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CAD Manager's Guide to Autodesk® Vault and Autodesk® Productstream™

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MA13-2 This course is designed for both CAD administrators and power users of companies that have Vault installed and ones that are considering implementing Vault. In this session, you'll learn concepts such as Vault implementation, configuration, and administration. We'll outline various Vault configuration scenarios, from a single user working on a small assembly to large teams working on assemblies composed of thousands of parts. Other topics include property tuning, working with libraries, and remote Vault user scenarios.

About the Speaker:

Brian manages the data management solutions at MasterGraphics. He has worked with large corporations to implement Autodesk data management solutions, specializing in Autodesk Inventor, Autodesk Vault, and Autodesk Productstream. Brian has taught numerous classes in Autodesk mechanical applications and is a recognized expert, instructor, and consultant in mechanical CAD and data management. He is both an Autodesk Inventor Certified Expert and Autodesk Manufacturing Solutions Implementation Certified Expert.

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The Vault Environment

The vault is a true client server environment that allows users to manage their Autodesk Inventor data as well as all other related design data in a controlled manner. One of the keys to understanding how to work with the vault and the general use of the vault is to understand the way data is moved from the clients to the server.

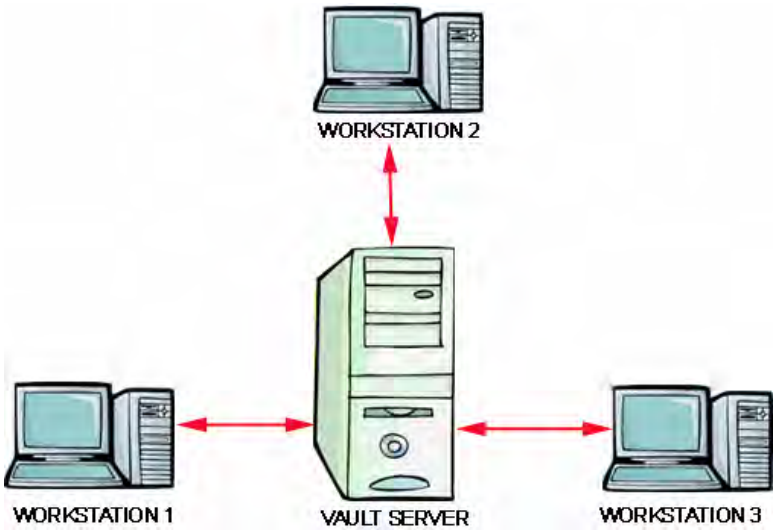


Figure 1 - "Client / Server" Vault Environment

The above figure shows how clients are connected to a central, common server where the master data is stored. When files are added to the Vault, the data is then considered "owned" by the vault. When a user wants to work on a particular file, the data is then "checked out" by a user for editing. When the user is finished editing the data, the changes are "checked in", updating the server and making the data available to the other users.

Another key to understanding the vault is that at any time, while another user is editing a file or not, you may have a copy of any file from the vault on your local machine. These files that are copied locally are just that, *copies* of the data and the master is still kept in the vault. These files are initially set to be read-only by the system. When you check out files, the read only flag is removed and the files are available for editing.

The read-only setting acts as the first line of defense for the user when attempting to edit a file that isn't already checked out. Because the file is read only, the operating system won't allow you to save edits to the file, therefore alerting you that you have to check it out first.

Check In / Check Out

One of the core functions of the vault is its ability to lock, or reserve, files that are in the vault to a single specific user. This allows you to be confident that the files you are working on are not going to be edited or changed by other users while you are working on them. This functionality in the vault is known as "check in" and "check out".

This ability is fundamental to the vault and allows you to avoid the "last saved wins" situations that are so common when working in a networked environment with Windows Explorer.

It is also important to understand that when working with the vault, you do not have to check out all files that you currently have open. It is possible to only check out the files that you actually want to edit, while maintaining a read-only copy of the rest of the dataset on your local computer.

After you are finished making changes to your designs or simply want to save an updated version of the file to the vault, you can check the file in. "Checking In" a file transfers an exact copy of the last saved version back to the vault and removes the lock on the file. This process doesn't overwrite the last version that was checked into the vault, but actually stores a new version of the file making sure that you can always retrieve a copy of any other version at any time.

