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PTC Product Focus

Part and Assembly Layers for Wildfire 2.0

Pro/Engineer unleashes power to group features and parts into logical layers. All items in each layer are selectable, and bulk operations can be performed such as suppressing features, excluding parts in simplified reps, editing and creating feature-level and part-level parameters, and much more.

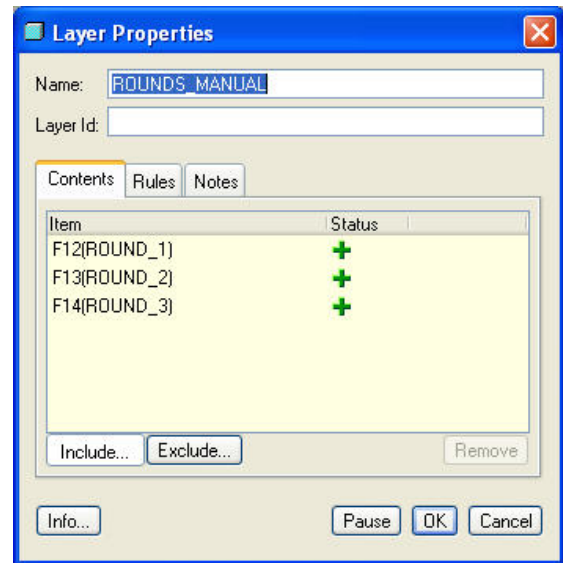
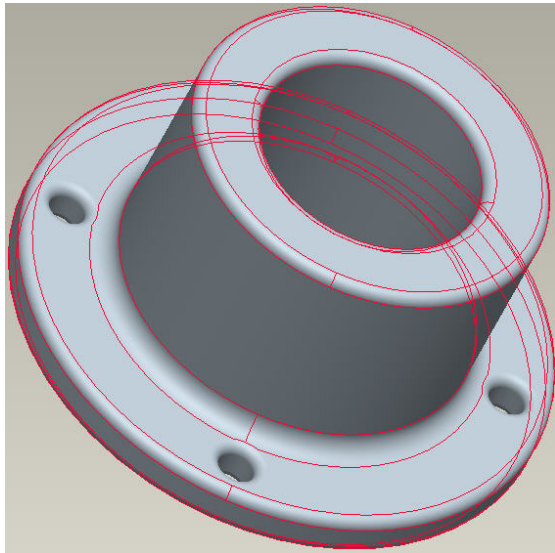
Additionally, layers can be created and controlled in a couple of different ways. For example, a layer can be created where you select each feature or part to include individually, or a layer can be created based on rules to automatically include those features or parts that match user-defined criteria. Furthermore, the rule-based layers can be configured to include any future feature or part created that match the criteria.

Once organized into layers, a group of features in parts can be selected quickly, which can then be suppressed, deleted, or grouped together in one single action. Or in assemblies, a group of parts can be selected simultaneously to suppress, regenerate, delete, edit/add parameters, or set the view state for simplified reps in one single action.

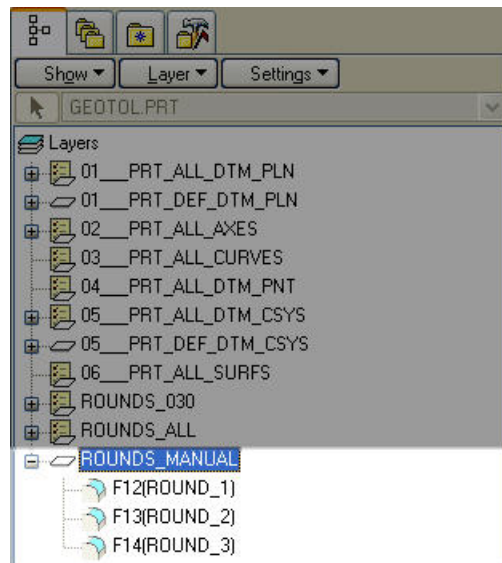
In this article, we will take a look at creating layers manually, creating layers based upon rules, and then utilizing layers to perform bulk operations on features and/or parts.

