PTC Technical Specialists E-Newsletter
Date: September 2009

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PTC Product Focus

Using the Global Reference Viewer

There are numerous factors that contribute to the complexity of assembly design and maintenance, including, but certainly not limited to the following: incorporation of legacy data, mergers and acquisitions, vendor and supplier model quality, and turnover in the CAD operator workforce. Add all of the previous items with the constant pressure to make products faster and with higher quality, and there is a perfect opportunity to rush the design process and force assemblies together. While it's possible to make a finished product with all of these external influences, there can be unintended consequences which can cause Pro/E performance issues and data management nightmares.

With the release of Wildfire 4.0, there have been significant strides made in the area of resolving the modeling issues that negatively impact assembly performance; mainly external and circular references.

Accessing the Reference Viewer

The Global Reference Viewer has been around for many releases, but interrogating the available information, and actually fixing the assembly issues, was never quite as easy as Wildfire 4.0. To launch the Reference Viewer in Wildfire 4.0, simply select the component or subassembly of interest, click the **Right Mouse Button (RMB) > Info > Reference Viewer.**
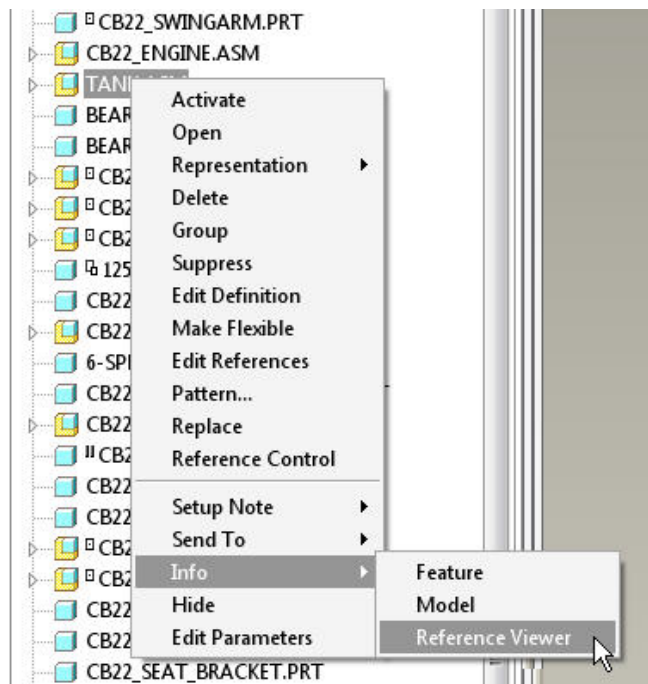
Figure 1: Accessing the Reference Viewer

