



PTC Product Focus: A) [What's New in Assembly Mode for Wildfire 4.0](#)

B) [What's New in Windchill 9.0 Replication](#)

Tips of the Month: A) [Great New Wildfire 4.0 Sketcher Functionality Grab-bag](#)

B) [Java Monitoring Tip](#)

Announcements: [Most Recent Announcements](#)

Upcoming Events & Training Schedule: [Events & Training Schedule](#)

PTC Product Focus

What's New in Assembly Mode for Wildfire 4.0

Pro/ENGINEER Wildfire 4.0 offers hundreds of enhancements that optimize four key global design processes and further improve personal and process productivity.

- **Accelerate your detailed design process.** Create both simple and complex designs faster with new AutoRound™ capabilities, improved assembly performance, direct surface editing, and automated 3D drawing annotations.
- **Improve your design outsourcing process.** Secure your valuable design data with new digital rights management capabilities.
- **Enhance your verification and validation process.** Analyze designs faster and easier with new tolerance analysis, improved meshing, support for nonlinear materials, better results analyses, and smarter diagnostics.
- **Optimize your manufacturing tooling and factory equipment design process.** Simplify and automate the transformation of engineering designs into manufacturing processes with the easy to use, powerful process manager for toolpath definition, annotation features, and other key capabilities in Pro/ENGINEER.

This article will focus on only some of the additional new functionality that Wildfire 4.0 adds to the Assembly environment within the software. For more detailed coverage of the enhancements in Pro/ENGINEER Wildfire 4.0, please review the links and additional material at the end of this article.

Let's begin by talking about assembly mode enhancements that everyone using Pro/ENGINEER will see and access.

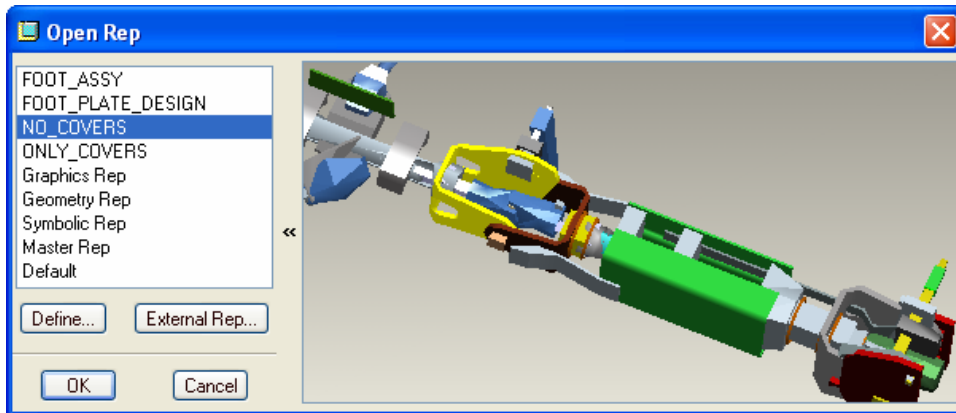
Automatic Memory Management with All Assemblies

The default working mode for Pro/ENGINEER Assemblies now includes automatic, on-demand retrieval or removal of models during selection, manipulation of components, and regeneration. *This means that up to 40% less memory is consumed when working with large assemblies!* If a component is unchanged, the component is automatically downgraded to a lower-level representation when an action is performed. You can easily change the default retrieval or removal for specific actions.

Graphics Only Representation ~25%	Geometry Only Representation ~50%	Complete Master Representation 100%
Single Pro/ENGINEER Part Database		

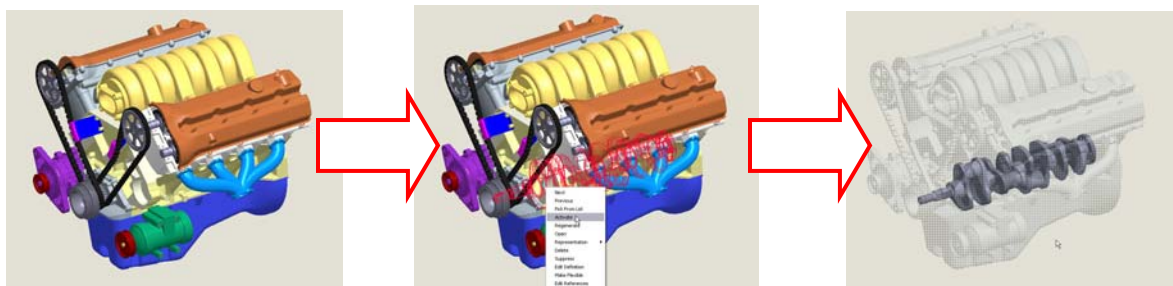
Simplified Rep Preview

You can preview a simplified representation before you open it. Preview simplification definitions in the Open Rep dialog box and preview external simplified representations.



