PTC Technical Specialists E-Newsletter

Date: June 2009

PTC Product Focus: A) 3D Drawings in Pro/ENGINEER Wildfire 4.0

         B) Windchill PartsLink Classification and Reuse

Tips of the Month: A) Integrating Designs using Pro/ENGINEER WGM

         B) Windchill PartsLink: Creating your own Parts Catalogue

Announcements: Most Recent Announcements

**Upcoming Events & Training Schedule:** Events & Training Schedule

---

PTC Product Focus

3D Drawings in Pro/ENGINEER Wildfire 4.0

Background

There is a cultural shift underway to define new standards and practices for communicating engineering designs to manufacturing and other stakeholders. The conventional approach has been the 2D orthographic drawing to present a complete and unambiguous representation. This must include not only accurate geometric views and dimensions, but items like tolerances, notes, etc. Advances in CAD technology over the past 20 years evolved from electronic drafting, to associative drawings of feature-based 3D models that capture a complete digital product definition. Consequently, the traditional standards are no longer the most efficient – nor even adequate for many designs.

In 2003, ASME International extended the Y14.5 drafting standard into the digital era – ASME Y14.41 Digital Product Definition Data Practices! This shift will realistically take many years to adopt and fully implement. Various industry segments, i.e. Automotive and Aerospace & Defense, have been leading the transition. The rich data encapsulated within the 3D models requires not only new functionality to capture it, but downstream tools to view and consume it. The vision of not only paperless, but also drawingless environments eventually will be realized…
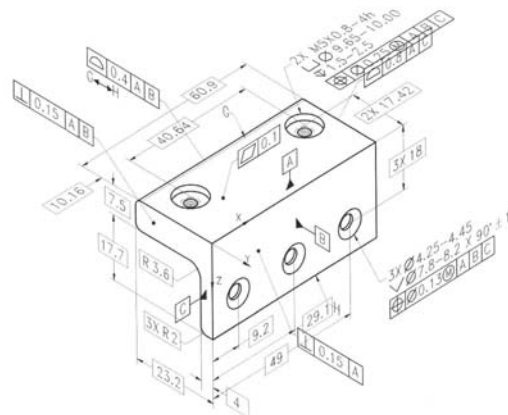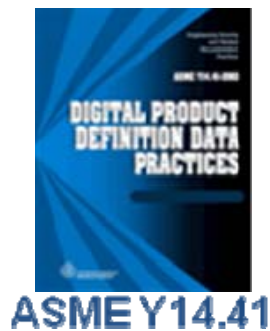
To support this, PTC has been making major investments in Pro/ENGINEER and ProductView to support this emerging standard. In particular, Pro/ENGINEER Wildfire 4.0 is a very significant release with functionality PTC refers to as "3D Drawings." This is not a new module, but pervasive functionality implemented in the core product as **Feature-Based Annotations**. They can be leveraged in many ways resulting in smarter models and process automation (for example, to drive Pro/NC templates). The focus of this article will be on the *drawing* aspects.
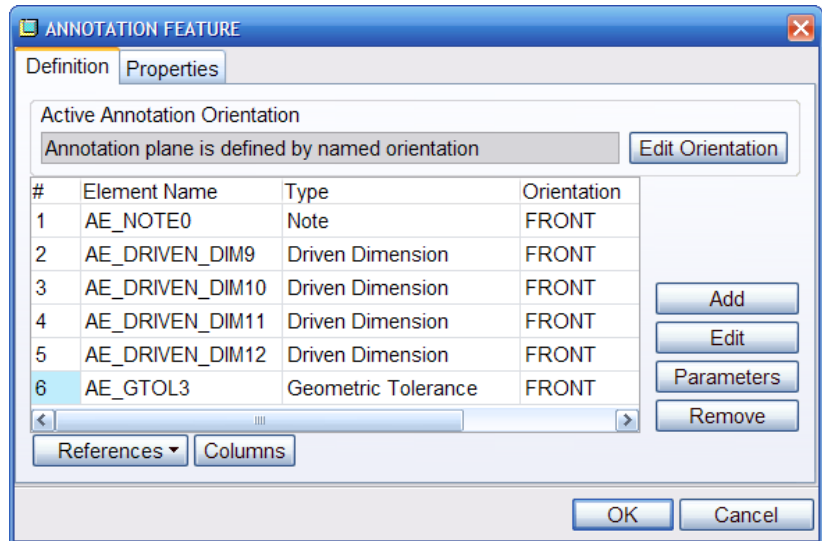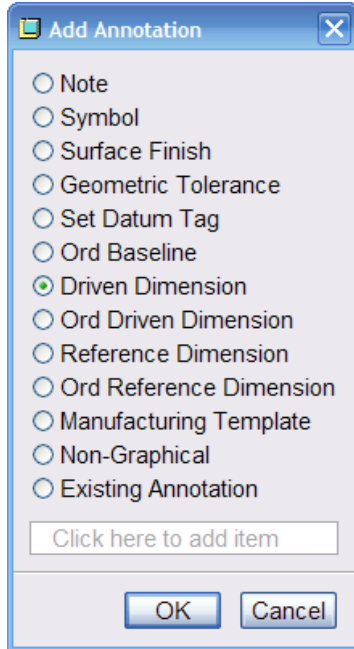
Example 3D Drawing
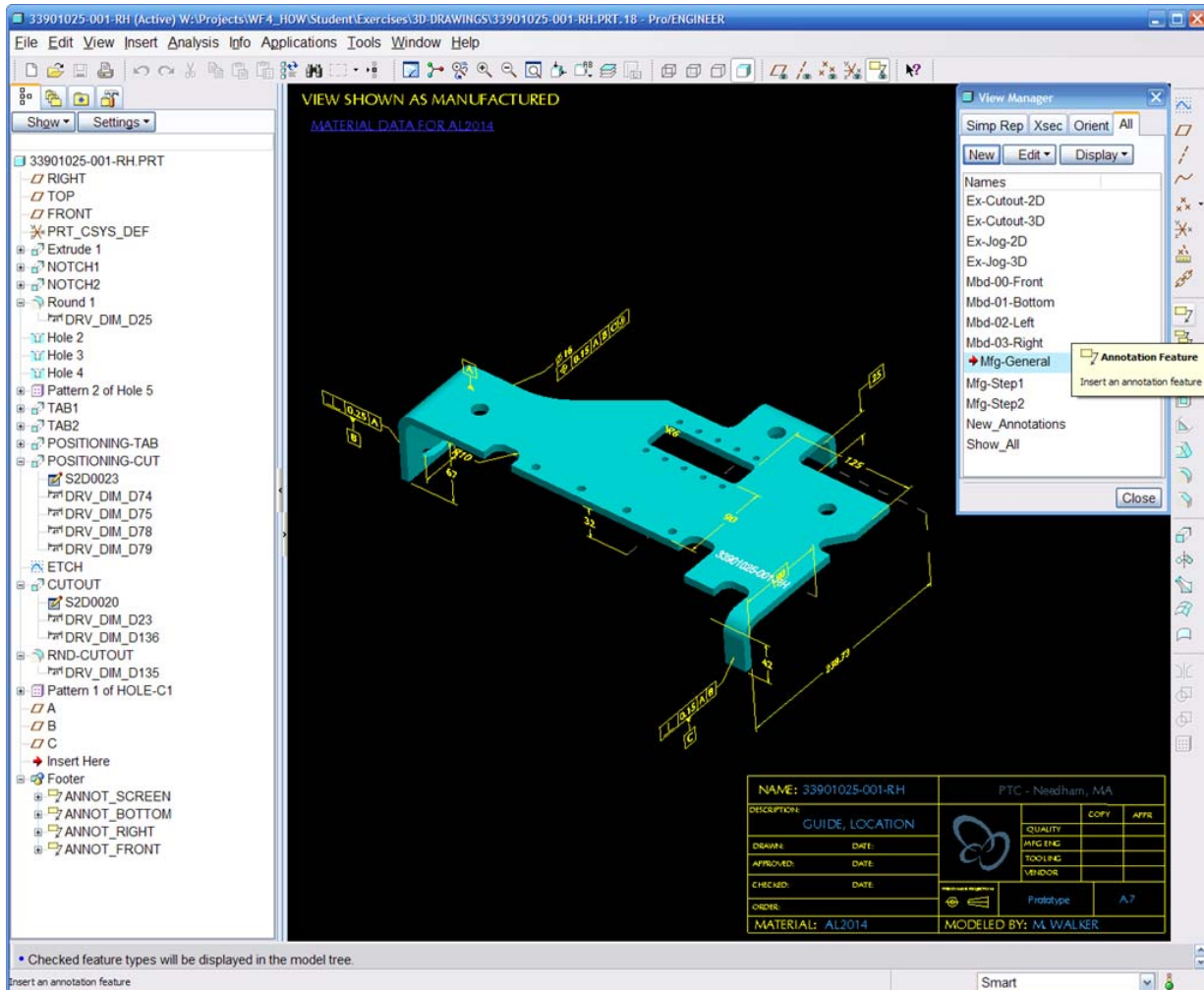
Annotation Elements