Centralize Storage

One of the biggest benefits to implementing the vault is organization. Users can feel safe that when they browse the vault, it is the location of their master files as well as the location of the latest version of their files. The vault provides a single source of data.

In addition to working with Autodesk Inventor and AutoCAD Mechanical files, you can manage any other file you have though the use of the Autodesk Vault Explorer. This means that all of your related engineering documentation can easily be managed, shared, and version controlled along with you Autodesk Inventor designs.

Indexing

Indexing is a very important part of the vault. Indexing refers to the way the data that is extracted from the files is cataloged for efficient use. The indexing in the vault works similar to popular search engine technologies like Google and Alta Vista, allowing them to find millions of web pages in a matter of seconds. The vault indexes all of the textual data from files, allowing the vault to handle massive quantities of data while still maintaining its performance. In addition to indexing Autodesk Inventor files, the vault is also able to index content from Microsoft office documents as well as others.

The most important aspect of the vault indexing technology is its ability to automatically adapt to new properties that are added to files or when new file types are added to the vault. When the vault finds a new file type, it instantly adds it to the system and makes it available for searching or display in a column.

Version Control

One of the key benefits of the vault is its ability to automatically version control files, with the understanding of any related files that are required in order to rebuild the data set. A perfect example of this in terms of Autodesk Inventor is the drawing file (*.idw) and its associated part file (*.ipt). In order for the drawing to be complete, the two files must exist as a perfect, matched pair. The vault automatically tracks these files and their relationships making them available whenever they are needed in the future.

Traditionally, you might do one or more of the following to save past versions of your designs:

- Print a hard copy of a drawing,
- Save a complete copy of your design into alternate folders,
- Archive your data into zip files or onto CDs.

All of these methods are manual, time consuming, and prone to error. Since the vault automatically tracks the appropriate relationships and versions, having to perform one of these manual steps isn't necessary. When you need a past version of data, all you need to do is retrieve it from the vault. The files, plus any associated files, are automatically retrieved from the vault exactly as they were when they were originally checked in.