This will open the Reference Viewer Dialogue Box. At first glance, there are several differences between the new viewer and the viewers available in previous releases of Pro/ENGINEER. The new viewer is a much more visual user interface, with more options for navigation and interpretation of the Pro/E model.
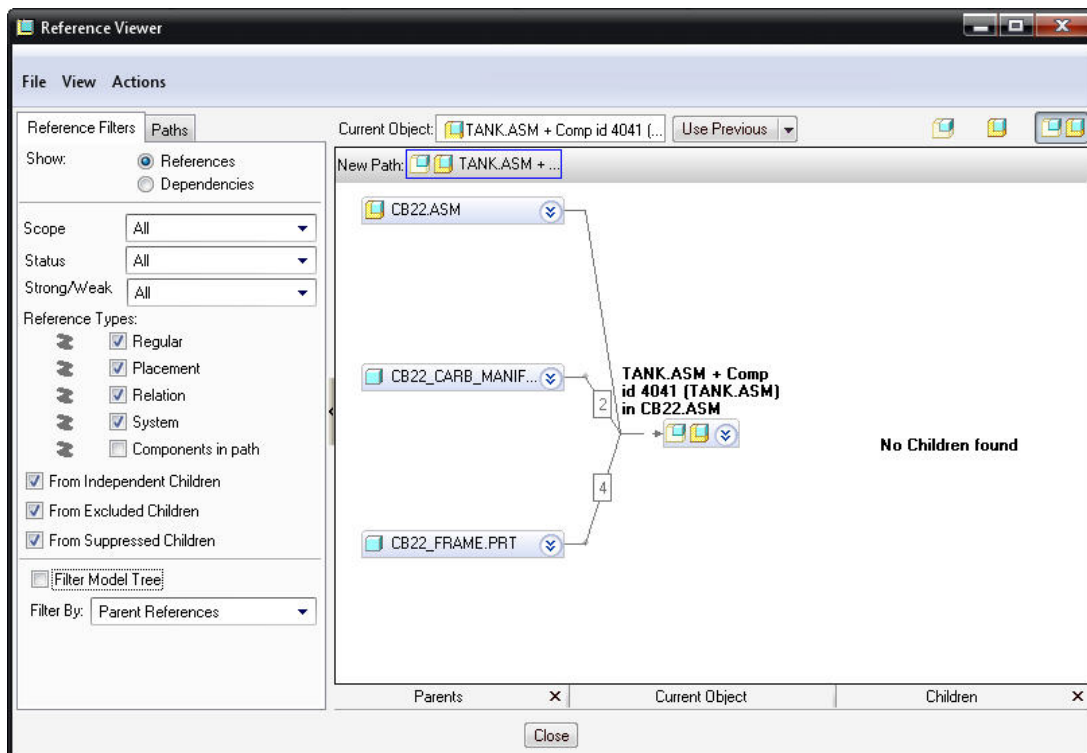

Figure 2: The Reference Viewer Dialogue Box