3D detailing items are referred to as "**Annotation Elements (AE)**". Create them with **Insert > Annotations > Annotation Feature…** or [icon] on the feature Toolbar.

Notice that an **Annotation Feature** is a collection of one or more **Annotation Elements**. These appear in the Model Tree and can be manipulated in many ways like other features.

Unlike old-style 3D notes whose text remained flat to the screen, these can be flat to a particular orientation reference. Wildfire 4.0 allows the designer to designate an active orientation reference, which will remain modal for subsequent elements.

Select **View > Annotation Orientation**

There are many flexible options available to define this reference – including existing Named Views! The yellow arrow indicates the direction of viewing; the red arrow indicates the text direction.
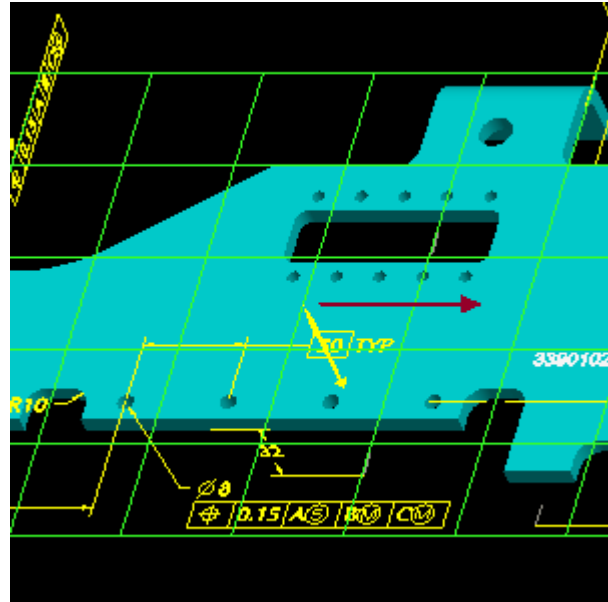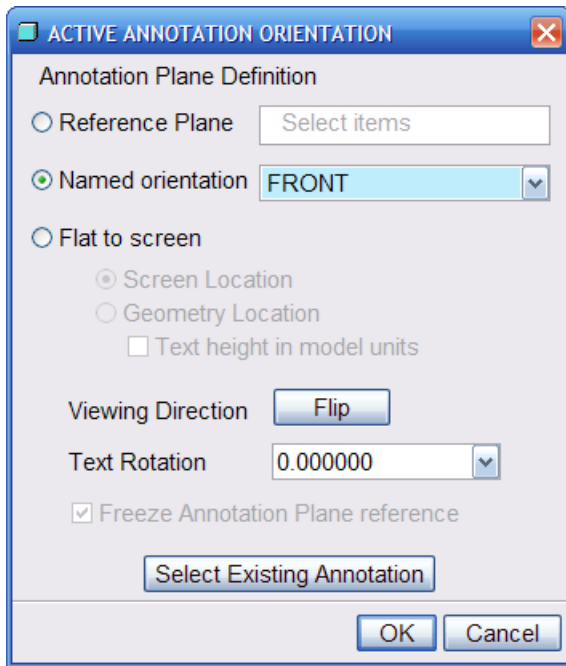
