



new!

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new!

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## PTC Product Focus

### Pro/ENGINEER Wildfire 2.0, Why Upgrade?

This list is subjective. There are a lot of great enhancements to Pro/ENGINEER Wildfire 2.0. This list is our top ten, however all of the details on enhancements for this release can be found in the [Pro/Engineer Wildfire 2.0 Customer Portal](#).

#### Reason #1: Global Undo



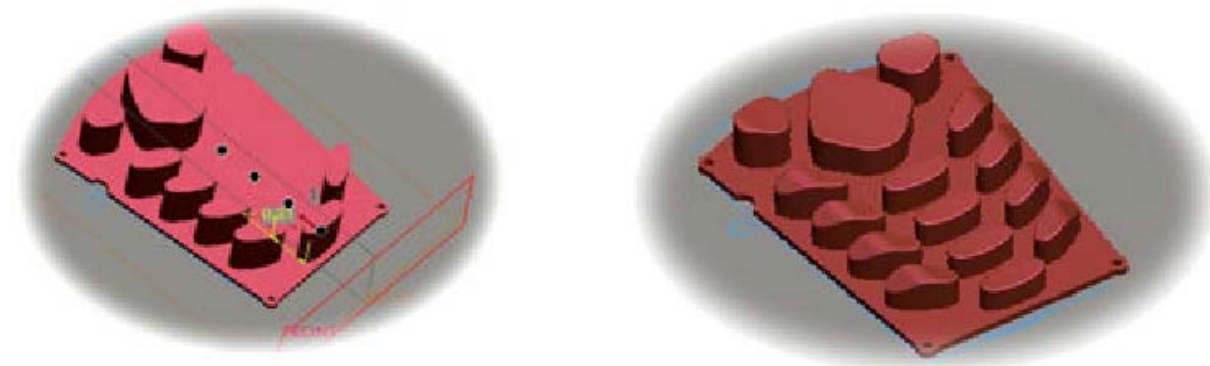
Finally!!! With the Undo and Redo commands it is easy to revert to a state prior to the last action. These capabilities have been added for a vast number of operations (Part, Drawing and Assembly mode) you can perform these actions using shortcut keys, icons, or commands. Access by selecting: **Edit, Undo or Edit, Redo**. Shortcut keys CTRL+Z and CTRL+Y

#### Reason #2: Copy and Paste



It is now possible to copy and paste features to other parts of the model. Use the familiar **Copy** (CTRL+C) and **Paste** (CTRL+V) commands and key combinations in Windows applications. Interact during the paste operation with the dashboard and see an immediate preview of the copied features. Access by selecting: **Edit, Copy (CTRL+C) and Edit, Paste (CTRL+V)**

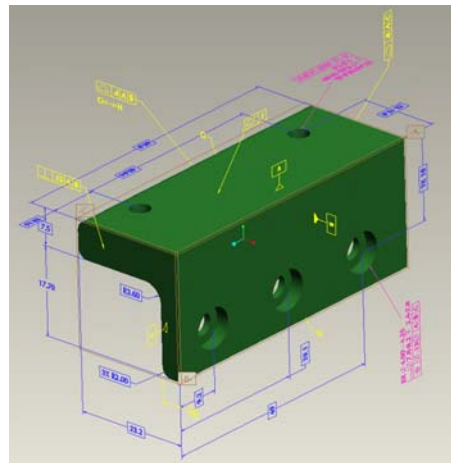
### Reason #3: Directional and Axial Patterns



It is now possible to pattern features around an axis or in any direction, regardless of how the original feature was created. Previously, a Make Datum feature was required for axial patterns to incorporate the angular dimension for patterning. It is also possible to pattern any feature in any direction without having to select any dimensions as defined in the original feature.

Access by selecting: **Edit, Pattern**

### Reason #4: Paperless Drawings



Pro/ENGINEER Wildfire 2.0 supports the recently released ASME Y14.41 standard and developing ISO 16792 standard that pertains to paperless drawings.

The time and expense of creating 2-D drawings has taken its toll on manufacturers. They are interested in capturing information for production and procurement within the 3D models.

