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PTC Product Focus

New for Wildfire 4.0 – Feature Recognition Tool (FRT)

Ever open a part and all you want to do is change the size of a hole, modify the depth of a pocket or change the size of a round or chamfer, only to realize the part is one big import feature? Turns out the part was imported by using IGES or STEP and now what do you do? Does the idea of plugging the hole to create a new hole, turn your stomach?

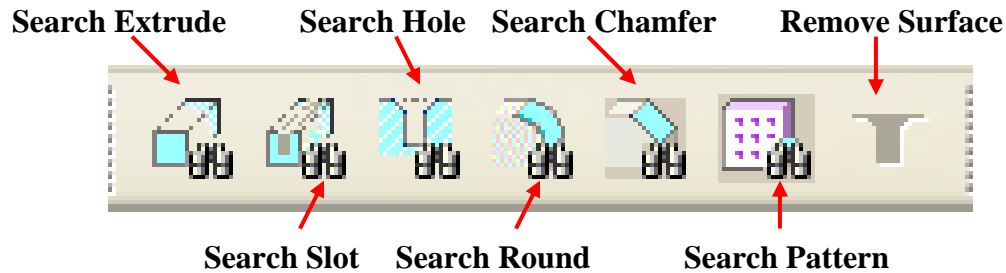
With FRT, a plug-in application for Pro/ENGINEER Wildfire 4.0, design engineers have a comprehensive set of tools to selectively replace geometries representing features in imported solid boundary representation models (IGES, STEP, etc) with true parametric features.

FRT enables design engineers to build design intelligence into imported models for easy, predictable changes. For example, a user can replace hole geometry with a hole feature and use it down-stream for assembly, detailing, and manufacturing. The non-modifiable static geometry is replaced with a fully editable hole feature. FRT can even recognize and place patterns of features.

Capabilities

- Identify and replace static geometry with feature rich content.
 - Simple or Sketched Holes on Flat or Curved Surfaces
 - Protrusions or Pockets on Flat or Curved Surfaces
 - Extruded Slots
 - Constant Radius Rounds
 - Chamfers
 - Table Pattern for Holes
- Easy – to – Use Action/Object approach

- Select the desired Feature Search Tool
- Then Select the Surface or Edge to start the search from
- Have constant visual feedback during feature creation.



Benefits

- Facilitate down-stream re-use of imported models.
- Rebuild flexibility into imported designs from a single, familiar modeling environment.
- Increased efficiency over remastering 3D imported designs by hand.

More Information on Feature Recognition Tool

To find out more about Pro/ENGINEER's [Feature Recognition Tool](#) visit our Website.

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PTC Product Focus

Workgroup Manager for ECAD

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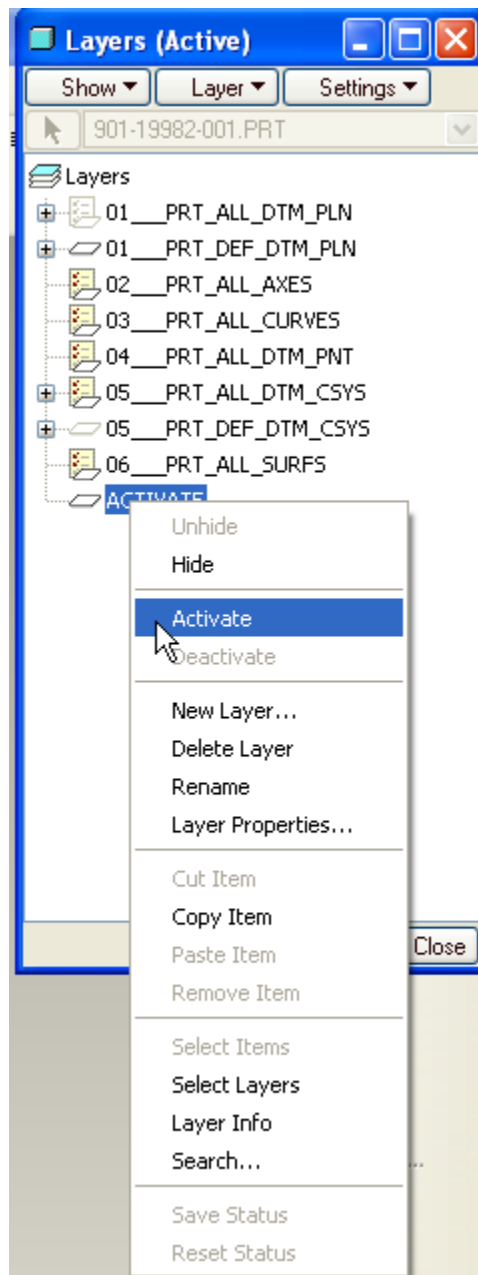
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Tips of the Month

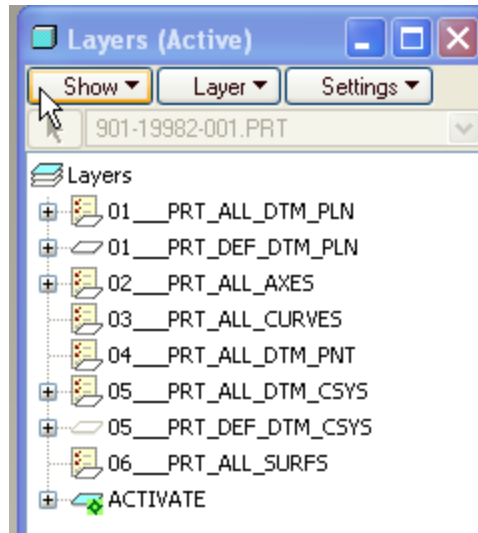
Activate Layer, Autorounds and Holes in Pro/E Wildfire 4.0

Pro/Engineer Wildfire 4.0 now provides the ability to populate Layers using a new command called “Activate”, create concave and convex rounds over an entire part without the need to pre-select edges using Autorounds and at long last provides designers the ability to create a Simple Hole with an option for a drill point at the bottom of the hole and a chamfer and/or counterbore at the top of the hole.