Creating Layers Manually

To create a layer manually, click **Layer, New Layer...** from the *Layer Tree*, then click on the **Contents** tab in the *Layer Properties* dialog box. Make sure the **Include...** button is depressed, and then if working on a part, select any feature(s) to add to the layer. If working on an assembly, select any part(s) to add to the layer. Press **OK** in the *Layer Properties* dialog box when finished.



Layer Properties dialog box with selected Round features shown on part. Below is the resulting layer found in the Layer Tree.

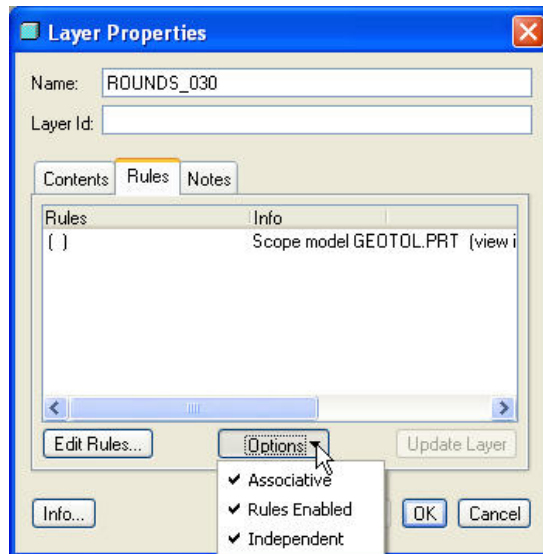


Creating Rule-Based Layers

To create a rule-based layer, click on the **Rules** tab in the *Layer Properties* dialog box, and press the **Options** button. A drop-down menu will appear with the following options:

- **Associative:** Searches the model for any current features/parts that meet the rule-based criteria.
- **Rules-Based:** Adds any future feature/part to the current layer if it matches criteria.
- **Independent:** Must be set in order to create rule-based layers.

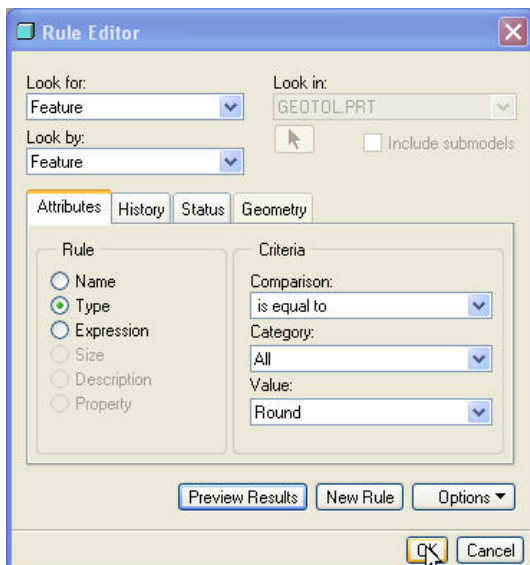
Make sure the **Independent** option has a check mark, which will enable the **Edit Rules...** button. Then select any additional options according to your preferences.



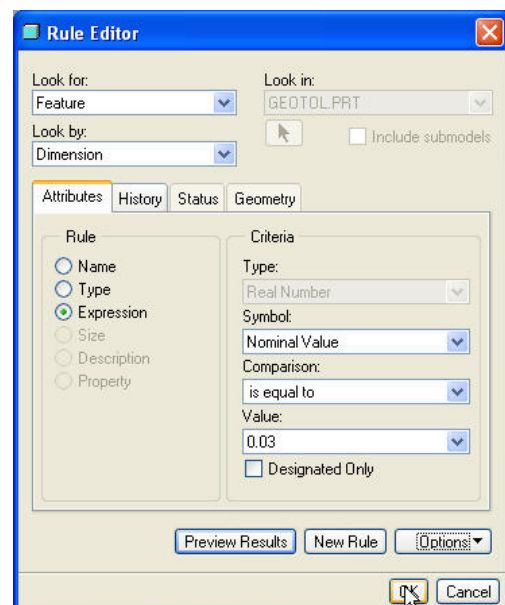
Layer Properties dialog box with Rules tab selected, and the three Options shown.