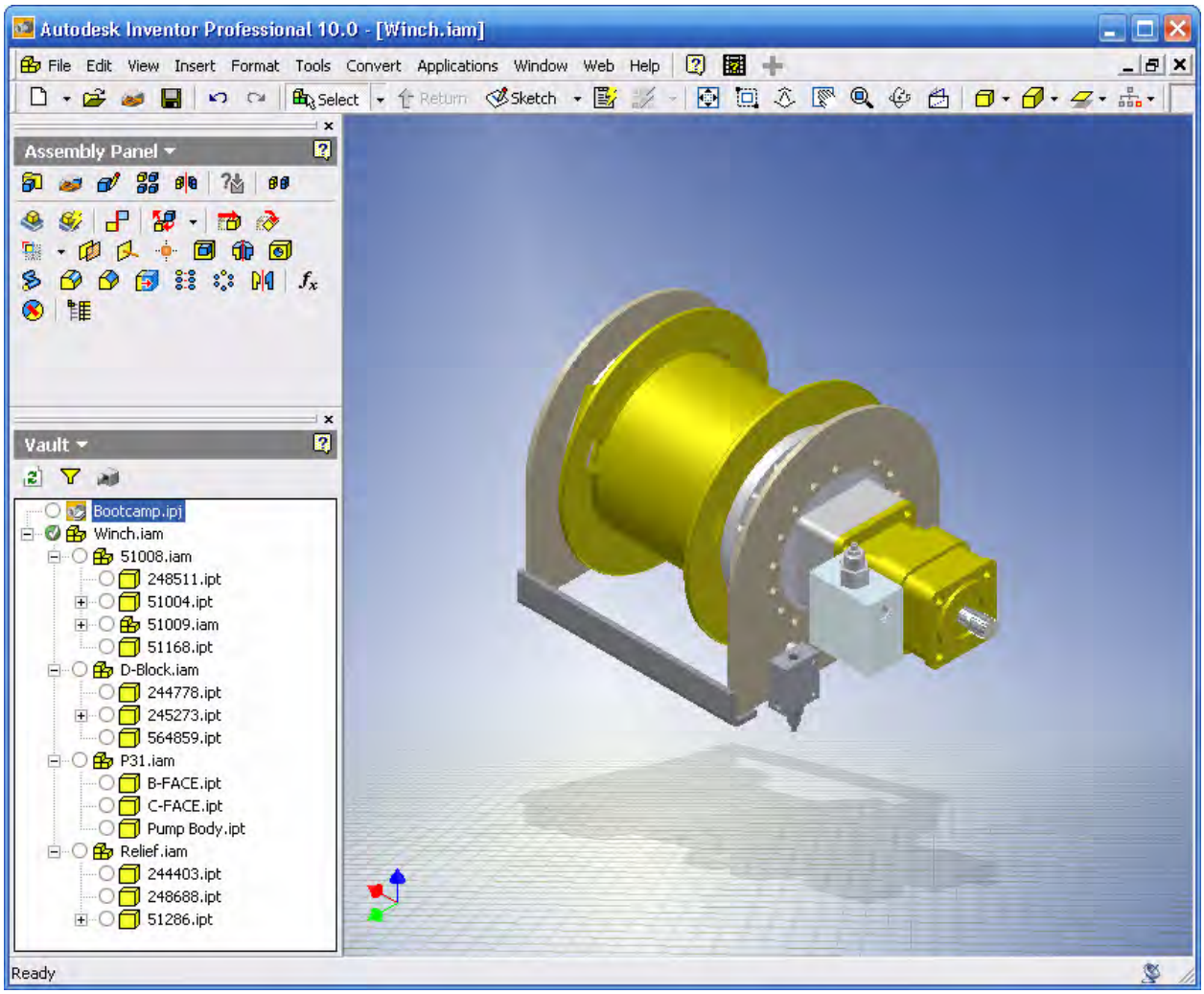
Autodesk Vault for Autodesk Inventor

Autodesk Vault is integrated into the Autodesk Inventor user interface allowing you to perform nearly all of your file management and vaulting tasks without ever leaving the Autodesk Inventor environment. From adding files to the vault, to checking them into the vault after changes are complete, the vault browser is a central location for working with your files, in relation to the vault.

Another key to the browser is its ability to display the status of each file via an icon. This allows you to see any of the following states:

- Files that are new and not yet added to the vault.
- Files that are currently checked out to you.
- Files that are checked out to anther user.
- Files that are newer or older than the last version checked into the vault.

These icons allow you to constantly be informed as to the current state of the data you are working with.