First we’ll look at Layers and the new “**Activate**” command.

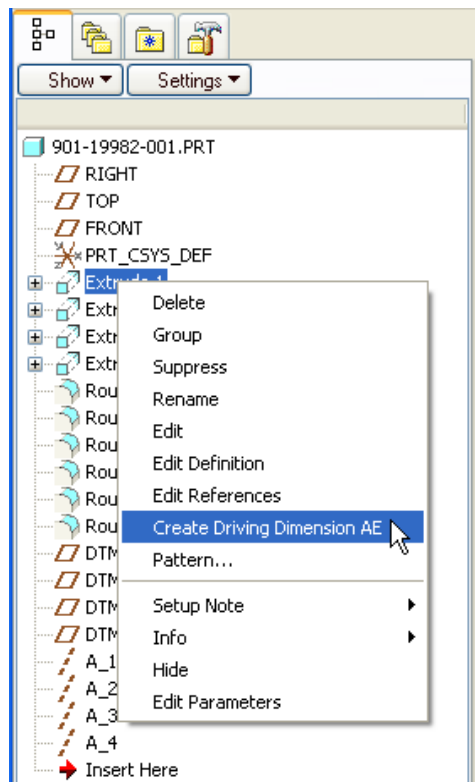


After creating a layer, the user can hold down the right mouse button and select **ACTIVATE**. The icon next to the active layer will appear as shown below.

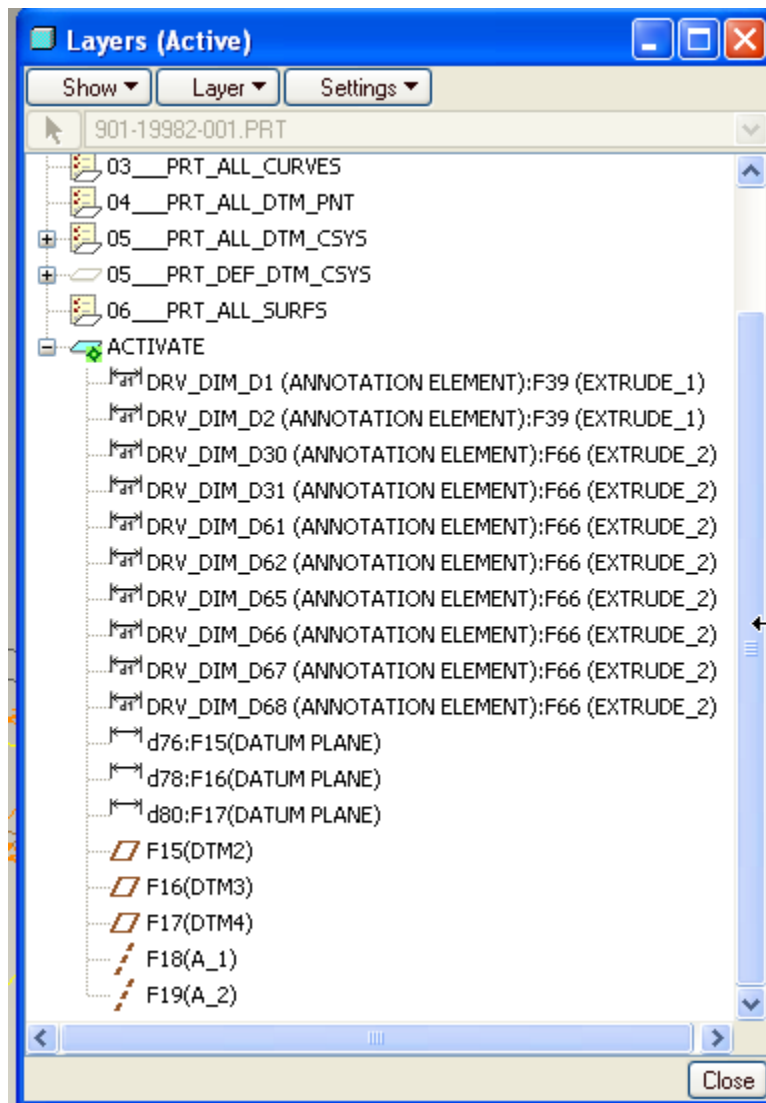


Once in this mode, any features created in the model will be automatically placed on that layer without the need to individually select the specific features or create rules. This works for axes, points, planes, coordinate systems, 3D notes, etc. This works especially well when creating annotation elements in a model in conjunction with model based definition.