In Wildfire 4.0, existing feature dimensions may be selected to <u>directly</u> create an Annotation Element. Simply select the feature from the graphics area or model tree, then **RMB > Create Driving Dimension AE.** The familiar **Dimension Properties** dialog box can be used to modify the appearance. Additional RMB commands are available as shown below.
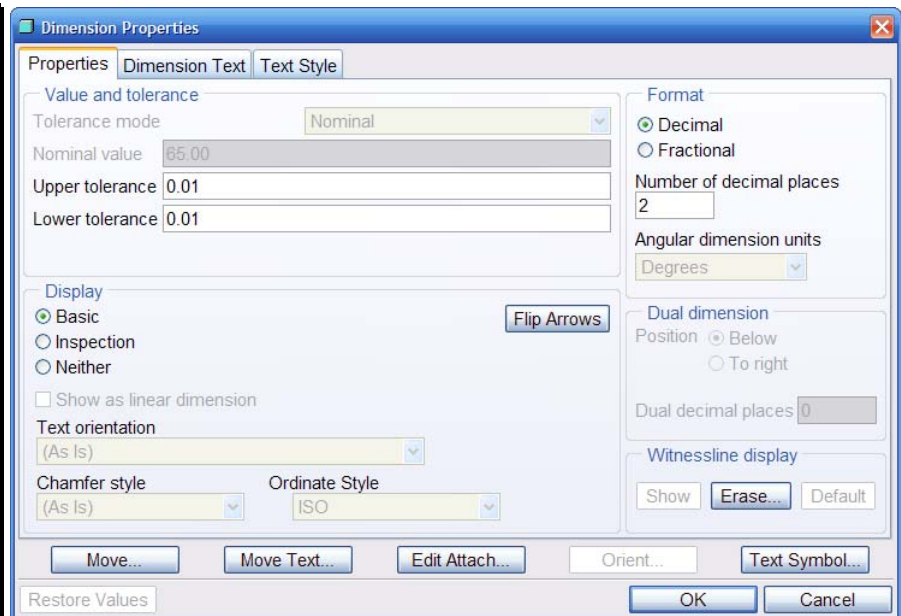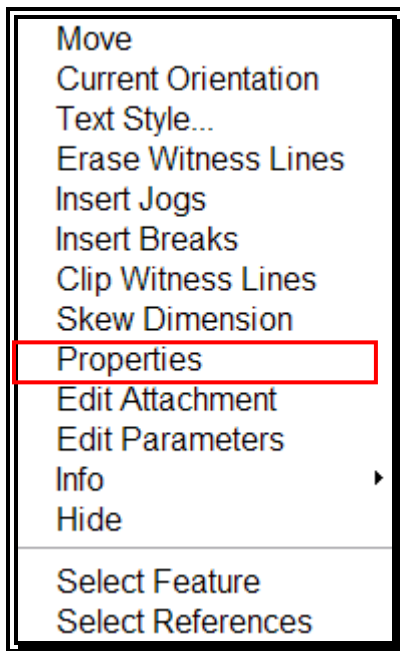
These annotation elements are added as nodes under their driven parent feature, not as separate Annotation Features. There is a Model Tree filter to control their display.

It is helpful to set the selection filter to **Annotation** when working with these features. With an annotation selected, the designer can also use commands on the RMB pop-up menu to highlight corresponding geometric **Features / References**.

Visibility & View Manager

Toggle global visibility using the new icon on the standard datum display toolbar.

Achieve more granular display control using layers. Rule-based layers enable items to be added to particular layers based upon criteria, like feature parameter values. This enables elements to be automatically grouped according to process requirements.