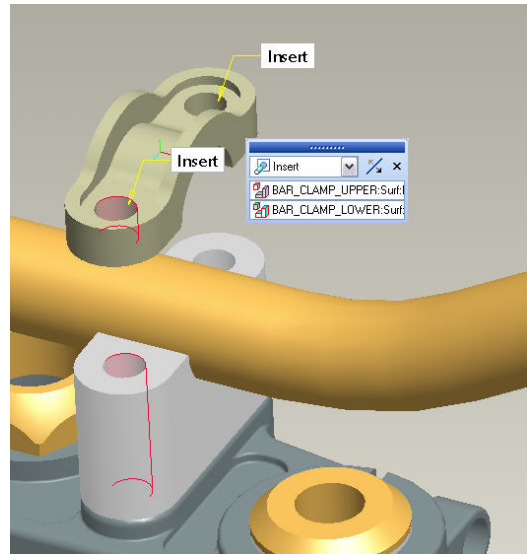
Clearer indication of “Active” Component

The active model is accentuated because the inactive models appear gray and transparent. As a result you can easily select model entities and minimize unintended model changes. Use the configuration option: dim_inactive_components to turn off or turn on the dimming of inactive components. The default value = Yes.



Improved Assembly Process Progression

A new assembly process option minimizes the need to access the dash board panel. Set the configuration option: `comp_assemble_start = move_then_place`. The component being assembled now attaches to the pointer. Move the component to the required location, click, and select references to place the component. A preview color for a partially constrained model and a color for a fully constrained model can be set. Reference handles appear indicating the need to choose a matching reference. Color coded constraint tags indicate status. The tool-tips of the constraint tags show references. Double-click the constraint tag or right-click the tag and choose Edit Reference.

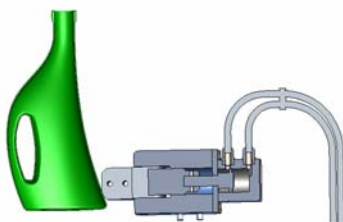
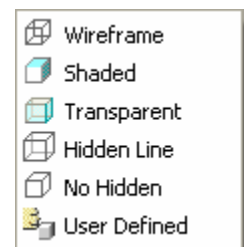


Improved Family Table Instance Retrieval Performance

In the past, when retrieving an assembly family table instance, Pro/ENGINEER retrieves AND regenerates the generic instance and all nested instances prior to retrieval of the selected instance, causing retrieval times to be long. Also, Instance Accelerator files become out of date very easily and make managing and using them difficult. But in WF4, the user now has the option to retrieve only the instance itself, without retrieving OR regenerating the upper level instances or the generic. If the Instance has a dependency on additional models in the generic, the system will retrieve the model(s) on demand prior to the instance regeneration.

Transparency is now a Display Style not just a color setting

This one button command from the top level View menu will apply a default transparency to all selected components at any assembly level. It will apply to components the same way the other settings (Wire Frame, Hidden Line, No Hidden, and Shading) work. All currently assigned colors will be retained but will have 75 transparency assigned. This command will only work if Transparency is checked off under the Shaded tab of the Model Display dialog. The setting will not be stored unless a Style state is stored using the View Manager.



Cross-sections display states can be saved with the model

A new `config.pro` option saves the Cross-section clipping state of a model when you save the model. When retrieved, the model appears with the cross-section view. The `config.pro` option is: `save_clipped_view = yes`. Please note that this does not effect *Visibilities > Clip* functionality.