Wildfire 2.0 does this with annotation features: (**Insert, Model Datum, Annotation**). These features can be controlled with layers and prioritized. The result is a fully detailed 3D model!

### Reason #5: AutobuildZ

AutobuildZ, the free plug-in application for moving to 3D is integrated into the base Pro/ENGINEER package.

AutobuildZ allows you to build 3D parametric, feature-based solid models from 2D drawing data. Feature creation wizards guide you through the selection of entities and references directly in the 2D drawing to create 3D Pro/ENGINEER features. The features supported include Extrusions, Revolves, Simple Holes, Datum Curves, Datum Points, and Datum Planes.

AutobuildZ is integrated into drawing mode. Using the simple but powerful suite of tools you can now directly reuse your legacy 2D data in a 3D context.

Access by selecting: **Tools, AutobuildZ** (available in Drawing mode only)

## **Reason #6: Resolve Mode**

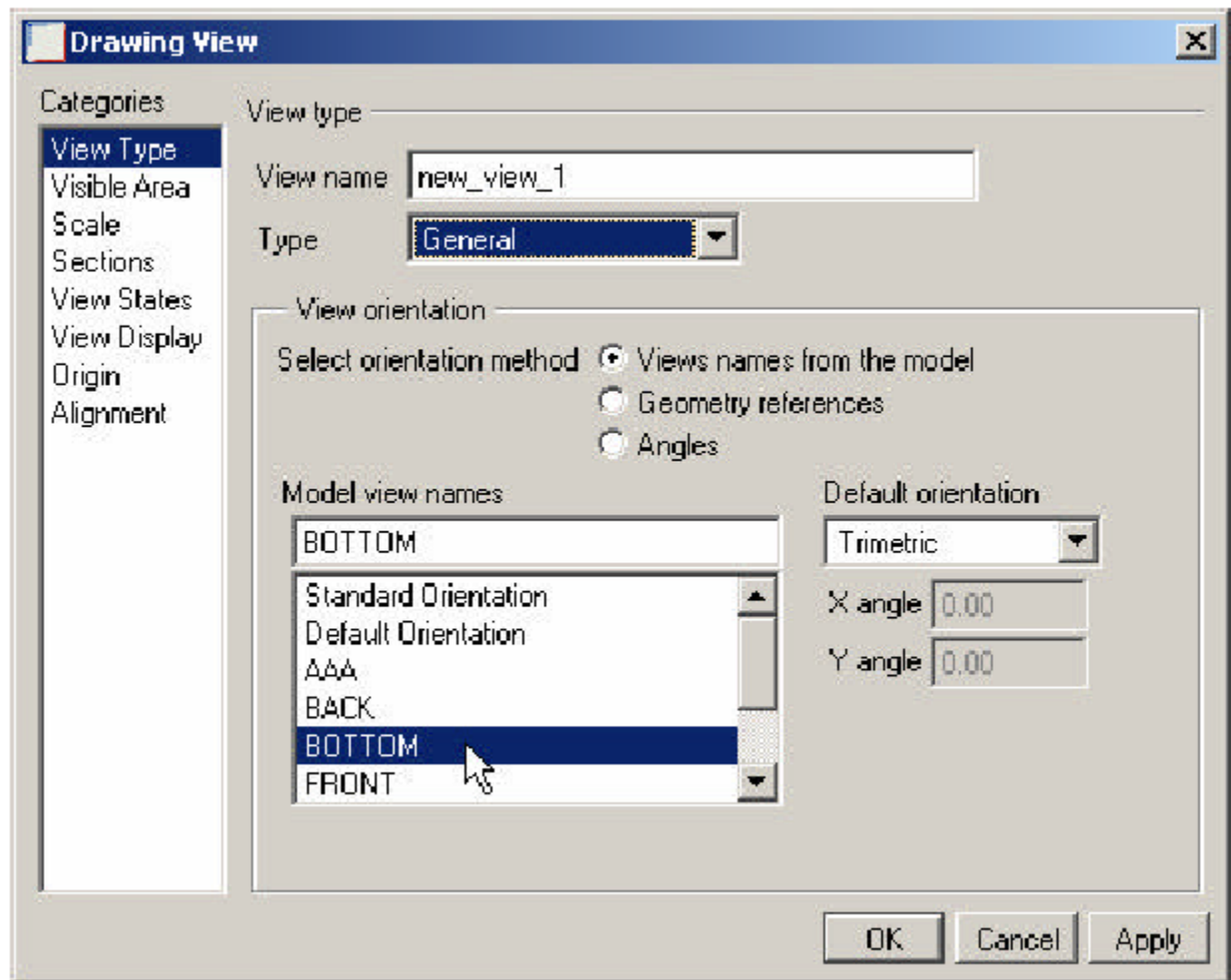
Resolve mode isn't as daunting any more with the new Troubleshooter Dialog box in Wildfire 2.0. With the Troubleshooter Dialog box you can quickly identify why and where failures have occurred, document the failures for future reference, and obtain a possible solution to correct the failure. The Troubleshooter also allows users to display the failure messages as model notes for documentation or future correction.