This new layout, shown in Figure 2, shows any existing parent child relationships in a flow-chart style format. This makes tracing assembly structures very quick and efficient. To see the actual references being used in the Parent/Child relationships, simply click on the Arrow Icon ⊗ to expand the subitems window in the Parent object.
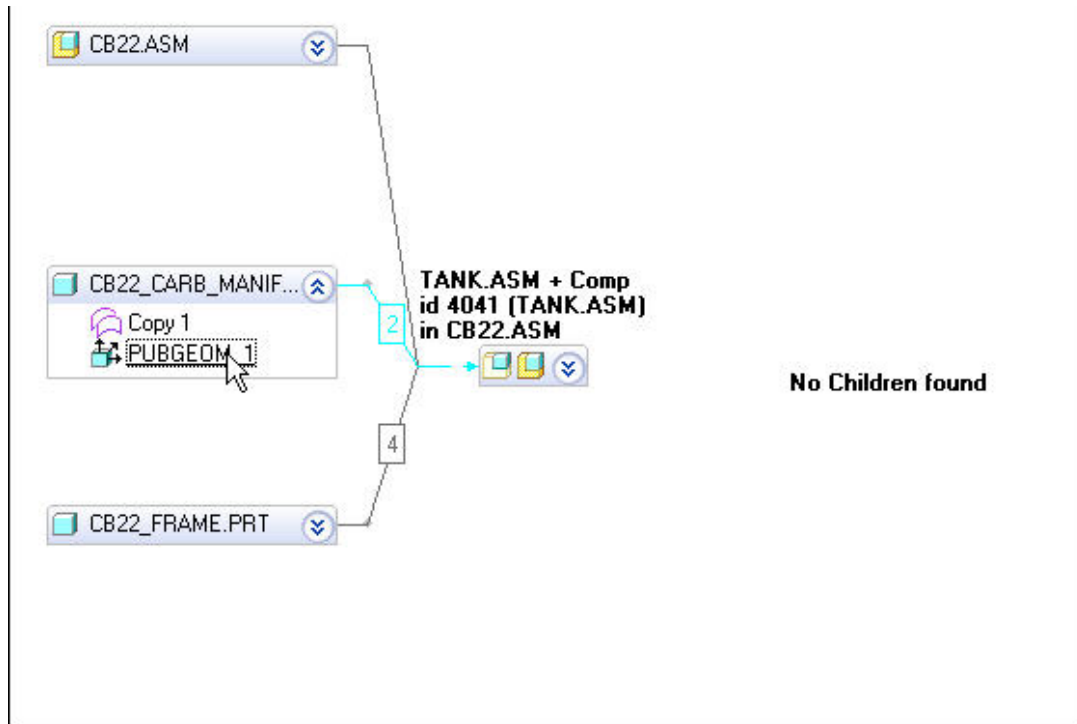

