

SolidWorks improves PDMWorks integration

October 10, 2003 -- More than two years have passed since we looked at PDMWorks, a data-management program for small groups of SolidWorks users. (See "[Small vendor tries harder](#)" in the June 2000 *Product Data Management Report*.) Since then SolidWorks has purchased the program from its original developer, **DesignSource Technology**. With SolidWorks 2003 (described in the article "[What's new in SolidWorks 2003?](#)"), SolidWorks has incorporated the PDM software into a new package called **SolidWorks Office Professional**. Priced at \$5,495, this product includes all the programs of the regular SolidWorks Office plus PDMWorks.

PDMWorks provides only the most basic data management functions, but they are the most important ones. The software creates a secure storage area -- a kind of virtual plan vault -- on a workgroup server. When designers are ready to share their models or drawings with others in the local group, they can check them into the vault. Others can check them out for review or reference and may be prevented from making changes unless they are authorized.

The vault also provides a secure place to store drawings and models that are released for production. Each time a drawing is checked into the vault, the software increases the revision number or letter according to a scheme set up by the administrator. Workers can search the vault for the most recent files, or look at past revisions to see what was made in the past.

PDMWorks keeps track of relationships among SolidWorks drawings, parts, and assemblies. If an engineer checks out an assembly, makes a change to one part, and checks the assembly back in, PDMWorks knows to increment the revision of only the part that was changed and the assembly itself. It doesn't check in the other assembly parts or change their revision levels.

PDMWorks also lets engineers create lists showing where parts are used. For example, before making a change to a bracket, a designer should find out how many assemblies use that bracket and whether the proposed change will make it unsuitable for any of them.

PDMWorks lets administrators create project folders for each product or for libraries of company-standard parts and drawings. Purchasing people can generate parts lists and lists of design models and documents for any product or project without needing to have a SolidWorks license or open any SolidWorks models.

Underlying technology

As described in the August 1999 *Product Data Management Report* article "[Another SolidWorks PDM](#)," PDMWorks doesn't employ a relational database such as Microsoft Access or SQL server to index information about SolidWorks files. Instead, it stores each version of a part model in a separate Windows folder in a directory called the "pdmworks vault." In the folder, along with each part, drawing, or assembly model are three small binary files that hold information about the following:

- Where the file is used
- The SolidWorks properties data, such as the author, title, and revision level
- Information about configurations of the part or assembly

Moving or deleting any of these files will render PDMWorks incapable of finding this data. So administrators of SolidWorks networks should install the vault on a server drive that's protected by a password.