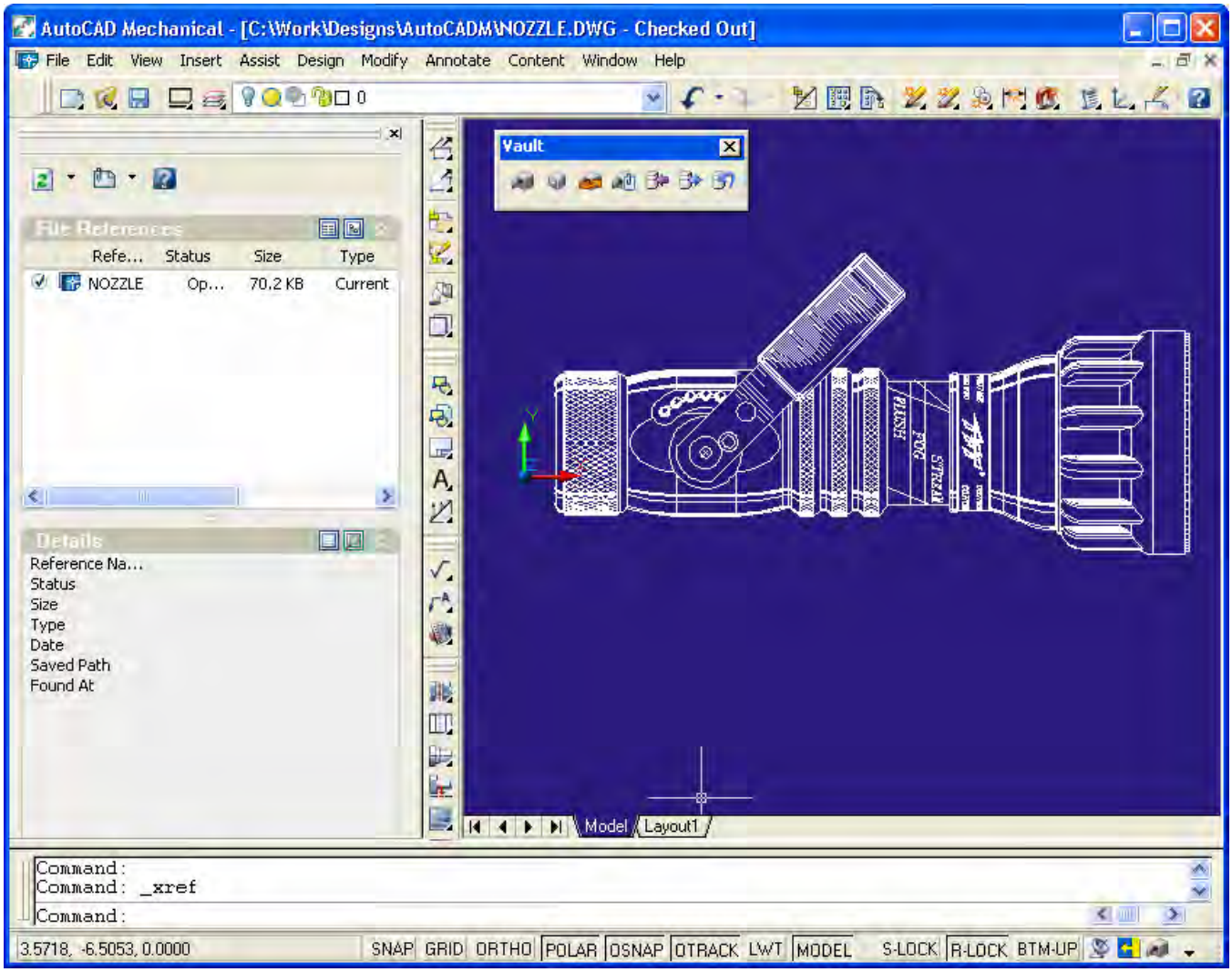


Autodesk Vault for AutoCAD Mechanical

Autodesk Vault integrates into the AutoCAD Mechanical environment using the AutoCAD Vault ARX. The Vault is accessed from a pull down menu, command line, or toolbar within AutoCAD Mechanical, allowing you to check files in and out from within the application, and also keep track of attachments to the files.

Similar to the Autodesk Inventor interface, you can perform most of your file management tasks without leaving the AutoCAD Mechanical user interface. This is achieved through the Xref Manager. Inside the Xref Manager you can:

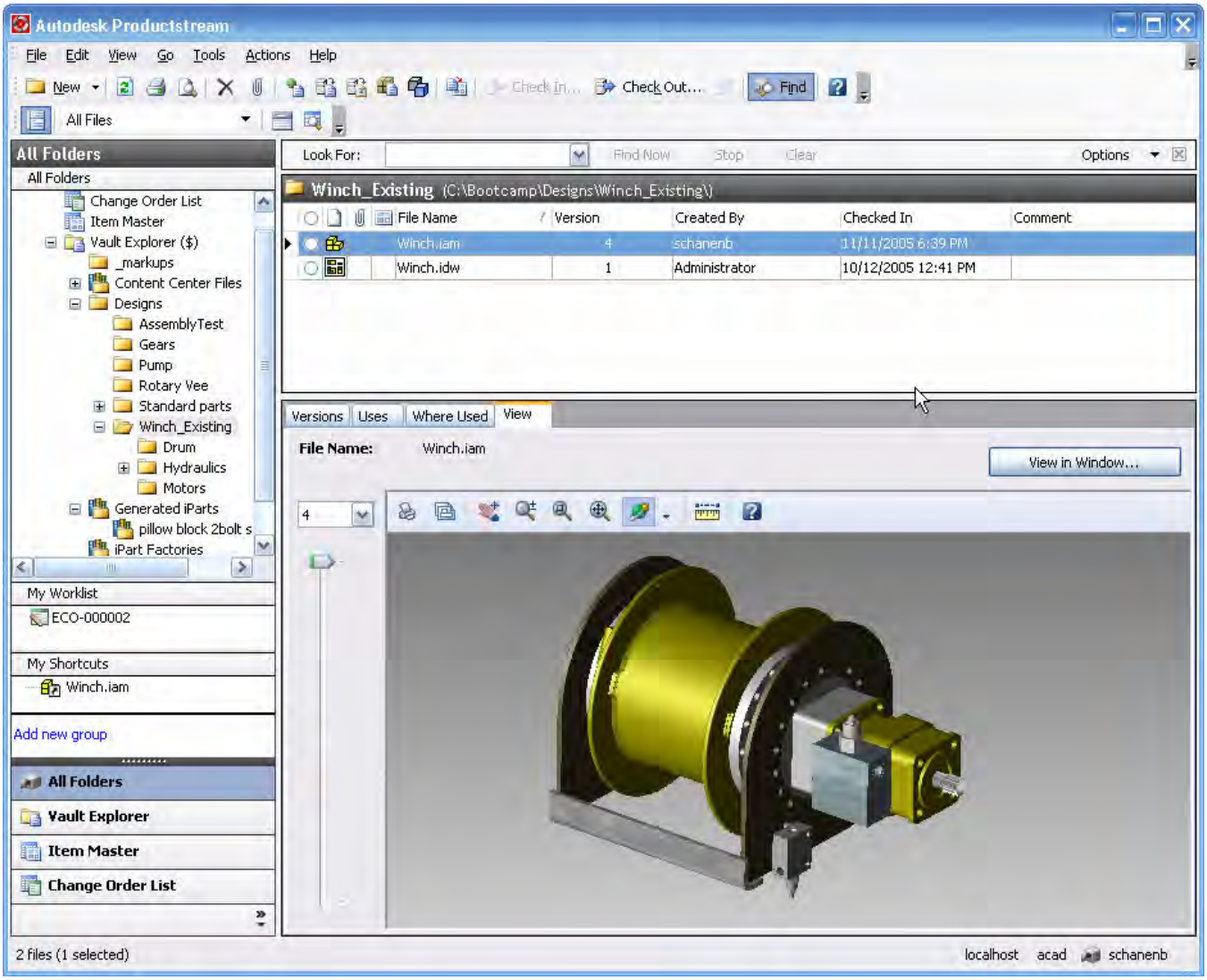
- Check in and check out AutoCAD Mechanical files from Vault.
- Perform Xref commands such as Reload, Detach, Bind, and Open.
- Determine the Status of vaulted dwg files.
- Attach from Vault to share and reuse dwg data.



Autodesk Vault Explorer

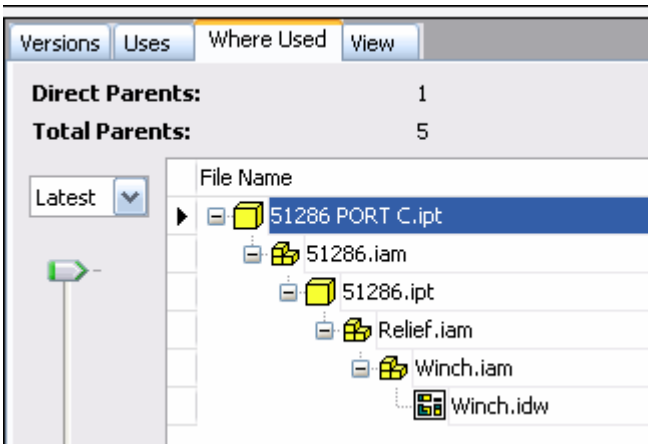
The Vault Explorer is a stand-alone application that runs alone side of Autodesk Inventor or simply by itself and allows for a complete view of the data that is within the vault. The vault explorer is organized like Windows Explorer or Microsoft Outlook with a folder tree on the left and the list of associated files on the right. The vault explorer however is different form working with a standard file system. When browsing the vault via Autodesk Vault Explorer, you are essentially looking at a virtual file system that only exists in the vault and not anywhere that is on disk. Because the vault requires a secure username and password to access it, your data is safe from unauthorized access, preventing unwanted change.

In addition to providing access to the virtual file system, Autodesk Vault Explorer also offers the ability to search for files based on any combination of criteria, view "Where Used" information about designs, giving you the ability to make an informed decision on change as well as the ability to manage any non Autodesk Inventor files that you have on your system.



Where-Used

The "Where Used" capability in the vault is fast and extensive. Autodesk Vault allows you to see any related files either directly related or related through any number of higher-level parents. A benefit of the Where Used capability over the Autodesk Inventor Design Assistant is its ability to perform the "where used" throughout the entire vault, not just the scope of the current project. This means that files such as standard parts from your company's library that are used over and over again throughout many assemblies, presentations, and drawings can be accurately investigated and changed when needed.