Visual Basic API

A new Parametric Foundation Classes (PFC) based Visual Basic API is available in Wildfire 4.0. Users versed in Visual Basic programming can use this new API to automate and customize Pro/ENGINEER from Visual Basic.NET applications and Visual Basic macros in applications like Microsoft Word, Excel or Access. This API contains the same capabilities as asynchronous

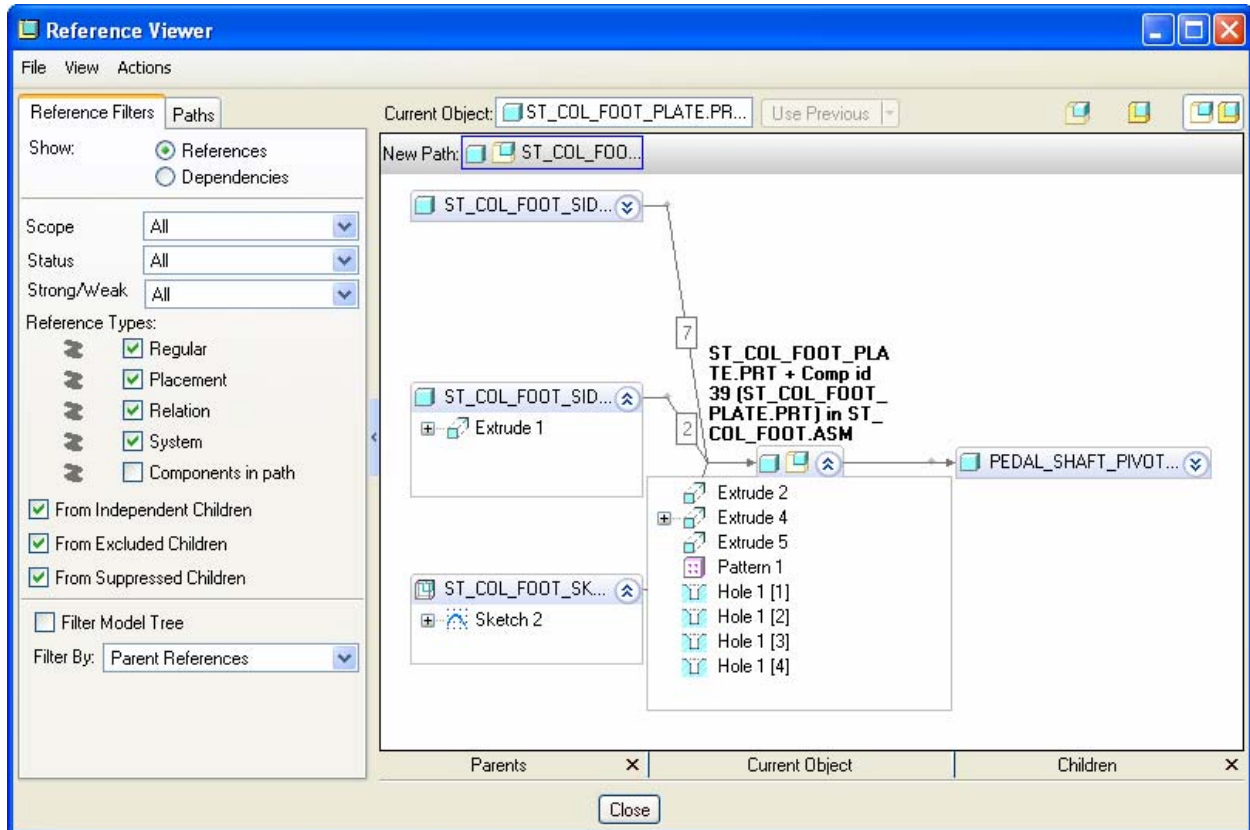
J-Link (core Pro/ENGINEER part, assembly, and drawing manipulation; Windchill server access; data exchange; Pro/ENGINEER user interface customization and applications triggered by events occurring in the Pro/ENGINEER session.) It is compatible with Visual Basic .NET 2005 and Visual Basic for Applications. Also please note that it requires no license beyond the license to run Pro/ENGINEER. See more information in PDF form by referencing: **[Pro/E WF4 load-point]\vbapi\vbug.pdf**

Now let's talk about a few of those things that customers who leverage the power of the Advanced Assembly Extension (AAX) get access to with Wildfire 4.0.

New Reference Viewer

The Reference Viewer has a great new consolidated user interface with expanded functionality. The Parent/Child and Global Reference Viewer tools are combined to share one user interface. You can:

- Quickly investigate a model.
- Follow and detect a reference path.
- View the full reference path.
- Easily find all objects with external references.
- Graphic Display of Circular References.
- View data management type dependencies.
- Highlight references and dependencies in the graphics area.
- AND VERY IMPORTANTLY: Break non-required dependencies that cause data management ghost objects before you submit to your data management system.