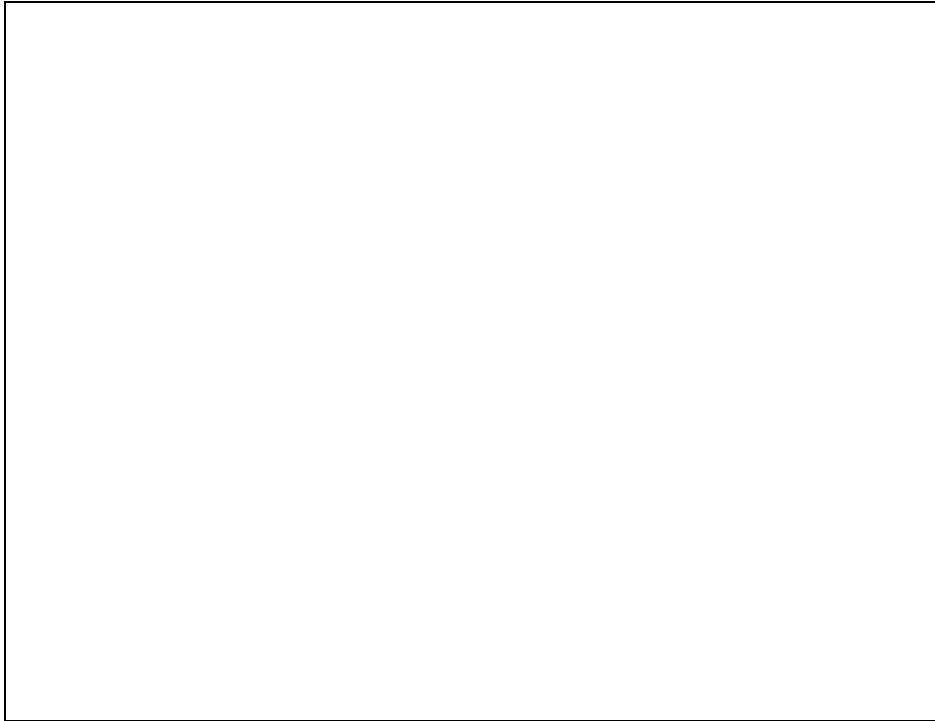
New in 2003

The version of PDMWorks that comes with SolidWorks Office Professional 2003 has important new functions. Chief among these is the ability to create various lifecycle states that affect who may change a file. For example, during the development phase of a project, the creator of each document may be the only one allowed to change it. But when an administrator changes the status of a model to "in checking," control may transfer to a designated checker.

With the 2003 version, PDMWorks is able to understand SolidWorks configurations. This capability is important because a single model with different configurations can represent physically different parts. When models are released for production, factory workers need to know which of the configurations to make. PDMWorks now gives workers the ability to open a specific configuration of a SolidWorks model.

PDMWorks 2003 lets users display custom properties of SolidWorks models, such as surface finish, material type, or supplier. Users can then ask the PDM system to display all parts that are made of a given material or by a particular supplier.

Companies that use standard components from the SolidWorks Toolbox (formerly Cimlogic Toolbox) no longer need to store those components in the PDMWorks vault.



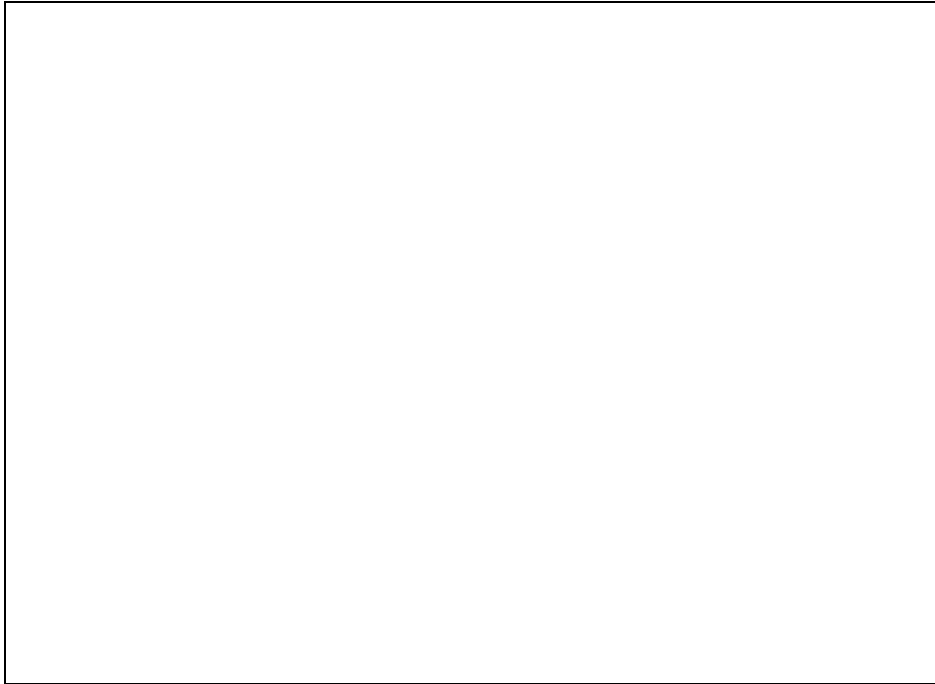
PDMWorks 2003 enables workers to add information, such as part numbers, to drawing and model files after they have been checked into to PDMWorks. (Click image for a larger view.)

Life with PDMWorks

DesignSource literature used to boast that PDMWorks had “a five-minute learning curve,” and SolidWorks sales people have picked up the refrain that PDMWorks is easy to set up and use. Compared with other PDM programs, it is simple to install because it doesn’t require a relational database. It is also relatively easy to set up because it doesn’t do much.

However, the PDMWorks training manual fills 117 pages (excluding the title pages and index). Even the speediest of readers isn’t likely to digest all that in five minutes.