**Access by Selecting:** The troubleshooter appears after a feature failure or after you right-click a collector containing a red or yellow dot and choosing **what's wrong?** from the shortcut menu.

## **Reason #7: "Borrowing Licenses"**

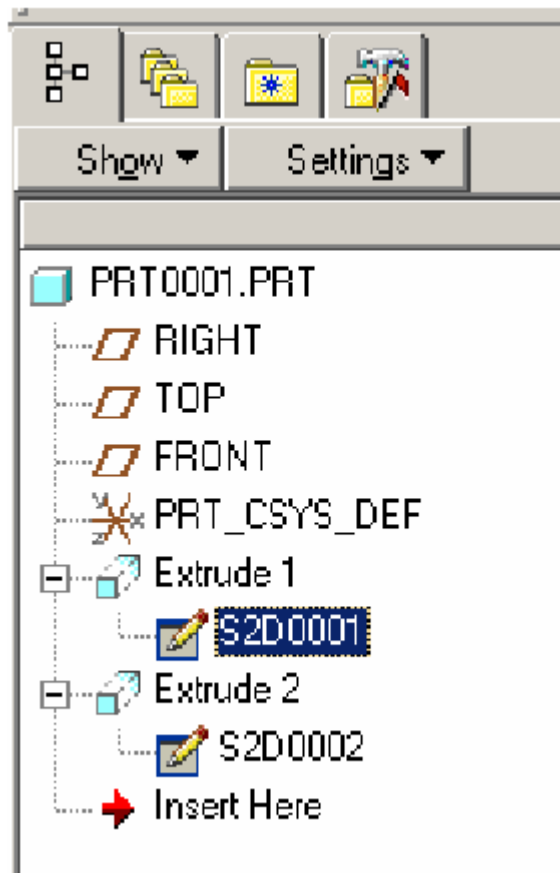
It is now possible to "borrow" licenses from your license server. Do this by "checking out" the desired license for a determined time, no longer than 5 days. This allows customers greater flexibility allowing users to work with Pro/ENGINEER on the road, at customer sites or at home.

## Reason #8: Drawing Mode Modernized



Drawing mode User Interface has been converted to the “Wildfire” icon-based workflow. New intuitive dialog boxes replace the menu manager for easier creation and manipulation of drawing views.

## Reason #9: Flexible Sketched Sections



Sketches in previous release of Pro/ENGINEER were either embedded in an Extrusion or completely separate with no associativity. Pro/ENGINEER Wildfire 2.0 now splits the section from the extrusion to enable the user to more easily manipulate the section. Associativity is in the hands of the user. This gives the user the ability to:

- Reference an existing sketch datum curve and to sketch inside the Sketched Based Feature
- Reference an external sketch as “Modified Copy Dependent ”
- Allow usage of any “Sketch Datum Curve ”
- Enable full break of dependency when an external sketch is used
- Add sketch control to the Model Tree Item Display dialog box
- Use sketches from different models more easily

## Reason #10: Real-time dragging of packaged components in Assembly Mode

Design assemblies no longer require a skeleton or the reconstruction of assemblies in Mechanism to evaluate mechanism behavior. Basic dragging allows full interrogation of the assembly through its full range of motion based on the degrees of freedom in the placement constraint. For a component with no placement constraint you can move it into the vicinity of the assembly for placement. This capability vastly improves the workflow and usability of assemblies and

mechanisms. Access through: During Component Placement hold down CTRL+ALT and middle-click

### **Additional information:**

PTC has a lot of new information to help your migration. Below are a couple of good links to get started.

