland's blog, Part 8](http://wp.me/pvn8U-6M) at <http://wp.me/pvn8U-6M>.

About the author

Raymond Kurland is president of TechniCom Group LLC and its principal consultant and editor. His firm, founded in 1989, specializes in analyzing MCAD and PLM systems and has been involved in reviewing and comparing such software since 1987. Ray frequently consults with both vendors and users. Ray has degrees in Engineering from Rutgers University and from NYU. His career included stints with Bell Telephone Laboratories, IBM, and Dassault Systemes. Ray can be reached at rayk@technicom.com.

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