Rename Wizard for Inventor

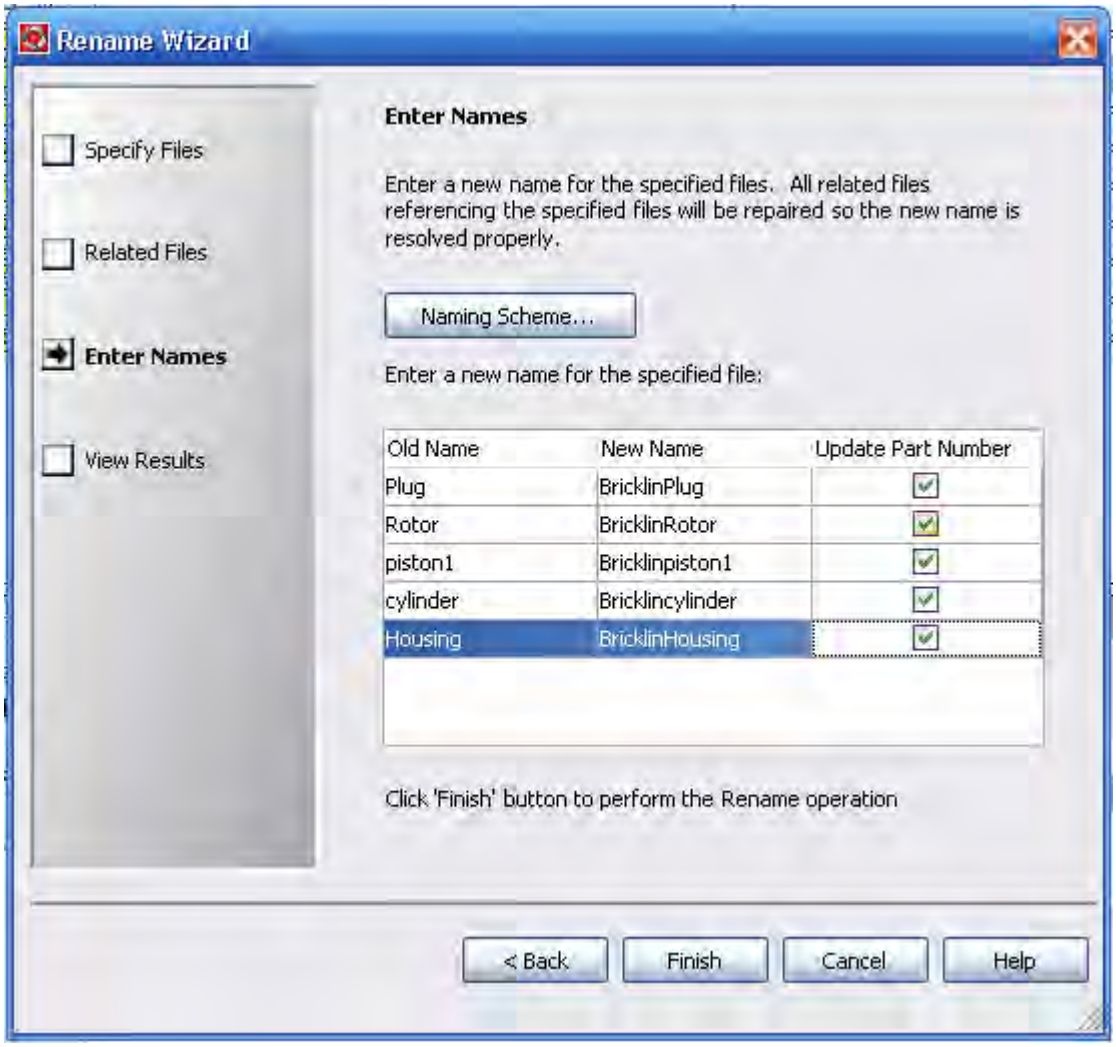
Renaming files with Autodesk Inventor is traditionally a very painful task. There are a couple of ways to go about it:

- **Design Assistant** – Can be cumbersome and difficult to use. When there are many files located over a network, the performance (due to the "brute force nature") can be horrendous.
- **Windows Explorer** – Rename the file on disk, open it in Inventor and fix any broken references. This method can be very dangerous and time consuming.