Online web tools: <http://www.ptc.com/community/proewf2/newtools/index.htm>

Online Tutorials: <http://www.ptc.com/go/wildfire/get/index.htm>

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## **PTC Product Focus**

### **ProductView Composer Option**

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## Tips of the Week

### Understanding the Search Tool in Pro/ENGINEER Wildfire 2.0

Did you know that you can search through a part or assembly for a specific feature or component? Did you know that you can use the same search tool to find specific features or components whenever Pro/ENGINEER prompts you for a selection? The search tool is just a



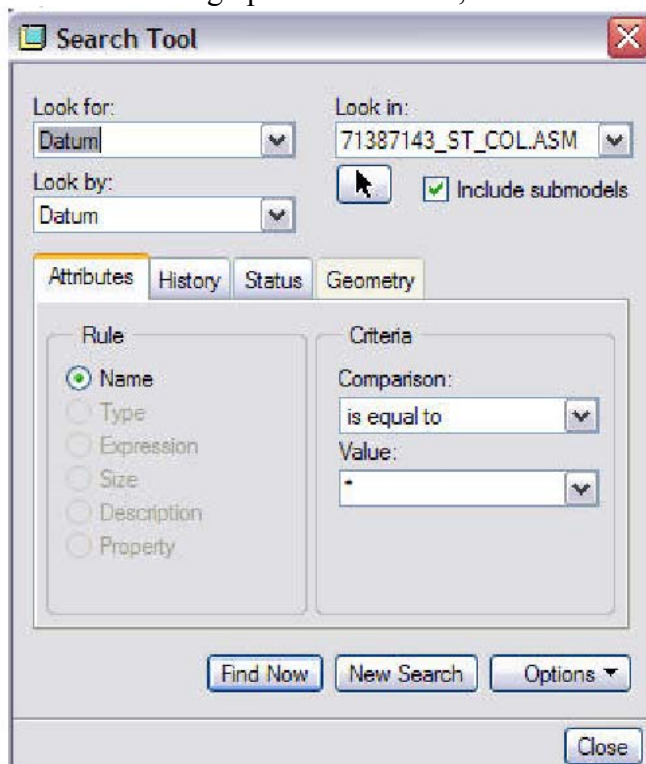
or **<ctrl> f** away whenever you need it! The following examples show how the search tool works in three different situations.

1. When you need to find a feature on a part
2. When you need to find a part in an assembly
3. When Pro/ENGINEER prompts you for a selection