A great new capability in Wildfire 4.0 introduces the ability to save layer status with a **Combined State** in the **View Manager**. Various model-based definitions can then be turned on / off as needed using a meaningful name.
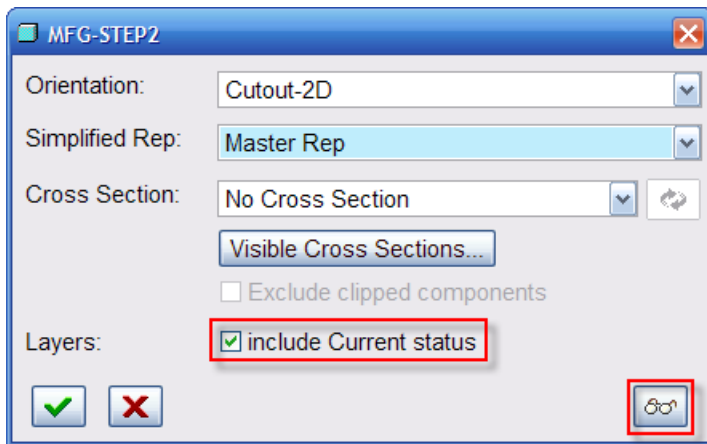
Additional Capabilities

- Fixed / Flat to Screen, including symbols for title blocks, etc
- Embedded Hyperlinks
- Ordinate Dimensions

We have just scratched the surface of what is possible. There are MANY other capabilities available, with more improvements scheduled for Wildfire 5.0 in mid-2009!

More Information

There is an on-line tutorial available to try the concepts presented in this article.

Smarter 3D Drawings capabilities were one of the Top 10 Reasons to upgrade to Pro/ENGINEER Wildfire 4.0!

> *"Increase design information reuse and deliver production ready drawings with the **automated display of dimensions in 3D drawings**. Organizing the drawing content is also made easier through **layer status control in the View Manager**, editorial control of **multiple annotation types**, and tabulation of restricted parameters. You can also **publish and access saved 3D drawing views in ProductView™**."*

To learn more about exactly what else is new, please visit the Interactive Release Notes.

<div align="center">

Back To Top

</div>

<div align="center">

PTC Product Focus

</div>

Windchill PartsLink Classification and Reuse

<div align="center">

Click Here To View

Back To Top

</div>

Tips of the Month

Integrating Designs using Pro/ENGINEER WGM

Introduction

The combination of Pro/ENGINEER Wildfire 4.0 and Windchill PDMLink or Pro/INTRALINK 9.1 is <u>very</u> powerful. The functionality gaps with Pro/INTRALINK 3.x are effectively closed, and many advanced and improved capabilities are available. Designers are able to collaborate very efficiently, with complete confidence that they are free to explore design alternatives while maintaining configuration control.

- What's New – Interactive Release Notes
- Workgroup Manager for Pro/ENGINEER Compatibility Matrices

WGM Defined

WGM is an acronym for "WorkGroup Manager." Windchill can support heterogeneous data management and offers optional workgroup managers with a unified user-interface for use with mainstream MCAD and ECAD systems.   With Pro/ENGINEER Wildfire, this component is seamlessly integrated to enable PDM functionality for use with Windchill-based solutions. The techniques in this article apply 100% to Pro/INTRALINK 9.1 as well, which is a pure subset of PDMLink specifically focused on managing Pro/ENGINEER designs.

This tip will present a very common practice of integrating data into the Windchill repository after a new implementation or migration. It is also common when vendor parts need to be introduced into the system. This will be a short tip and will make practical sense once you go through it. As with most things, there is more than one way to accomplish this task.

Understanding Retrieval Precedence

When retrieving objects into a Pro/ENGINEER session, there is a very specific and predictable search order for source files. This is important to understand and can be used to your advantage when loading a design from the hard disk that may also be partially or entirely in the Commonspace. The use case could be when a design was exported from the data management environment, and now needs to be re-integrated as the next iteration.

The search / load order is as follows for the specified object AND DEPENDENTS:

1. In Session
2. Retrieval Directory   (folder browsed to; may be same as #5)
3. Workspace   (active only)
4. Commonspace   (automatic search into contexts that user has access)
5. Set Working Directory

*Note: search_paths are ignored when registered with a Windchill server.

Showing Model Retrieval Path

First object found "wins" and is loaded In Session. It is sometimes helpful to see where objects were loaded from. The trail file shows extended information about load times and source. In addition, the following config.pro option will update the window title, and paths reported in reports, like Model Info.

    display_full_object_path yes

In this example, an assembly drawing was exported, and the following changes were made:
- Added new part acme-54173.prt to top-assembly 72100.asm
- Modified one part - 72130.prt.