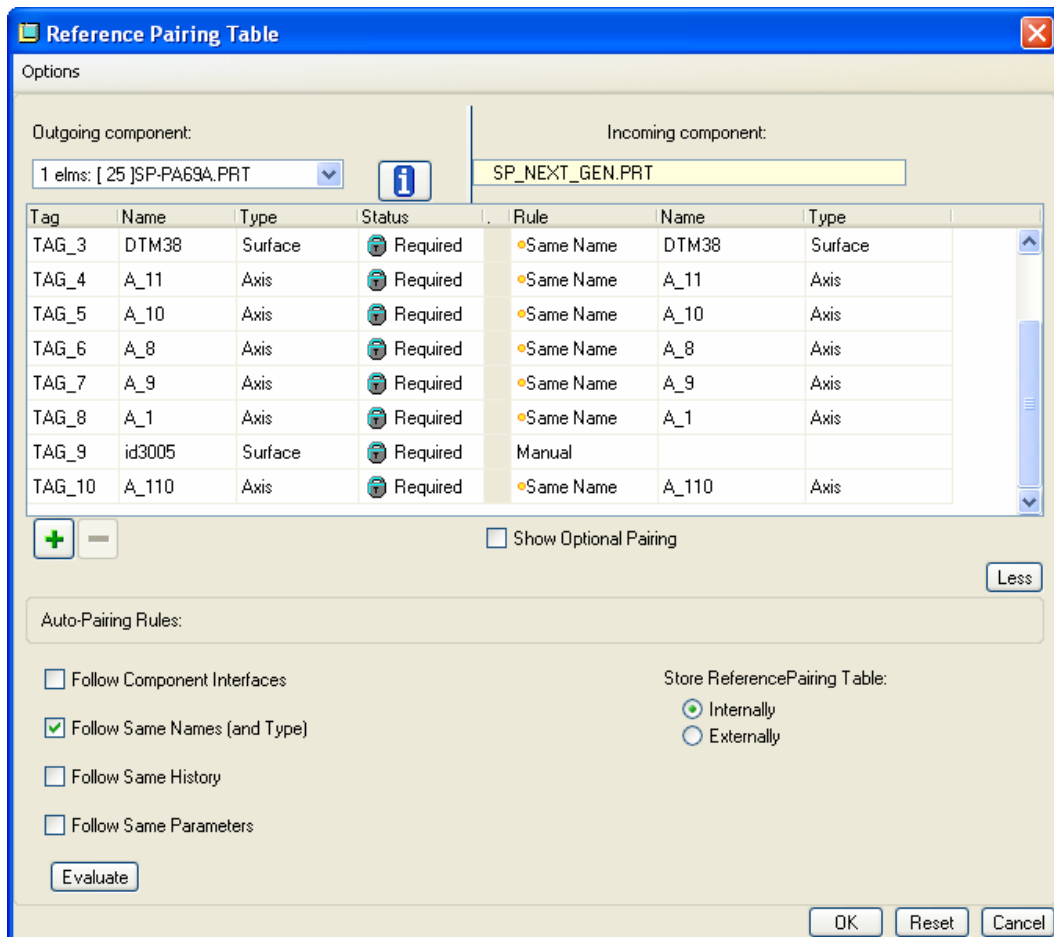


Assembly Process Planning Improvement

In Assembly Process Planning, also referred to as Pro/PROCESS for Assemblies, you can start the process planning sequence with a fully assembled design model. Creating disassembly instructions no longer requires a first assembly step. You can now choose to start with a fully assembled design model.

On-the-Fly Replace of Unrelated Components

You can now replace a component with an unrelated part or subassembly without the work of creating an interchange assembly. Pro/ENGINEER provides the tools to map the references required from both objects to ensure that there is no feature failure due to missing references. You can replace a part or assembly component at any assembly level and the target component and all its children are automatically replaced. From the Replace dialog box, you can map references from one object to the other. A table walks you through each reference tag by highlighting and color-coding the target and source references. This mapping table is also stored in the assembly so that you can automatically replace multiple occurrences of the same model. Advanced and more granular reference definitions are also available.



More Information on Pro/ENGINEER Wildfire 4.0

[What's New in Pro/ENGINEER](#) - Find out what's new in the latest release of Pro/ENGINEER.

[Pro/ENGINEER Wildfire 4.0 What's New - English](#) – PDF Documentation of the release.

[Pro/ENGINEER Resource Center](#) - Learn Pro/ENGINEER with the latest Pro/ENGINEER release essentials, alerts, menu mappers, tools & tutorials and more.