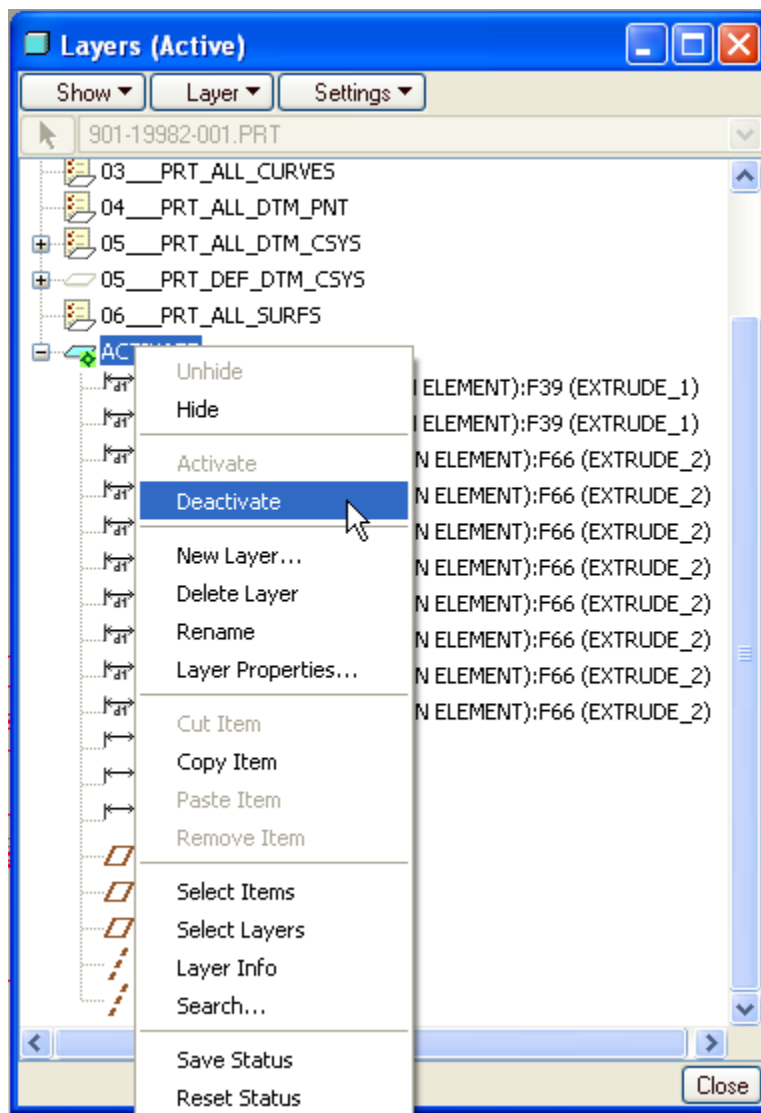
If you are going to be creating a group of annotation elements specific to a feature either manually or by using the new “Create Driving Dimension AE” command, the user can define and activate the layer and have all of that information automatically populated to that layer.



To verify layer content simply select the layer from the dialog box.



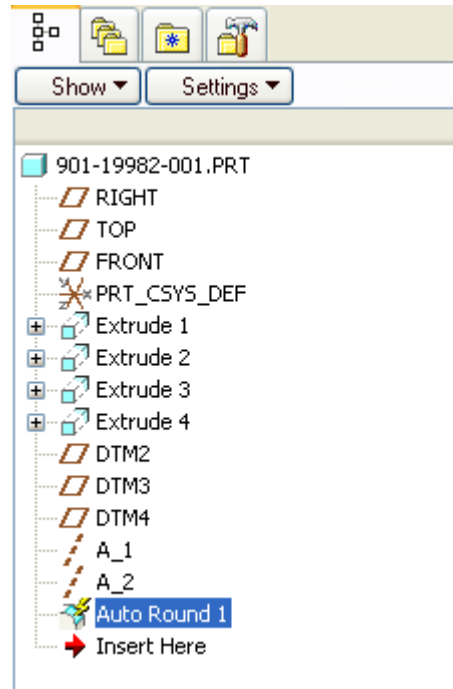
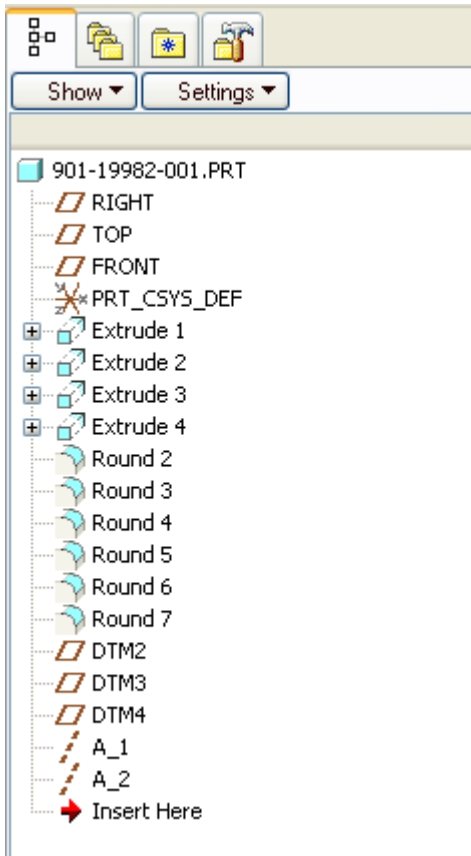
To deactivate a layer, simply select the layer to highlight, hold down the right mouse button and select “Deactivate”. New features created will no longer be added to that specific layer and will be subject to any layer rules and config option that have been defined.



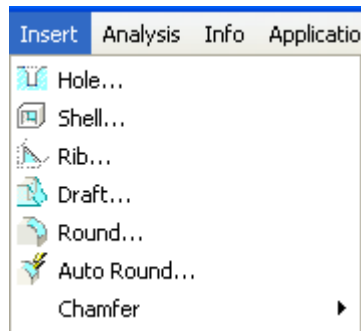
Note: “ALL” states in the View Manager can now be set to include layer status.

Let’s move on to the new “**Autoround**” feature.

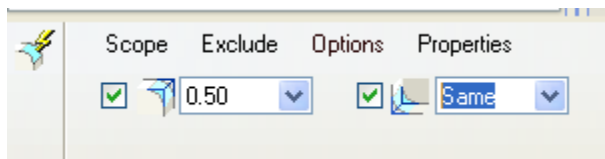
Do you work with castings or molded parts? Do you find that you spend a lot of time adding rounds to complex parts? Autorounds was created to save you time and simplify creation of multiple rounds. All of the rounds shown in the Model Tree on the left below can be replaced with a single Autoround feature as show in the Model Tree on the right below.