Figure 1- trail file retrieval details

Figure 2 – Info > Model > Top Level

If the entire assembly was opened from a single hard-drive folder, then saved to the Workspace, there would be <u>many</u> conflicts and unmodified objects could potentially be iterated.

BOTTOM LINE -- Moving ONLY modified objects to a sub-folder for retrieval forces Pro/E to search in the Commonspace for pre-existing [unmodified] objects. The **Add to Workspace** is automatic and occurs in the background!

Save the Assembly to the active workspace (default).

Compare Status

The table displays can be customized by the system admin or individual users to include various columns and filters. The status columns are very important to understand and determine the state

of an object. By default, the **Compare Status** is not shown (in order to improve performance).
A conflict here is often the cause of upload / check-in failure(s)!

Select **Customize…** in the drop-down **View** list, and add the columns as shown.
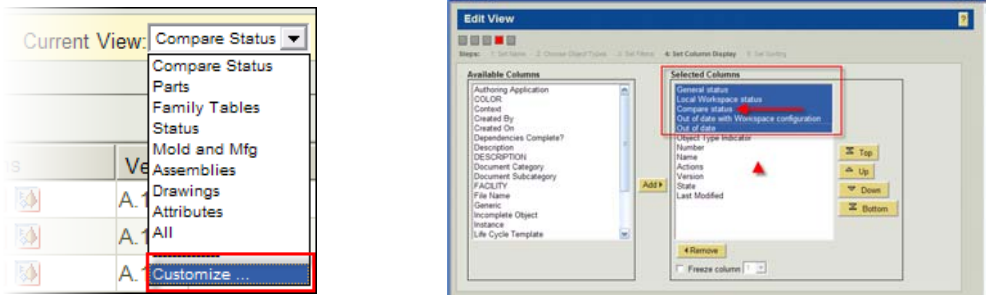
Resolving Conflicts

After saving the design, notice that the BASKET and END CAP indicate a **File name conflict**.
That is because these objects already exist in the Commonspace – but the WGM doesn't have
local metadata to recognize that they are truly the same objects – that is up to the designer!

The status columns indicate the following:
1. the item is **NEW**
2. the item is **LOCALLY MODIFIED** in the cache
3. the item name **CONFLICTS** with an existing item in the Commonspace
4. the **Number** field is not populated until initialized with Windchill (upload, etc.)
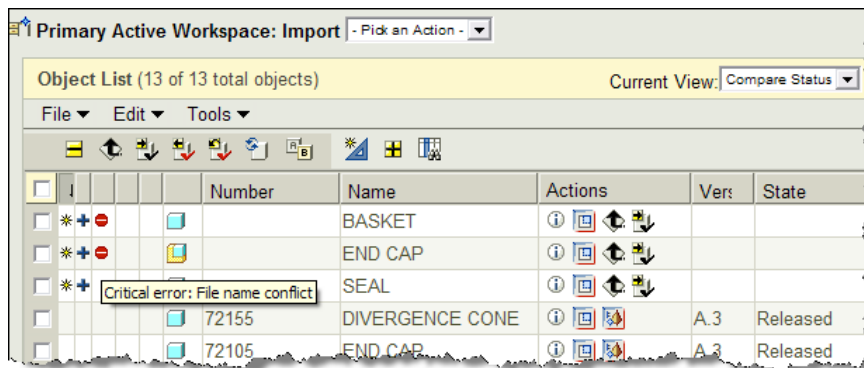
There are a few ways to address this…conceptually, you need to preserve the local file content,
while linking with existing metadata. The desire is to have a locally modified object which
represents the next iteration.