Figure 3: Expanding the Subitem Window

The reference path will highlight in blue, and any components referenced will highlight in the Pro/E graphics window. This is the basic approach to interrogating the assembly.

Filtering the Reference Display

Once inside the Reference Viewer, it's time to start interrogating the reference and dependency information. There are several enhancements made to the reference viewer to assist in this task. Starting on the left-hand side of the dialogue box, there are several options to configure the display of information.

Figure 4: Reference Filters

Changing the shown relationships (References or Dependencies), Scope, Status, Reference Types, etc… will dynamically filter the displayed information. Changing the colors for each of the reference types is also recommended. This makes it possible to narrow the focus of the Reference Viewer search to the exact assembly issue under investigation. When used in conjunction with the Pro/ENGINEER Message Log and/or ModelCHECK, it is possible to throubleshoot assembly errors faster than ever before.

Troubleshooting Circular and External References

The two most common headaches encountered with assemblies are external references and circular references. To find any circular references in an assembly, simply utilize the "Paths" tab from the reference viewer. The Paths tab will display two key pieces of information. The path of the specified objects (user defined), and circular paths if they exist. In the event that a circular path is found, the next step is to identify the depth of the circular reference (single or multi-level).

A single level circular reference will show up as shown in Figure 5 when you select it from the reference viewer.
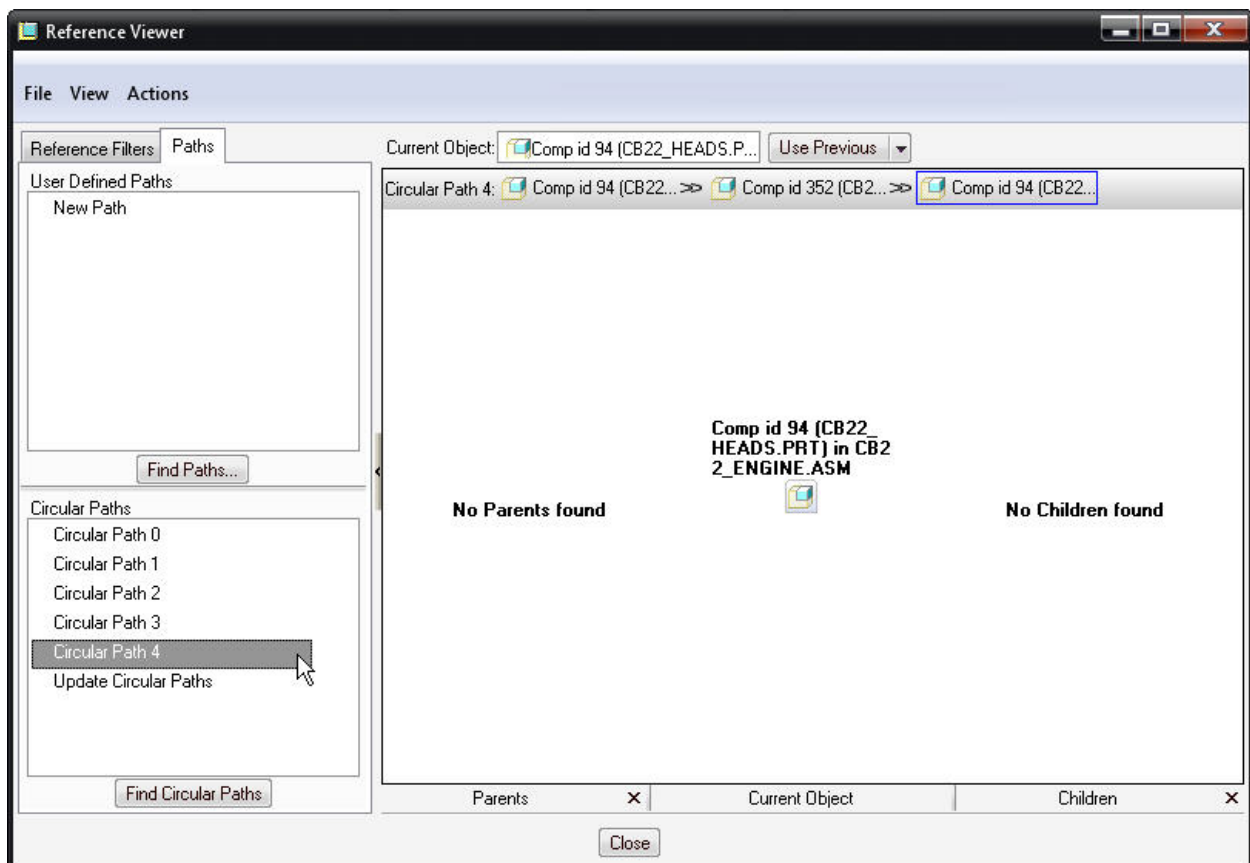
Figure 5: A Single Level Circular Reference