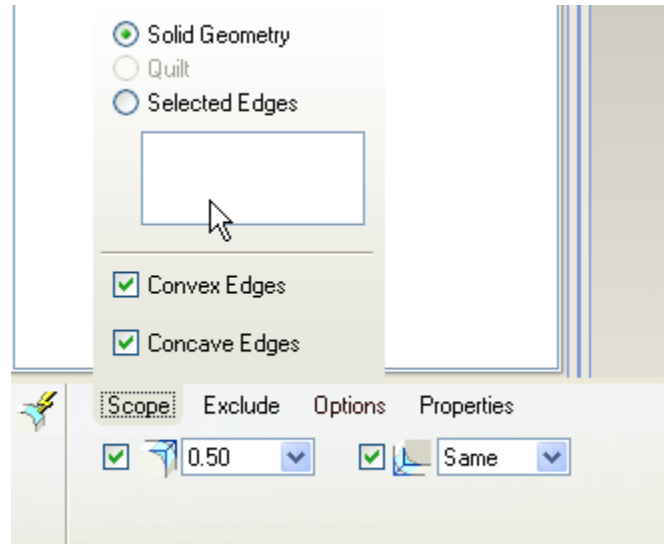
Simply select the Insert pull down menu and browse down and select the Autoround command.



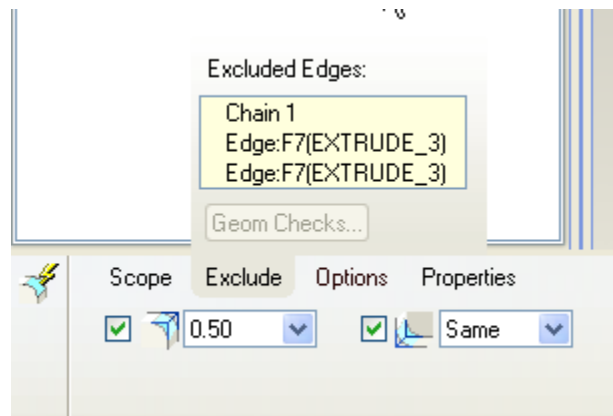
The Autoround dashboard will then appear. Check the boxes for concave or convex shaped rounds and also enter a value for the size of the rounds. The user can choose separate values for the size of each type or use the Same option.



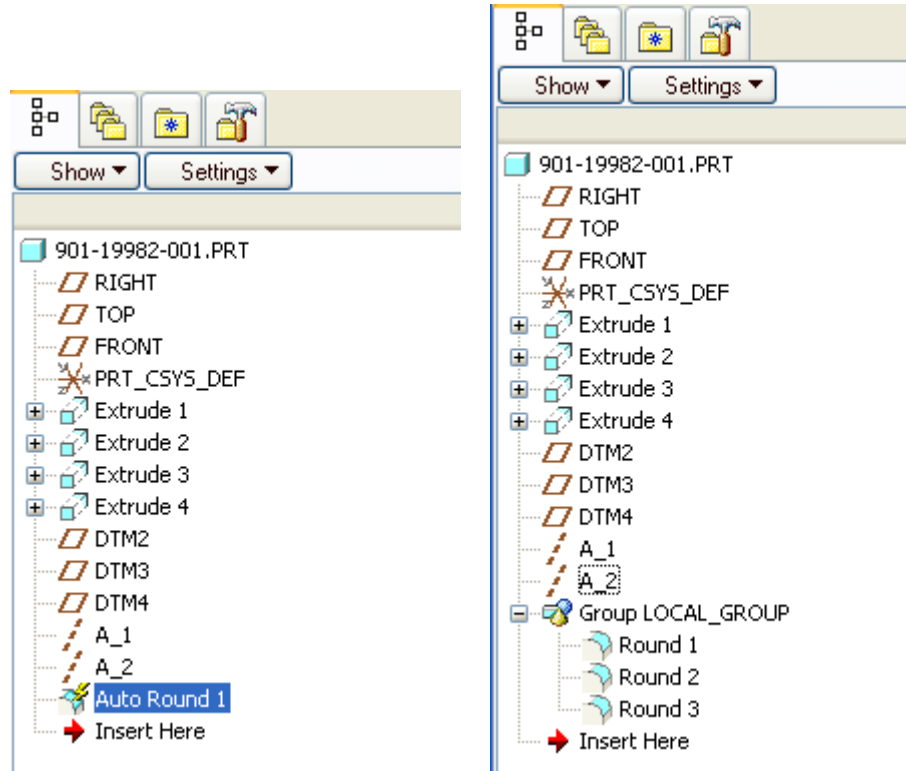
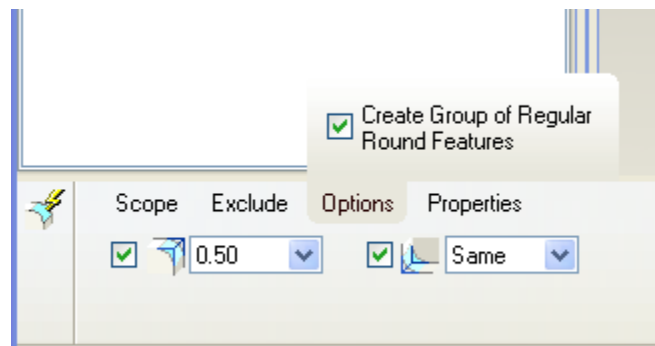
Other additional options can be defined in the pull up menus titled Scope, Exclude, Options and Properties. Under the Scope menu, the user can specify whether to round all sharp edges that exist by selecting Solid Geometry which is the default option or choose to add limits by activating the Selected Edges option and then selecting specific edges. The user can also choose to turn on or turn off, Convex Edges and Concave Edges.