With Autodesk Vault you have the ability to rename file with a simple wizard interface. The Rename Wizard lists all parent file dependencies and updates these references as needed.

Often times, in Top Down Assembly modeling, components are created with generic name for prototype purposes (Part1.ipt, Part2.ipt,etc.) This can be a huge time saver and, alone is worth the price of admission for anyone who needs to rename their Inventor files for tasks such as release to production (rename from name to part number, etc…)

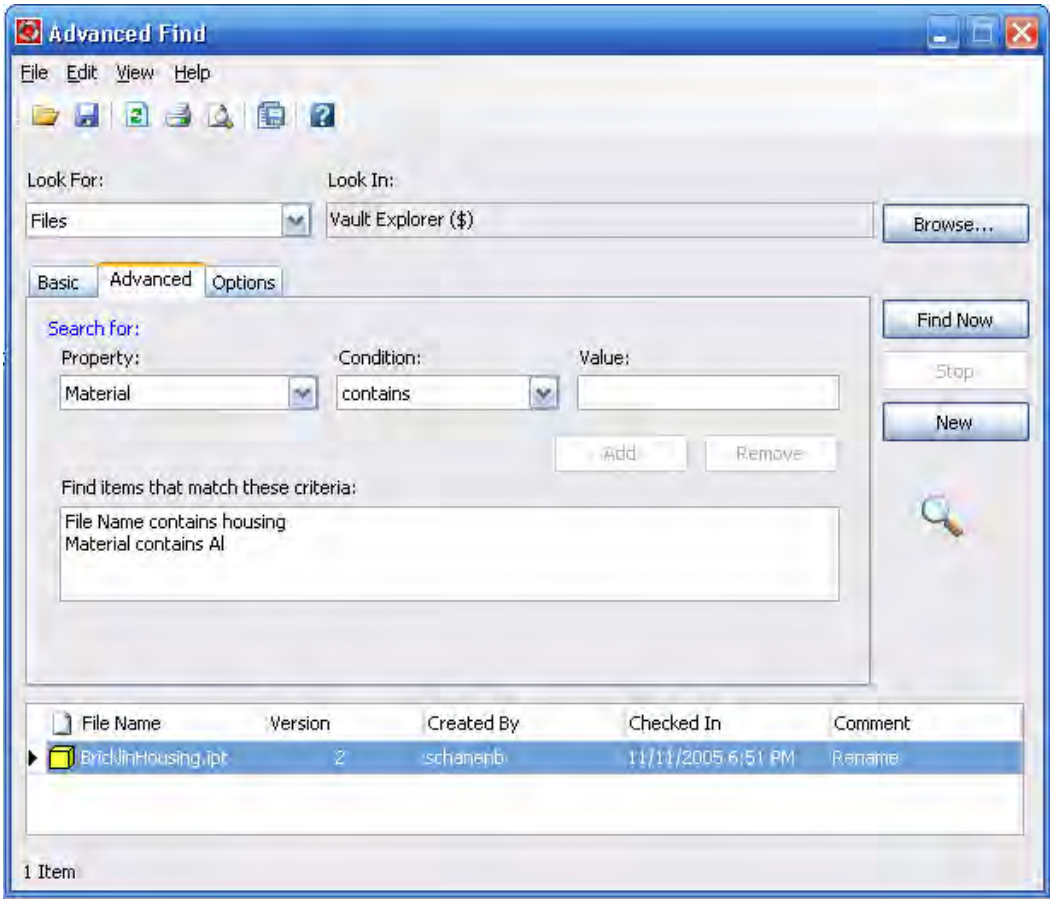
Note: Files being renamed as well as all related files must be checked into the vault before renaming.