Like all PDM software, PDMWorks is not intuitive. For example, after selecting PDM/Works Open from the SolidWorks file menu, the user is presented with another large panel that has no obvious function. To open a file in the vault, you have to select a project file in the lower left corner of the window, expand the subfolders to display the assembly models, then press the right mouse button in order to be shown the panel that lets you open a file. This procedure isn’t hard to learn, but it isn’t obvious either. Performing the same function in the panel on the upper left of the display will open models that are not in the vault. It’s all clear enough to users who understand what the vault is and why they need to use it, but it requires training.



When a SolidWorks user selects PDMWorks Open from the file menu, this window appears. To open a file, users click a model in the lower left panel and press the right mouse button. (Click image for a larger view.)

The biggest effort that SolidWorks customers will expend with PDMWorks is in learning how to use its functions to their best advantage. Customers need to plan:

- How to set up projects and subprojects
- Revision numbering schemes
- How to assign model and drawing numbers
- What permissions to give to each person in their workgroup
- Which lifecycle states to use
- Which custom properties to employ in the PDM system

Compared with most PDM systems, this is a short list. But meetings with relevant managers and supervisors will be required to make these decisions.



When setting up PDMWorks, administrators need to specify how revisions will be numbered in each stage of product development. (Click image for a larger view.)

Storing and retrieving documents from any PDM system takes longer than it does to load and save a file from a local disk. But even when a PDMWorks vault is set up on the users computer, it takes longer to open a model from the vault than from an ordinary folder. Searching for files also is subject to mysterious delays even with only one user on the system.

DesignSource founder Jim Foster says SolidWorks customers have as many as 50 users sharing a single vault. Some of these are storing models with thousands of parts, he says. Responsiveness will depend on how frequently users check out files and save them to the vault. PDMWorks gives users the option of saving checked-out files locally. In some companies, designers might go to the vault as infrequently as once a day. Workers in other firms might employ it more often.

PDMWorks eats up large amounts of disk storage because it retains a copy of each version of a file. Consequently, before using the software, we recommend investing in a fast server with a large RAID storage system and adequate DRAM. (See “[Storing product data](#)” in the February 2000 *Product Data Management Report* for discussion of this hardware.) Because PDMWorks sends large files to the vault over your local area network, it’s also worth upgrading 10-megabit Ethernet adapters and hubs to the 100-megabit variety, if you haven’t already done so.

PDMWorks won’t help your company collaborate with outside suppliers. SolidWorks has a service called 3D TeamWorks for that. Like Autodesk’s Streamline (see “[Autodesk Streamline](#)” in the September 2001 *Product Data Management Report*), 3D TeamWorks provides project folders where manufacturers can upload files and models for review by customers or suppliers.

Advanced Server

SolidWorks 2003 includes a new software option called PDMWorks Advanced server. Priced at \$4,995, this program provides an application-programming interface (API) that lets customers write programs to automatically extract data from the PDMWorks vault. The Advanced Server could be used, for example, to automatically generate data for other corporate systems, including ERP, cost estimating, and so-called enterprise PDM software. Specific events or processes can trigger programs to run in the Advanced Server, such as a nightly update of the ERP system.

The Advanced Server option also enables workers to view data in the PDMWorks vault with only a Web browser. Without this capability, workers must have licenses of SolidWorks Office Professional or the \$995 PDMWorks client license.

Who needs it?

Like most computer programs, PDMWorks has room for improvement. For instance, not all the panels have a help button. However, the defects we've encountered are minor annoyances rather than showstoppers. SolidWorks customers who don't yet have any PDM system ought to start with PDMWorks before attempting to use more complex systems, such as Dassault Systèmes SmarTeam.

When installing PDMWorks, it's helpful to get advice from someone who has done prior installations. As noted above, the hard part about setting up the system is understanding all PDMWorks' capabilities and how they might best serve your existing document control and release processes. An experienced SolidWorks dealer or consultant should help your company get off on the right foot with its first PDM system. Contact: **SolidWorks Corporation, 300 Baker Avenue, Concord, Massachusetts 01742** Telephone: **1 (800) 693-9000** or **1 (978) 371-5000** Fax: **1 (978) 371-7303** Web site: <http://www.solidworks.com>.