Note the explicit display of the circular path at the top of the current object window. In this particular case, Comp id 94 is referencing Comp id 352, which is in turn referencing Comp id 94 again. These are the easiest type of circular reference to fix since they can be corrected by reordering objects in the model tree. In this example, dragging Comp id 94 above Comp id 352 will eliminate the circular reference.

When Parent/Child relationships are involved, the circular references become more complex. These create multi-level circular references which are often combined with external references. These require a little bit more effort to solve, but the Reference Viewer makes the task much easier than it may have been in the past. A multi-level circular reference will be displayed as shown in Figure 6.
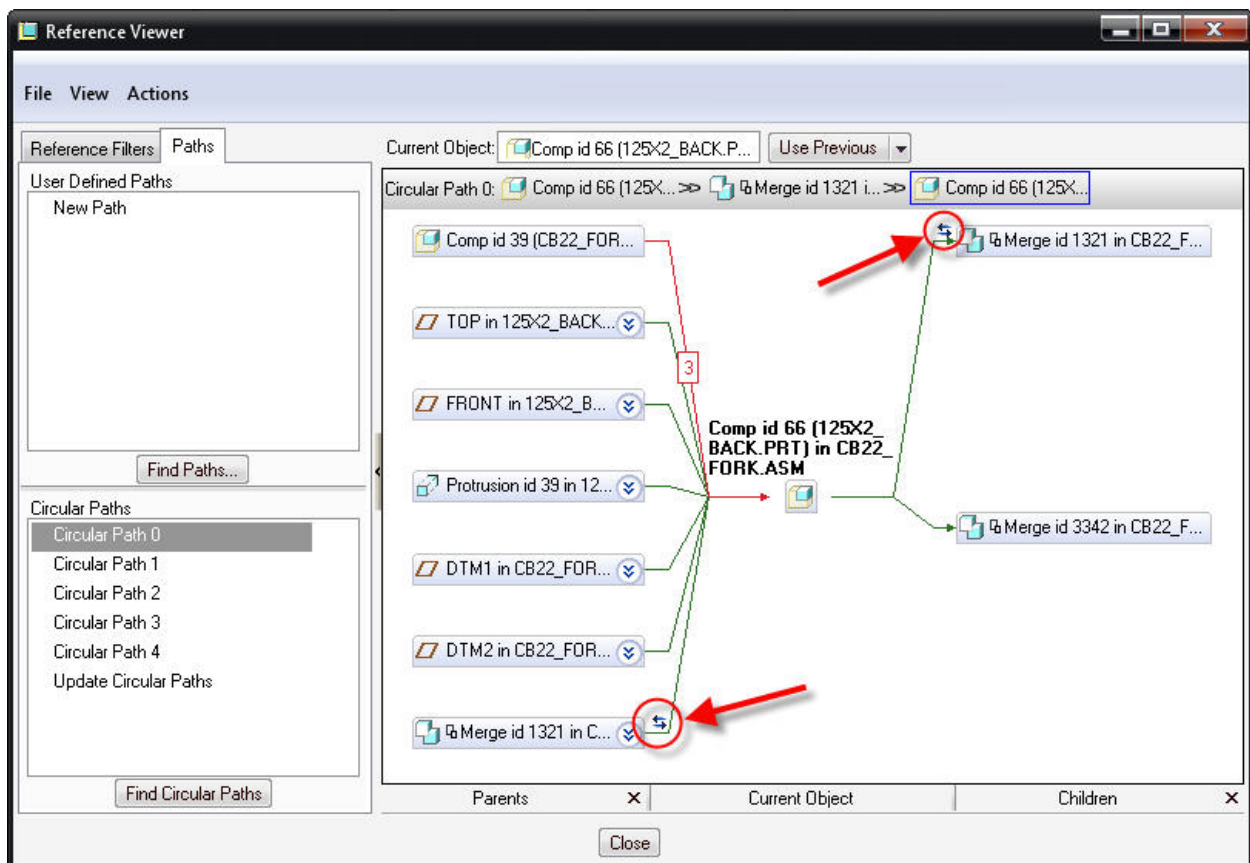
Figure 6: A Multi-Level Circular Reference