When the Exclude pull up menu is selected the user is provided a means to add limits to the Solid Geometry scope by defining edges to be ignored by this operation.



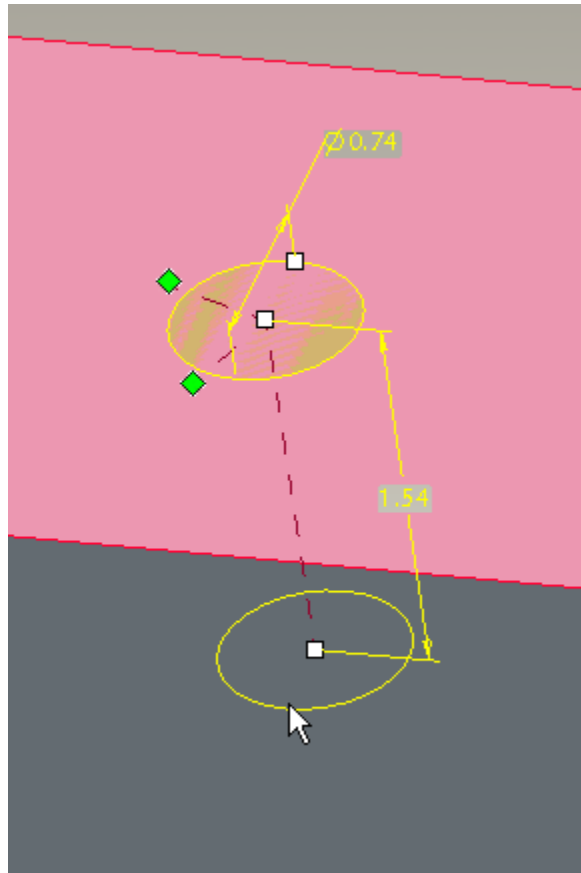
When the Options pull up menu is selected, the user can check the Create Group... option. This will change the way the rounds are displayed in the Model Tree. Instead of a single entry for Autoround, the user will see a group containing multiple rounds. Each of those rounds can contain individual round sets. The value for the radius in each of those sets can then be controlled independently as shown below.



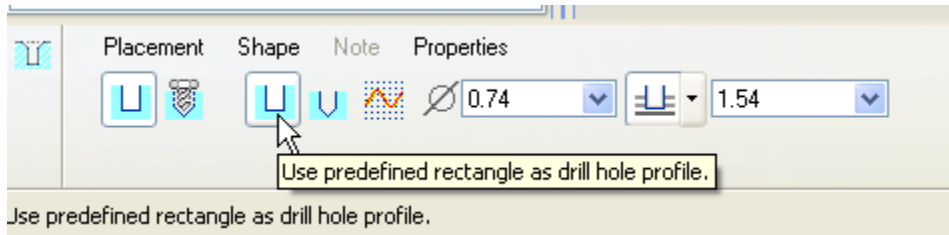
Lastly, let's look at the most basic of features the **"Hole"** and see what is new.

When creating a Standard Hole using UNC, U NF, ISO, etc. the user has always had the option to add countersinks, counterbores and can control the drill point. When using the Simple Hole the user was left with a flat bottom hole unless they used the sketch command which added additional steps to complete the process. Simple Holes have been enhanced to include countersinks, counterbores and drill points. At the same time Standard Holes have been enhanced to include tapered pipe threads.

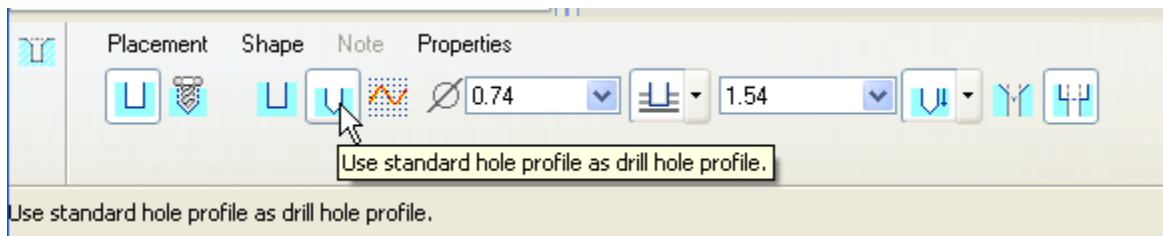
When placing a Simple Hole notice that the drag handles for the placement references appear green if they are not associated to a surface and become white once attached to a surface or edge.



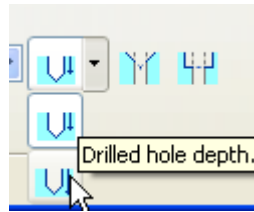
In the dashboard, with the Simple Hole icon selected, there are now icons for both flat bottom and drill point shapes.