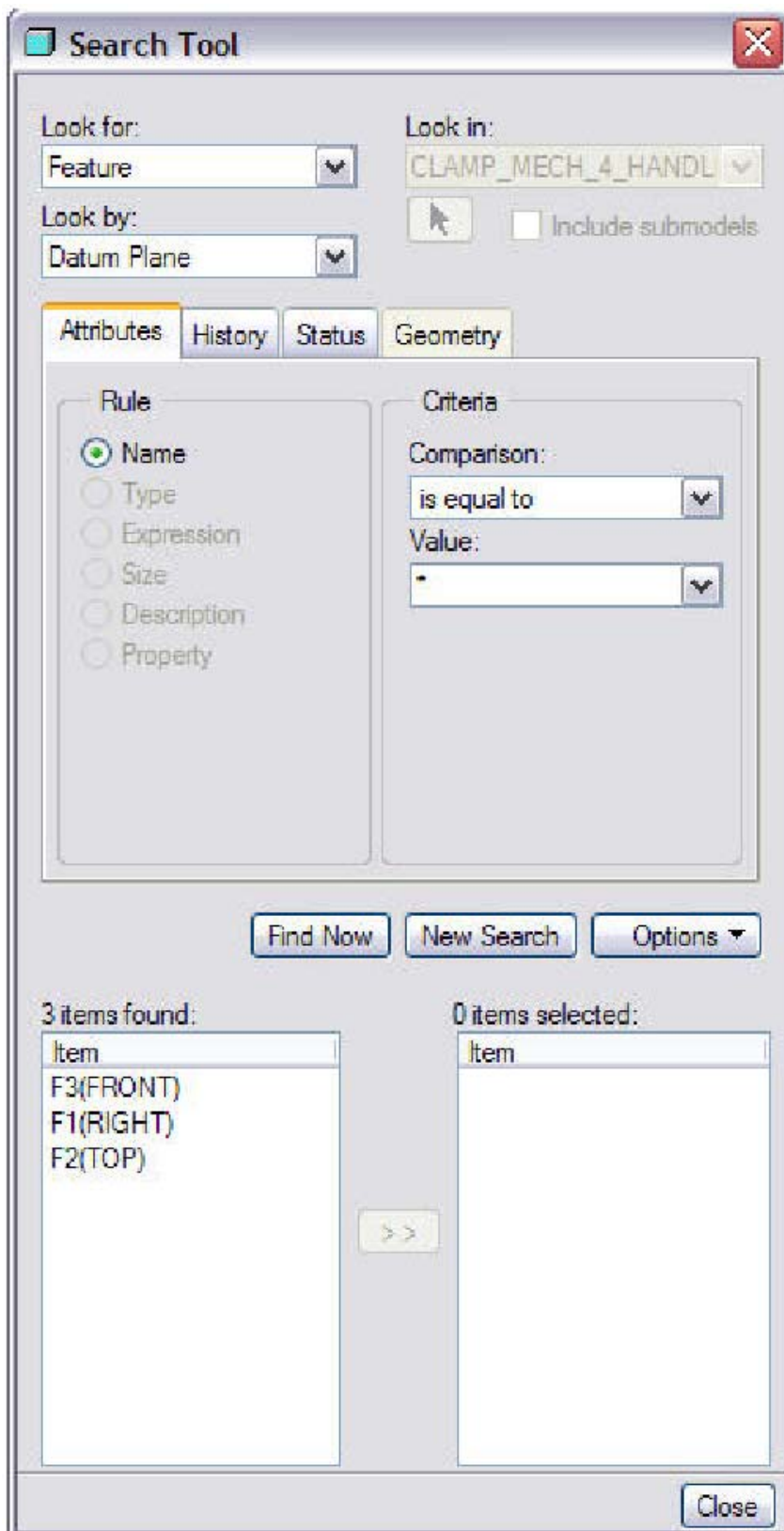
### Finding Features in Parts



With a part active, click the icon or press **<ctrl> f** from your keyboard. Either of these actions will bring up the search tool, shown below.



Notice that the Search Tool has a number of filters to help you narrow your search. For example, if you need to find all the datum planes in the part, you would configure the Search Tool as shown below, then press the “Find Now” button. Your search results will show in the lower section of the dialog box. Then it is a simple matter of moving the appropriate items from the “Items Found” panel to the “Items Selected” panel.