Once you have selected your options, click the **Edit Rules...** button, and the **Rule Editor** dialog box will pop up on your screen. This is the dialog that allows you to choose what type of entities you are searching for, and the criteria used to add matching entities to the current layer. When you have completed entering your criteria, you may hit the **Preview Results** button to verify that your rule delivers the expected results.

To illustrate the possibilities with rule-based layers, we will look at a couple of examples using the same model from the Manual Layers section. In the first example, we will create a layer named ROUNDS_ALL that finds all the rounds in the part. We will accomplish this by looking for all features that are of the type 'Round'. In the second example, we will create a layer named ROUNDS_030 that finds all the rounds in the part with a radius of .030". We will do this by looking for all features that have a dimension nominal value of .03.

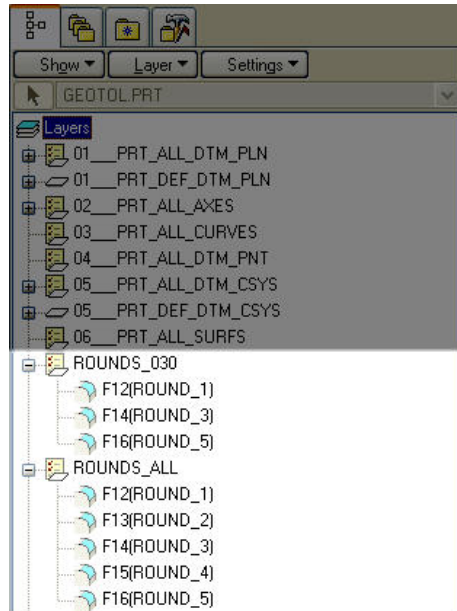


Rule Editor showing how to create a rule that finds ALL rounds.



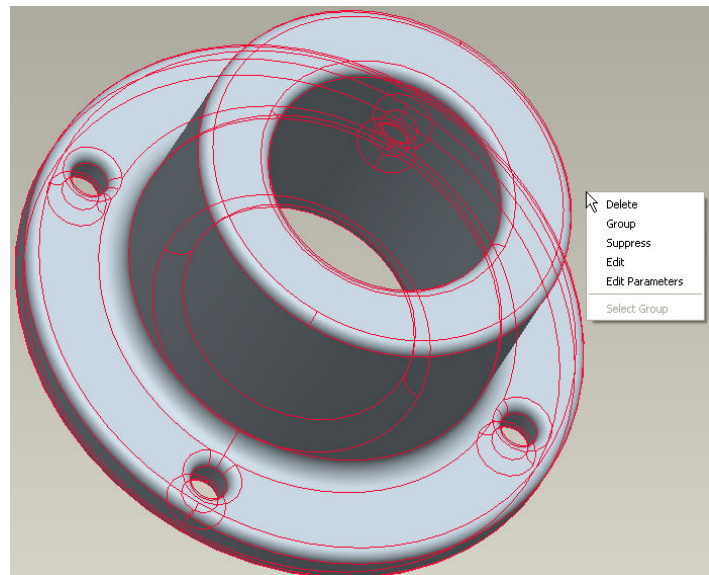
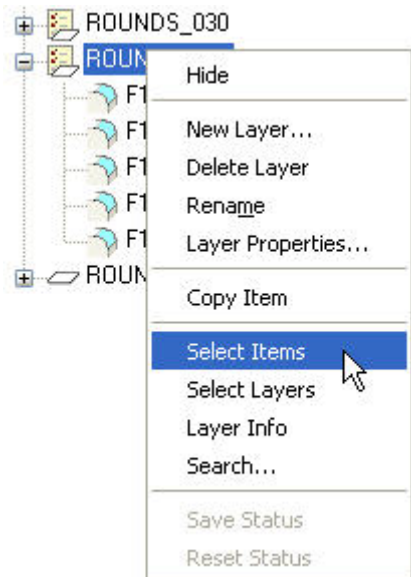
Rule Editor showing how to create a rule that finds all rounds with a dimension of .030.