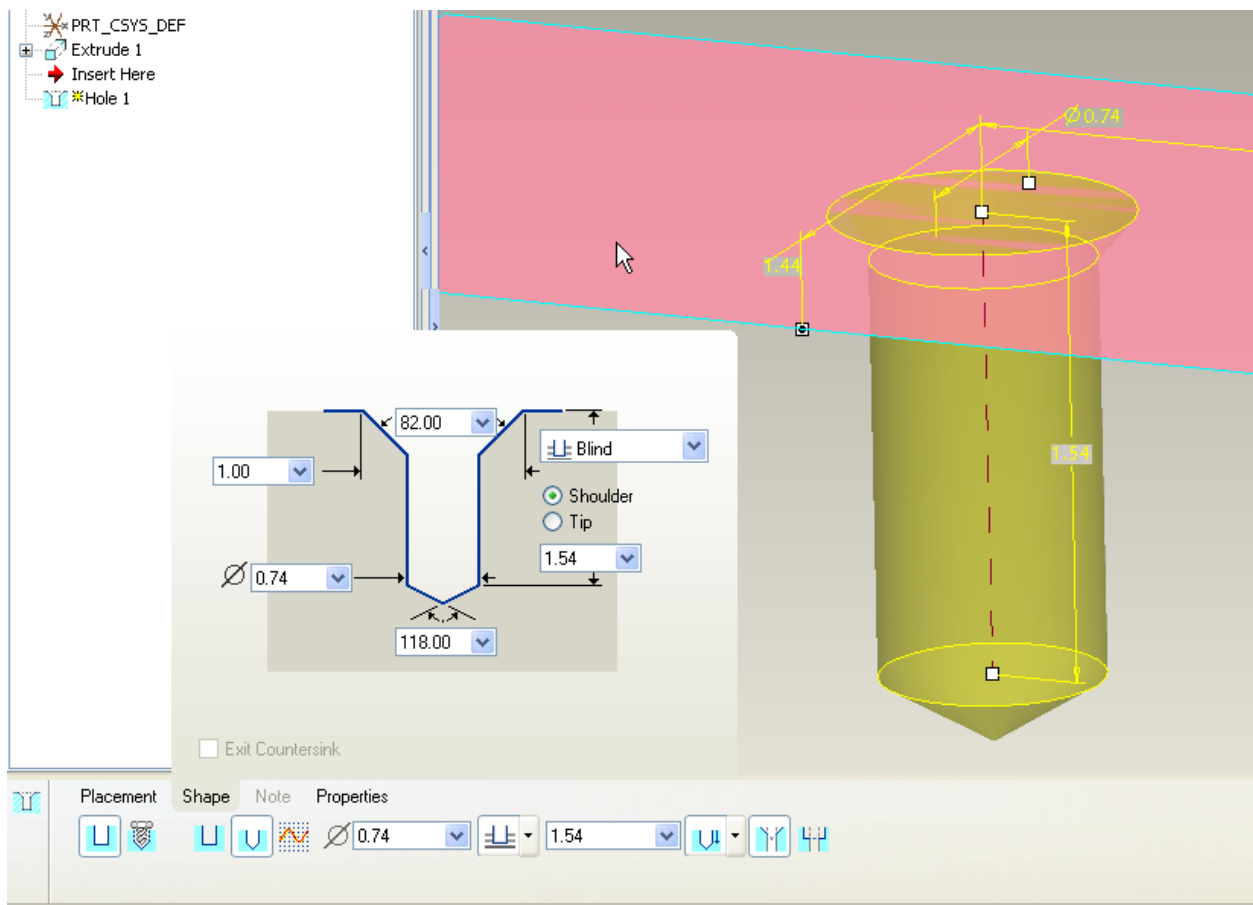
Select the Drill Hole icon and additional icons will appear to the right. The user can select the icon to add a counterbore, a countersink or both.



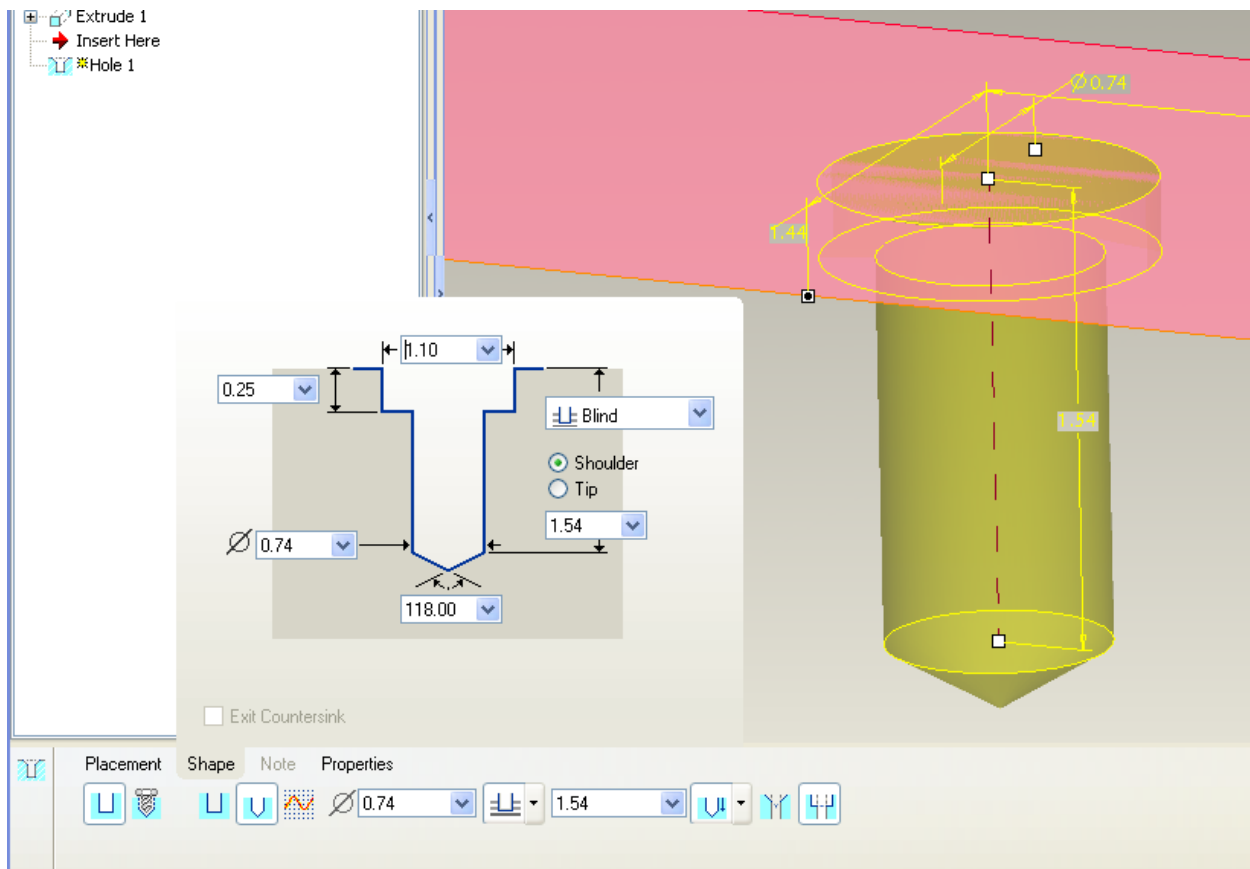
The user can also specify whether the depth of the hole is measured to the shoulder of the cylinder or to the tip of the drill point.