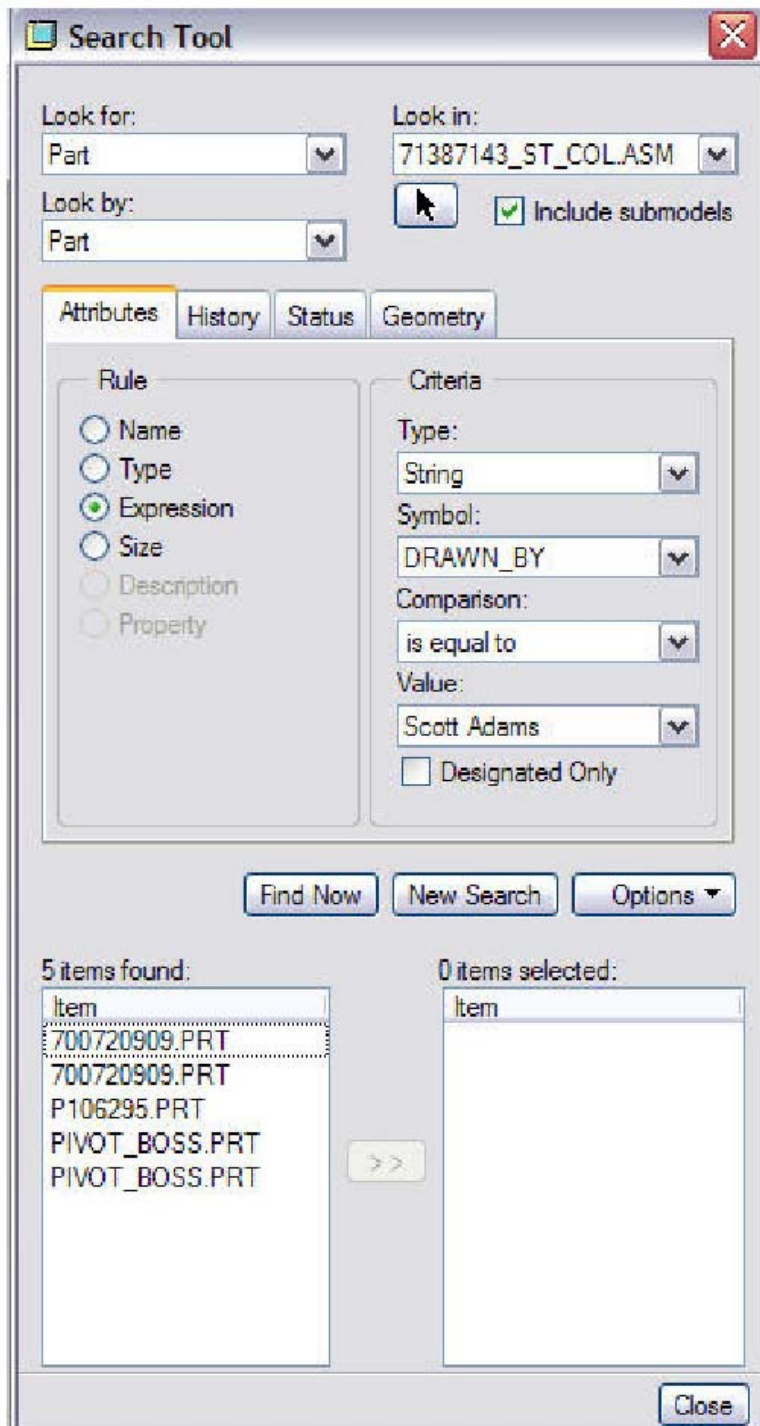


This example is very simple. The next example will show just how powerful the search tool can be.



## Finding Components in Assemblies


Let's suppose, for this example, that you need to find all parts in the current assembly that were modeled by Scott Adams. Finding parts that fit this criterion is a simple matter of configuring the search tool, and pressing the "Find Now" button at the bottom of the Search Tool dialog box. The example below shows the results of the search.