[What's New in Pro/ENGINEER Wildfire 4.0 Tutorial](#) - Pro/ENGINEER Wildfire 4.0 workshop for experienced users!

[Back To Top](#)

PTC Product Focus

What's New in Windchill 9.0 Replication

[Click Here To View](#)

[Back To Top](#)

Tips of the Month

Great New Wildfire 4.0 Sketcher Functionality Grab-bag

Sketcher Diagnostics

Have you ever created a Pro/ENGINEER sketch and had the system tell you something was wrong, but you did not know what it was? Well be confused no more! Four new diagnostic tools have been added to sketcher for improved debug/analysis of common sketching problems.



Shading closed loops – The area inside entities that form a closed loop is shaded with a predefined color. This clearly shows if you have a closed section or not.



Highlight open ends – The endpoints of entities that are not common to more than one entity are highlighted. This lets you easily find where those unconnected ends are located.



Overlapping geometry – Overlapping entities are highlighted. Highlight the problem entities for easy cleanup.



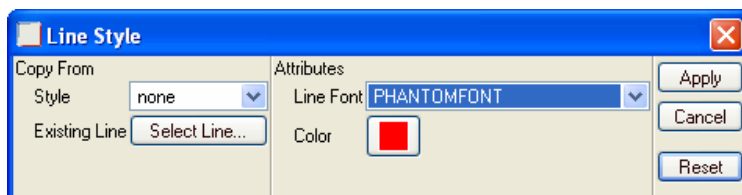
Feature requirements – Provides a report telling the user whether the sketch meets the requirements for the feature in which it is embedded. This is available/necessary in 3D sketcher only and will remind you to create that axis in a revolved section.

Intent Objects as Sketcher References

Are you using Intent References throughout your design and making extremely robust models? Well Wildfire 4.0 is your type of release! as Intent objects are supported as sketcher references. You can select composite curves and intent chains as sketcher references. As a result there are far more complex and robust relationships between features. You can also select intent chains and composite curves for Use-Edge and Offset-Edge operations. Intent chain modifications are automatically updated in the sketch. (If you don't know about Intent References, get going and start learning about them! Ask around! You are missing out!)

Line Style and Color for Sketched Entities

You can assign a line color and style to sketched entities by right-clicking the entity and selecting Properties. These settings remain with the completed sketch. For some entities you can preset line style and color from the System Colors dialog box.



BONUS ITEM: Exact Expressions in Wildfire 4.0

Have you ever ran into a rounding error for a calculated value typed into Pro/E due to the number of decimal places specified on a dimension? Well, never run into that again!

You can now use the syntax =() to force Pro/ENGINEER WF4 to calculate the precise value of the expression without the need for a relation.

In the Dimension Property dialog box, you can set the expression: =(expression). This expression takes the result of a calculation in the parentheses and displays the exact result on the dimension field. This exact result has the minimum number of decimal points needed to display the dimension in its exact value or to the number of decimal places defined by the configuration option default_dec_places.

Exact values override the number of decimal places currently set globally or locally for the dimension.

For example

Create a part with a simple protrusion

In a dimension field type: =(360/29)

The value read is 12.413793103448 and the number of digits is set to 12.

[To see what would have happened without the =(), go to the dimension's properties, set the number of decimal places back to 3 and try entering only: 360/29. See the difference!]

NOTE: In 3D, the dimension appears with trailing zeros sufficient to show the number of decimal places defined in the dimension properties. In 2D, the display of trailing zeros is controlled by the Detail Setup option, lead_trail_zeros.

[Back To Top](#)

Tips of the Month

Java Monitoring Tip

[Click Here To View](#)

[Back To Top](#)

Announcements

Educational Resource Library

Learn things you always wanted to do - but didn't know you could.

This one stop educational resource library will help you learn more about PTC Solutions and provide you with technical materials developed by the product experts to help you become more productive.

Get tutorials, how-to videos and expert advice for:

- Pro/ENGINEER
 - Conceptual and Industrial Design
 - Detailed Design
 - Simulation/Analysis
 - Production
 - Design Collaboration
- Windchill PDMLink
- Windchill ProjectLink
- Pro/INTRALINK
- PTC Online Tools

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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!](#)

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, 2008 Your local Pro/Engineer User Groups
<http://www.ptcuser.org/rugs/>

June 1 – 4, 2008 Long Beach, CA 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](#)