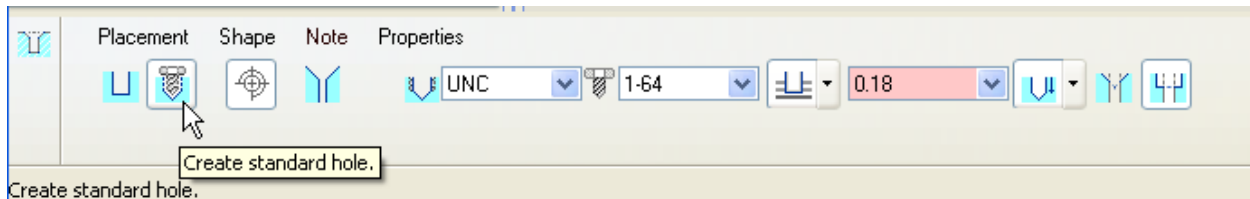
By selecting the Shape pull up menu, the user can view all of the dimensions associated with the hole including countersink diameter and angle, hole depth to Shoulder or Tip, and drill point angle.



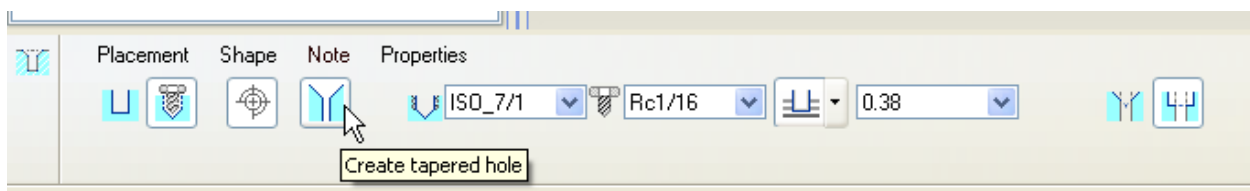
The same capability exists for editing dimensions specific to counterbores. This also works if both countersink and counterbore are selected.

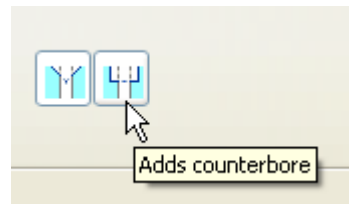
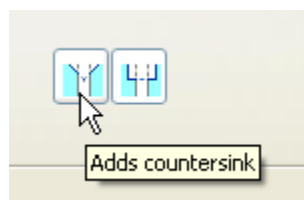
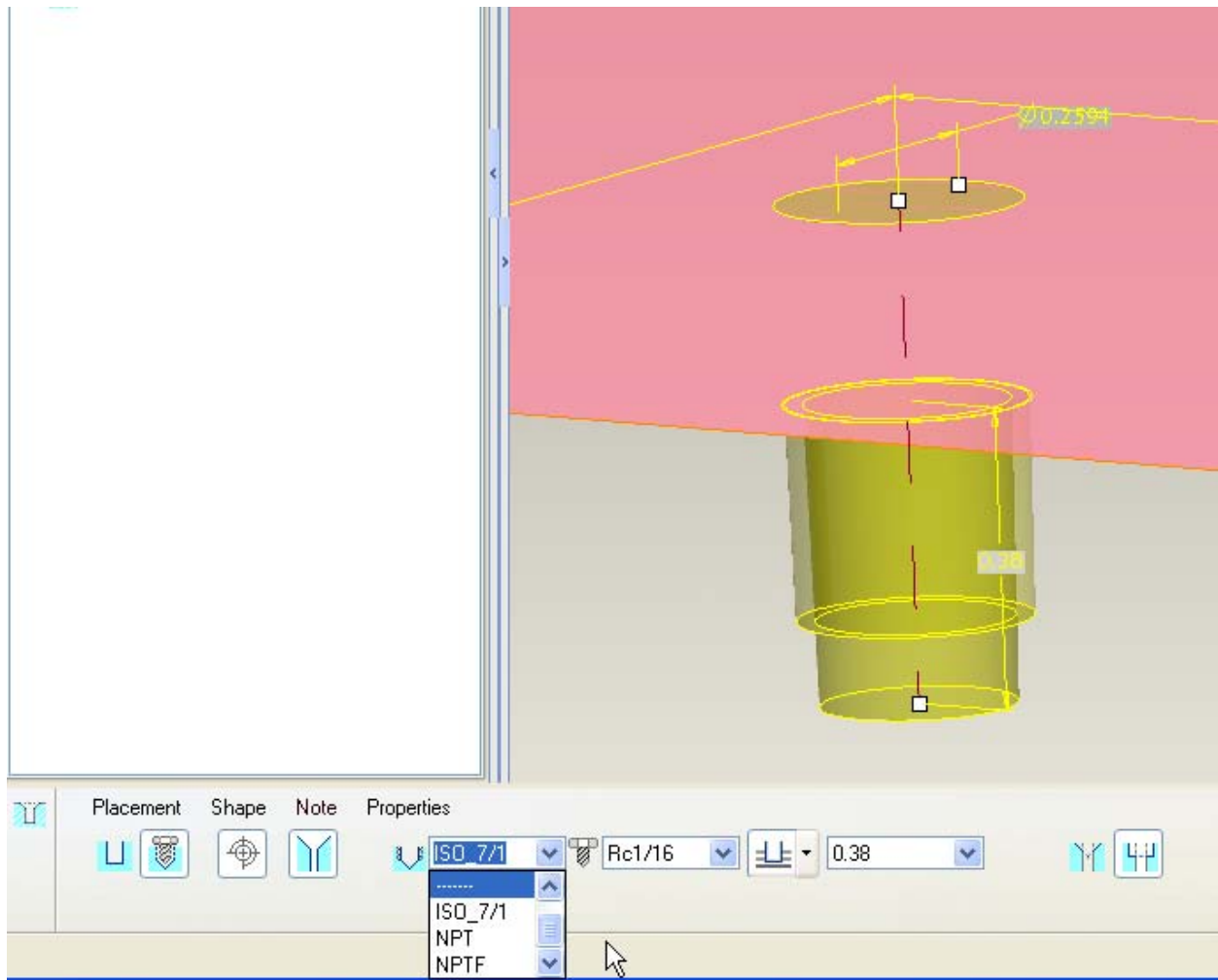