There are clear indications of the offending components or features causing the circular reference which makes fixing the issue much easier. In this example, an axis from a child object is being used to place a parent object. Simply redefining this reference will fix the circular reference.

NOTE: In Wildfire 4.0, the new Active Component Display Option makes it easier to avoid creating external and circular references since the eligible references are clearly displayed while the other objects are grayed out.

An issue that plagues the data management aspect of working with assemblies is the existence of external references and dependencies. This type of issue can cause exceptionally large workspaces when only a small component or subassembly is required. To find an external reference, simply change the filter criteria in the Reference Viewer as shown in Figure 4.

Breaking Model Dependencies

When external or missing model dependencies are found, it may be necessary to break that relationship in order to solve the assembly or data management problem. This can now be done from directly inside the Reference Viewer. Simply change the filters of the reference viewer to show missing dependencies as shown in Figure 7. Once the filter is in place, select the component reference, and click **RMB > Break Dependencies.**
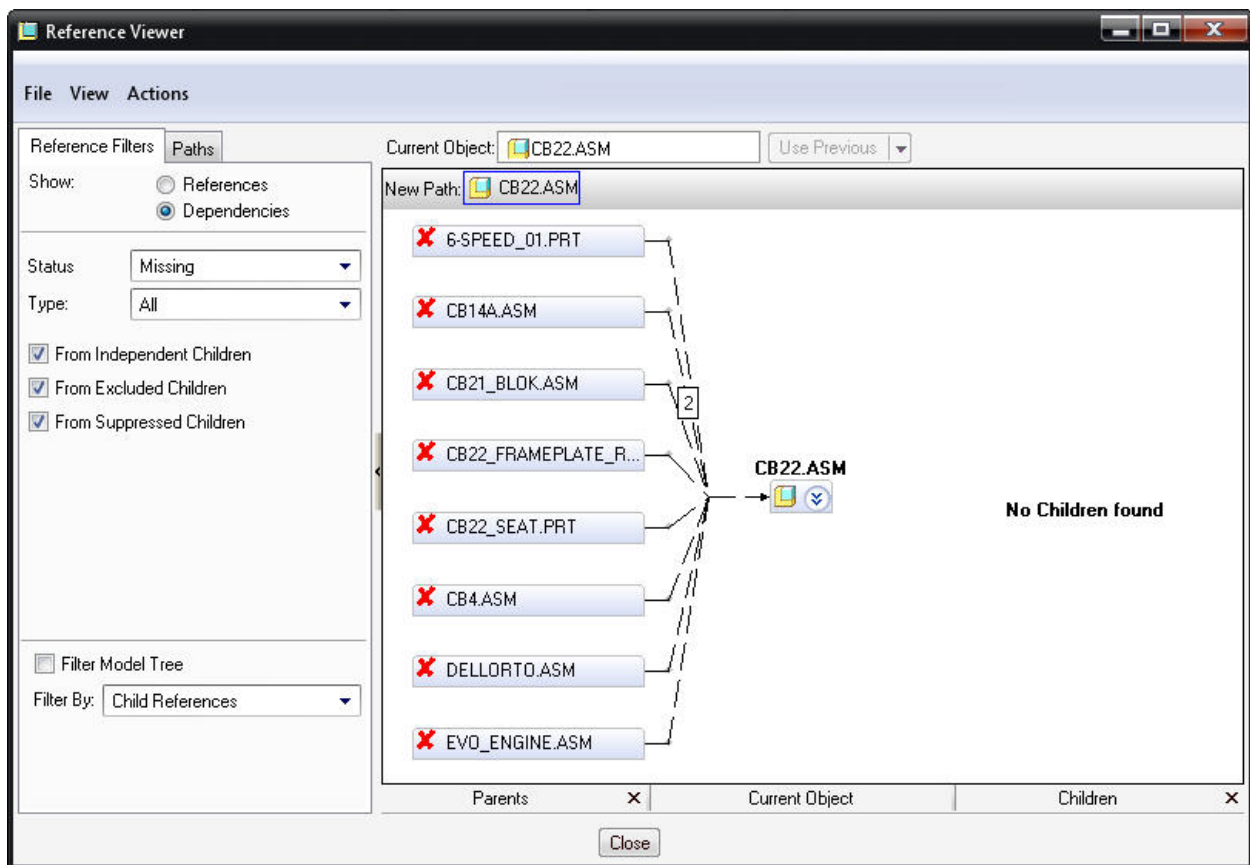

Figure 7: Breaking Dependencies

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PTC Product Focus

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Windchill Workflow Delivers Adobe Forms

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Tips of the Month

Creating a Volume Analysis Feature

There are many types of analysis features available in Pro/ENGINEER, with the most commonly used analysis type being the model Mass Properties. One of the more commonly overlooked analysis types is the One-Sided Volume Analysis. This is a great way to leverage the associative properties of Pro/ENGINEER for what may seem like an otherwise obscure task; calculating volume. While the Mass Properties can give you the overall volume of a model, the One-Sided Volume analysis takes things one step further, and provides a more practical analysis result, the usable volume of a container. The following will outline how to quickly setup and make use of this type of analysis feature. (This works best with shelled geometry)…

1. Create a Datum Plane feature at a location you would like to analyze the volume from. For example, just below the fluid inlet for the container.
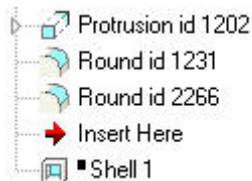2. If you have already "shelled" the model; move the ➜ Insert Here icon above the shelled feature