Basic and Advanced Searching

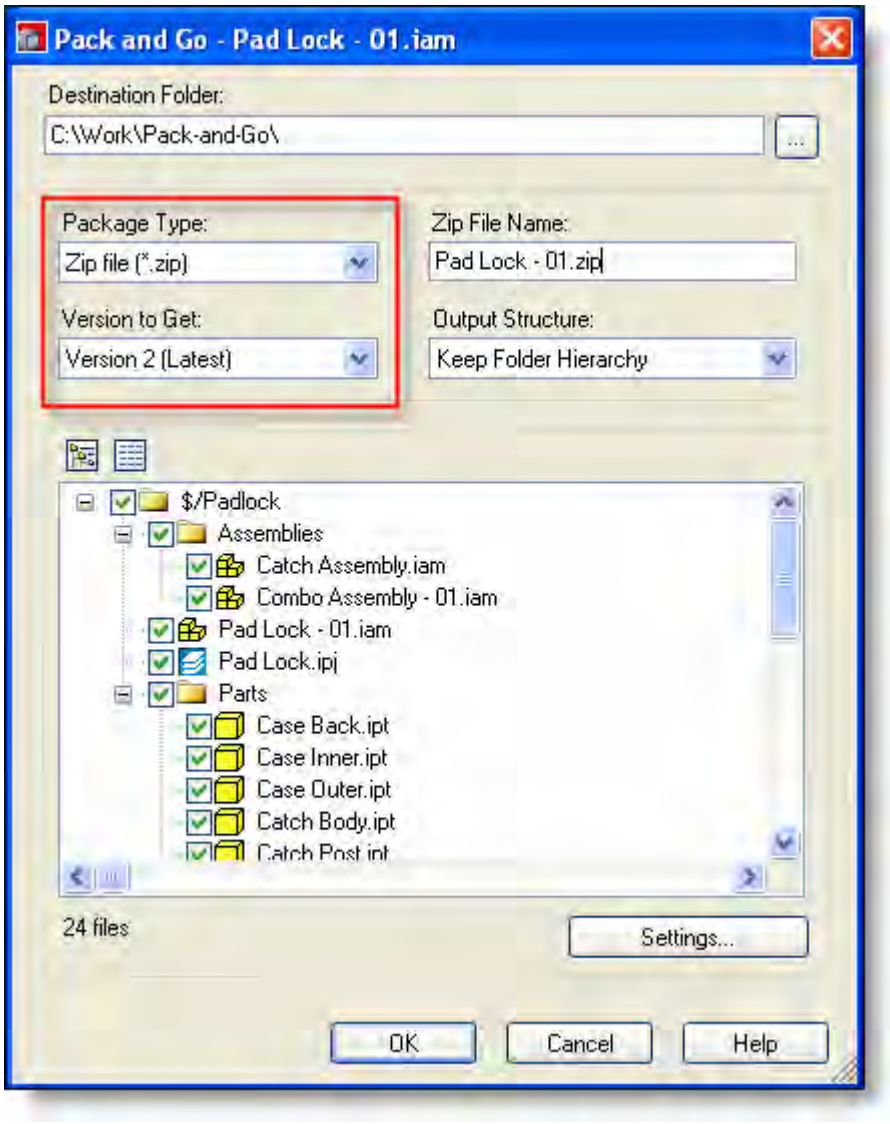
Leveraging the Indexing technology, the vault is able to search through the many versions of files in an extremely fast manner. The Basic tab performs a text string search on properties in the vault. The Advanced tab searches for specific patterns within the entire vault based on any criteria.

Typically, advanced searching involves compound criteria such as "iParts I have modified this month that are made out of steel but not stainless steel". The vault's searching provides a significant advantage over searching for files via the standard Windows Search interface or the Design Assistant.



Pack-and-Go

Autodesk Vault additionally provided the ability to "pack-and-go" a design right from the vault. Pack and go will take either all children or all related files (drawings and presentations) and collect them into a single location that can be used to send to external sources or potentially archive. In addition to working like the familiar pack and go with Inventor, the Vault brings the ability to pack and go both pervious versions of a design and the ability to pack and go to a Zip file.


Pack and Go - Pad Lock - 01.iam
Destination Folder:
C:\Work\Pack-and-Go\
Package Type:
Zip file (*.zip)
Zip File Name:
Pad Lock - 01.zip
Version to Get:
Version 2 (Latest)
Output Structure:
Keep Folder Hierarchy
$/Padlock
Assemblies
Catch Assembly.iam
Combo Assembly - 01.iam
Pad Lock - 01.iam
Pad Lock.ipj
Parts
Case Back.ipt
Case Inner.ipt
Case Outer.ipt
Catch Body.ipt
24 files
Settings...
OK
Cancel
Help