Another new capability is found under the Standard Hole. Users now have the ability to create tapered pipe thread holes based on NPT, NPTF or ISO_7/1 standards.

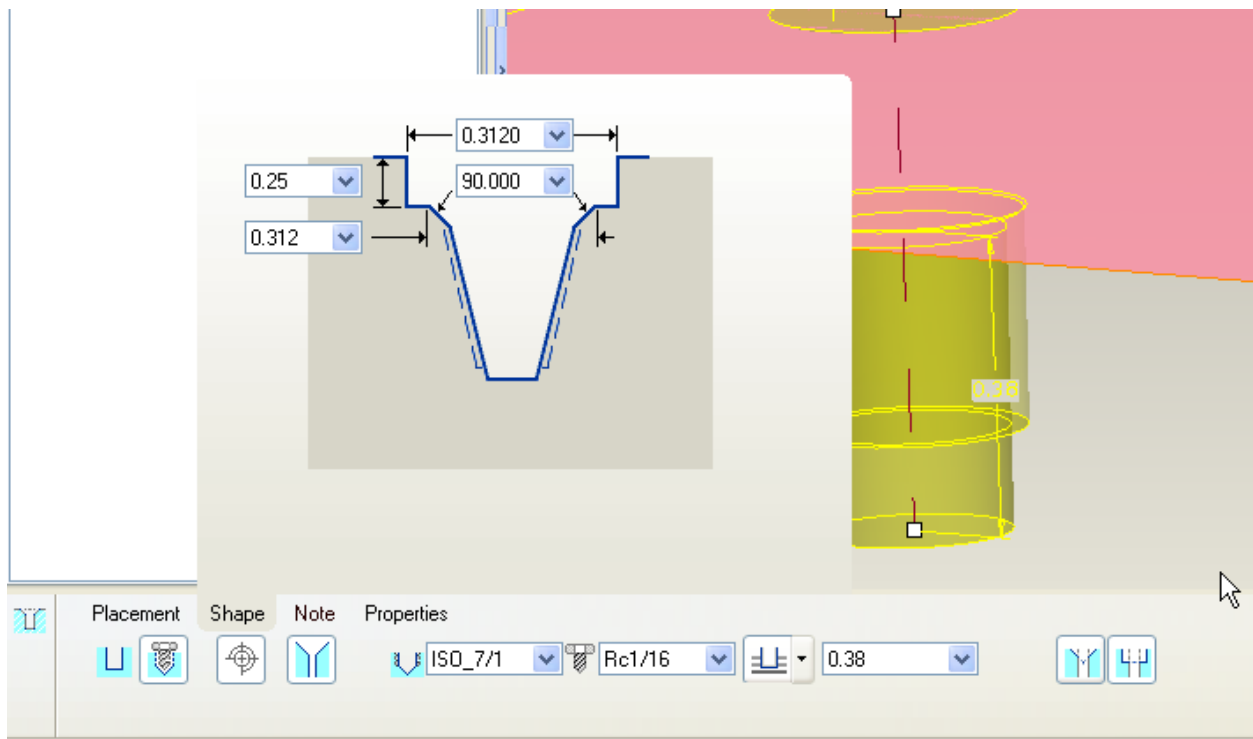


After selecting the Standard Hole icon, select the Create Tapered Hole icon. The user will be able to choose from a listing of hole tables specific to industry standard tapered threads. Look to the center of the dashboard for countersink and counterbore icons.





As always, selecting the Shape pull up menu, will provide the user access to a 2D cross section cutaway of the hole as well as any modifiable dimensions.



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Tips of the Month

Creating and Using Bookmarks in InterComm

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Announcements

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 - Conceptual and Industrial Design
 - Detailed Design
 - Simulation/Analysis
 - Production
 - Design Collaboration
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- Windchill ProjectLink
- Pro/INTRALINK
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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, 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.

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