Figure 1: Model Tree

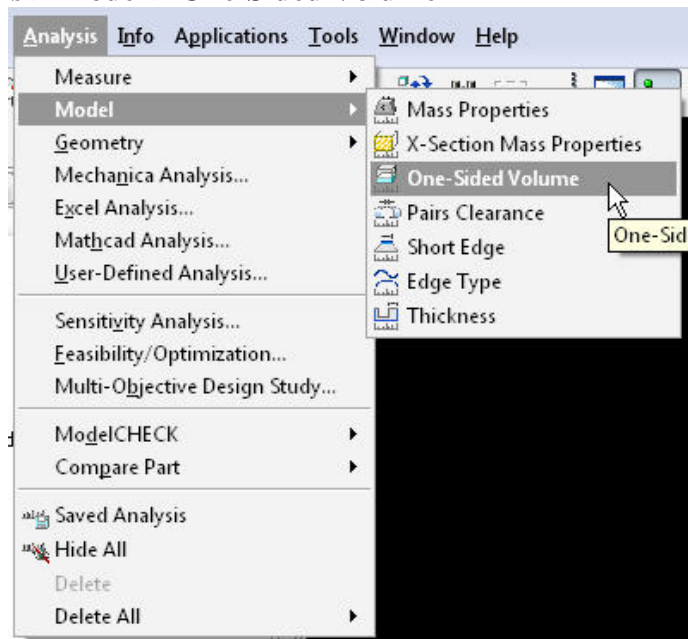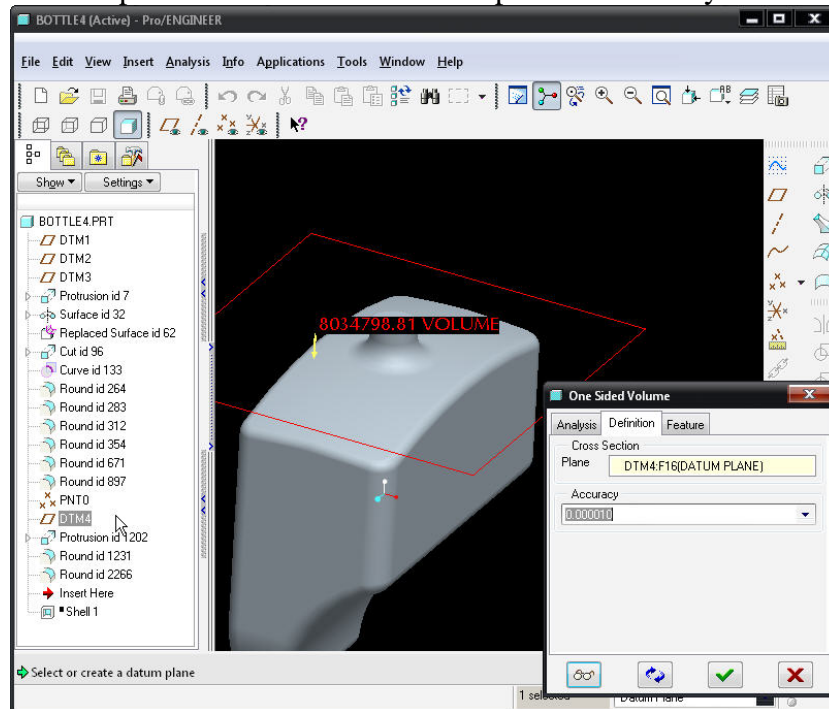3. Select **Analysis > Model > One-Sided Volume**

Figure 2: Selecting the Analysis Feature

4. Select the datum plane that defines the cut-off point of the analysis

5. Complete the analysis feature to add it to the model tree

6. Cancel the ➜ Insert Here operation

NOTE Be sure to check the direction of your existing shell feature (shelled to the inside/outside) as changing this direction will change the mass properties and volume properties of the model…

Now the useable volume of the container can be leveraged for design studies. Changing the depth of the datum plane used in the analysis will allow testing of various geometry changes and the impact on the fluid level at certain intervals (quarter full, half full, etc…)

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Tips of the Month

Separate Access Control Rules for Content and Metadata

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Announcements

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[Customer Care Zone](#)

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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!](#)

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

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- 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:

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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/

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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.

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