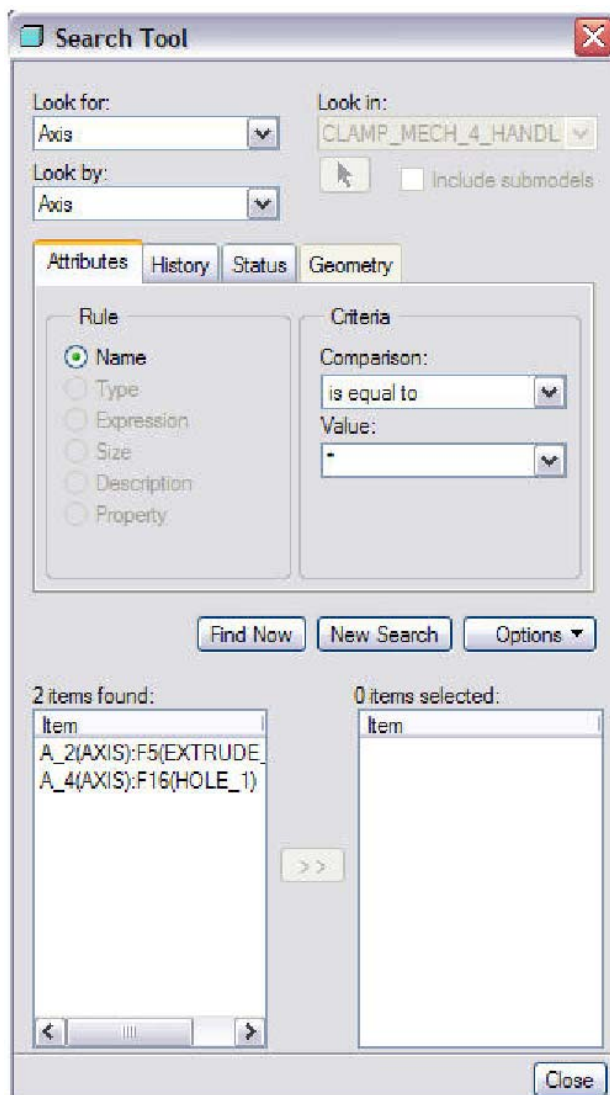


The drop down menus in the Criteria section of the dialog will contain the names of all parameters and values that are used in the model, so you don't even need to remember the exact names or values of the parameters you would like to search on. Is this cool or what?!?! In our example, Scott Adams has modeled five of the parts in this assembly. To select any or all of these parts, simply highlight them in the "Items Found" panel, and move them to the "Items Selected" panel.

## Searching When Prompted for a Selection

The Search Tool can also be used anytime Pro/ENGINEER prompts you for a selection (just

press <ctrl> f or  when prompted for a selection). In the example below, a coaxial hole is being created, and the system is prompting for an axis. In this case, the Search Tool is configured automatically because the system is looking for an axis. Pressing the "Find Now" button produces the output in the "Items Found" panel at the bottom of the dialog box.



## Summary

As you can see, not only is the Search Tool powerful, it is also accessible whenever you need it. When your model tree is long, or when you have lots of geometry displayed, try using the search tool when you need to find an item quickly, or when you are prompted for a selection.

Remember, it's just a  or <ctrl> f away!

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### Tips of the Week

#### Model Feature Dimensions Are Only a Double-Click Away

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### Announcements

#### PTC Tips & Techniques Newsletter Archives

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.

[Click Here To Access](#)

It's better than finding the Covenant of the Ark!

#### 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!

In addition to our Creation Tips & Techniques, here are some new events related to our Collaborate and Control solutions.

[August 19 - Tips & Techniques for Windchill ProjectLink Users](#)

[August 25 - An End User's Guide to Maximizing the Usability of Windchill Solutions](#)

Keep on the look out for additional events related to Windchill ProjectLink, Windchill PDMLink or Windchill in general coming soon!

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

[Tips & Techniques: Work Smarter Not Harder!](#)

## Hands-On Workshops

Experience and receive the next generation of CAD - Pro/ENGINEER Wildfire. During these workshops you can try it yourself to experience this breakthrough in simple, powerful, and connected in CAD software.

[http://www.ptc.com/appserver/it/icm/cda/template\\_lib/events/online.jsp?im\\_dbkey=17625&im\\_language=en](http://www.ptc.com/appserver/it/icm/cda/template_lib/events/online.jsp?im_dbkey=17625&im_language=en)

## Special Hardware offers for customers updating to Pro/ENGINEER Wildfire

[http://www.ptc.com/partners/hardware/current/wildfire\\_tlo.htm](http://www.ptc.com/partners/hardware/current/wildfire_tlo.htm)

<http://www.3dlabs.com/PTC/>

## PTC Sponsored Events

Click below to see PTC sponsored events:

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

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Thinking About Pro/ENGINEER Wildfire? Check this out.



<http://www.ptc.com/go/engineering/index.htm>

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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. Future plans include several foreign-language editions of Profiles for our many international readers. Currently, Profiles is printed in English and Japanese.

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, 2004      Your local Pro/Engineer User Groups  
<http://www.ptcuser.org/rugs/>

June 5 - 8, 2004      Orlando, FL  
PTC/USER International Conference  
<http://www.ptcuser.org/>

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>

### 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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