1. With NO objects selected, pick **Add to Workspace**
2. **Search** for the top-level assembly
3. Activate the **Advanced** tab to see collection
4. **Remove** or exclude objects that have NO status (no action required)

5. **Set** the selected objects to **REUSE** and **CHECKOUT**.
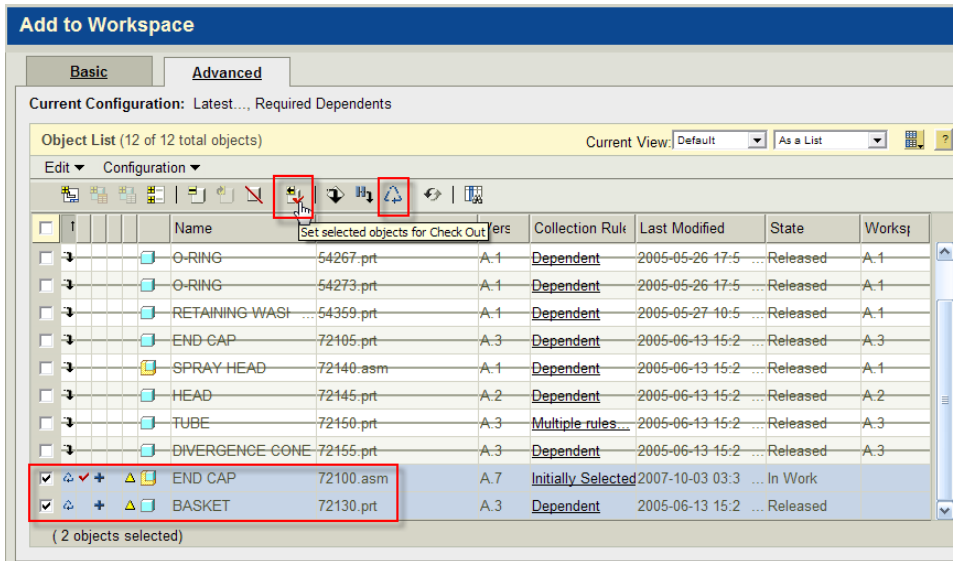
(Of course, the user must have adequate permissions to **Check Out** the existing object.)
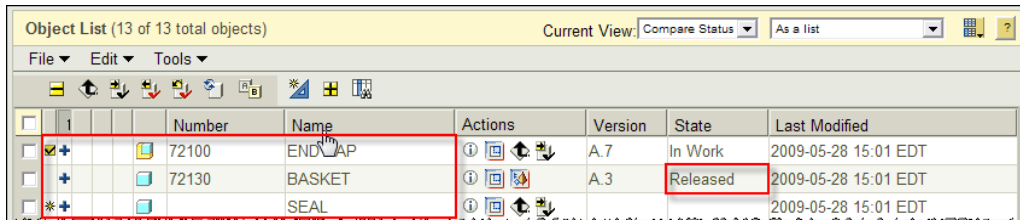
Notice that the metdata is now synchronized, and the END CAP can be checked-in as A.8. The BASKET is Released, so it must be Revised to enable check-in as B.1 | In Work.

Check-In

As a best practice, it is a good idea to perform an upload at regular intervals. This will protect your new and modified data prior to sharing with others. This copies the files from the local cache to your personal cabinet on the server-side. The file content is actually vaulted at this point, so if the upload is successful, the check-in will be trivial. Check-In just makes the object iterations visible to others and removes the exclusive lock.
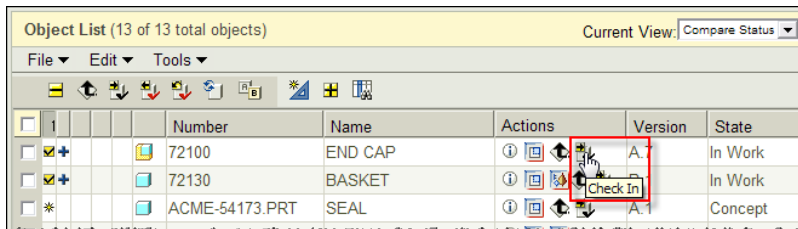
Select **Check In** from the in-line actions for the top-level assembly - it will automatically process dependent objects. You can also explicitly pick all items with status, and use **Check In** from the tool bar. **Design changes integrated successfully!**

Portable Workspaces (introduced in Windchill 8.0 + Wildfire 3.0)

As an alternative to exporting data from the workspace, **portable workspaces** provide a means to share both file content AND metadata. Data is exported as a package and can be re-imported to accurately track and control changes. Please review this old but good article on the Pro/INTRALINK Advisor site:

http://www.ptc.com/community/prointralink8/blogs/gerdes/gerdes_13.htm

Additional Enhanced Capabilities to Explore

- Workspace Save As; with Replace & Update Parent
- Workspace Frames

Good Designing!

Back To Top

Tips of the Month

Windchill PartsLink: Creating your own Parts Catalogue

Click Here To View

Back To Top

Announcements

PTC Tips & Techniques Newsletter Archives

Did you miss an issue? Can't find that awesome technique you read about? Fear not, you can click on the link below and go through our Customer PTC E-Newsletter archives.

Customer Care Zone

PTC Tips & Techniques Webcasts: Work Smarter. Not Harder.

Click below to see regularly scheduled Tips & Techniques technical Webcasts that are designed to provide you with the most popular time-saving tricks that Pro/ENGINEER users of all skill levels will find useful. Get more out of your maintenance dollars!

Tips & Techniques: Work Smarter Not Harder!

Special Hardware offers for PTC Customers

- http://www.hp.com/go/ptc
- http://www.hp.com/go/ptcworkstation

PTC Sponsored Events

- http://www.ptc.com/company/news/events/index.htm

Explore what is new with the Pro/ENGINEER Wildfire family!

http://www.ptc.com/go/showcase

Connect with PTC using the latest Social Networking resources:

Also visit http://social-product-development.blogspot.com/

E-PROFILES IS HERE!!
We have been eagerly anticipating the debut of the new electronic version of Profiles Magazine and now it is here! This new web site will supplement the print edition of the magazine and will provide new useful features not feasible with paper media. e-Profiles will provide you with 24x7, worldwide access to key information previously available exclusively in the print version. "Tips & Tricks," a popular feature pioneered by Pro/USER, has also moved to the web and will be expanded as the site matures.

Please take a few minutes to check out this new web site. We don't think you will be disappointed.

http://profilesmagazine.com/

Back To Top

Upcoming Events & Training Class Schedules

Upcoming, 2009   Your local Pro/Engineer User Groups
                                         http://www.ptcuser.org/rugs/

June 7 – 10, 2009   Orlando, FL USA
                                          PTC/USER World Event
                                          http://www.ptcuser.org/

Events

Our seminars and conferences seek to provide you with relevant information regarding product development trends in your industry as well as innovative software learning experiences. Think of them as a constructive day off where you can share experiences and swap ideas with your peers.

If you can't manage to get away, we'll bring it to you. Check back often for regularly scheduled live webcast events.

You're Invited to Attend…

Please visit the PTC Education Services website for the latest training information including course descriptions, schedules, locations, and pricing.

- Attend a course at any PTC Center and receive a free copy of Pro/ENGINEER Wildfire Student Edition!

                              http://www.ptc.com/services/edserv/index.htm

Live Instructor-Lead Virtual PTC Training Courses

Virtual Classrooms provide interactive learning with a trained PTC instructor in convenient and manageable sessions that last approximately 4 hours over a series of days. It's easy to join a class right from your desk using a phone or voice-over IP technology.

Sessions are performed just like a traditional ILT (including interactive exercises where you and the instructor can work on lab exercises together) and feature some of our most popular ILT courses. These sessions cover the exact same material as the traditional ILT in-center courses. Also look for some of our most frequently requested mini-topics delivered in the same format that are only an hour - two hours in duration.

If you have any questions about these sessions or would like to see getting other courses, not on this list, on the schedule please feel free to contact me for more details. They are a great way to bring training to you without you having to worry about location or being out from work for long stretches.

You can register for these sessions just as you would for any normal ILT class either by:

1. calling order admin at http://www.ptc.com/services/edserv/training/registra.htm or

2. you can go to PTC University directly at http://www.ptc.com/learning and submit a registration request directly. All you have to do is search the catalog by typing in "virtual" in the search field and you will see a listing.

PTC

Note: This PTC E-Newsletter will continue to be used for the following:

1) Inform you on events related to PTC products (user groups, conferences, training schedules, etc.)

2) Educate you on solutions that are available at PTC

3) Tips & Techniques using PTC Products

Note: These messages are compiled in the local PTC office and will be distributed via e-mail.

Back To Top