The following picture shows the resulting features added to each of the layers created using our rules.



Using Layers for Bulk Operations

Once the new layer has been created, whether manually or by rules, you can right-click the layer from the layer tree, and choose **Select Items**. Then right-click in the graphics area to see all available options for bulk operations, such as deleting, grouping, suppressing, or editing parameters.



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PTC Product Focus

Windchill PartsLink Module for PDMLink 8.0

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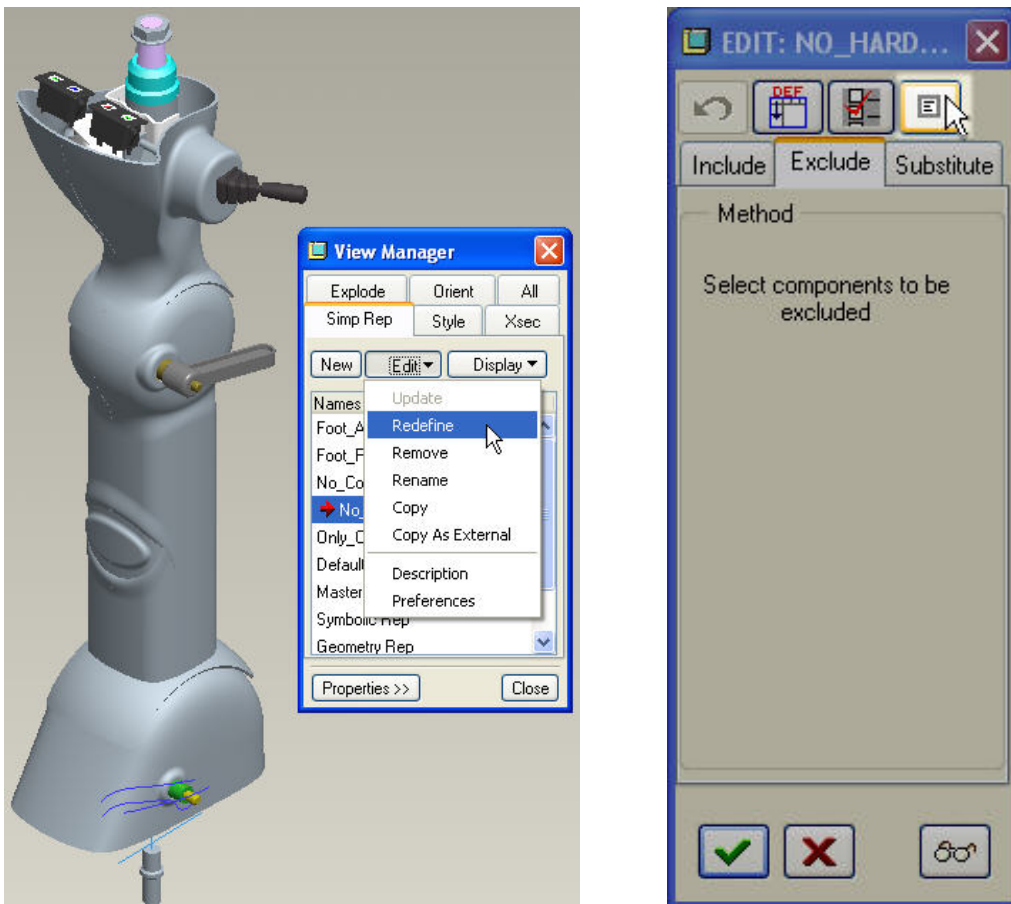
Tips of the Week

Creating Rule-Based Simplified Reps w/ AAX

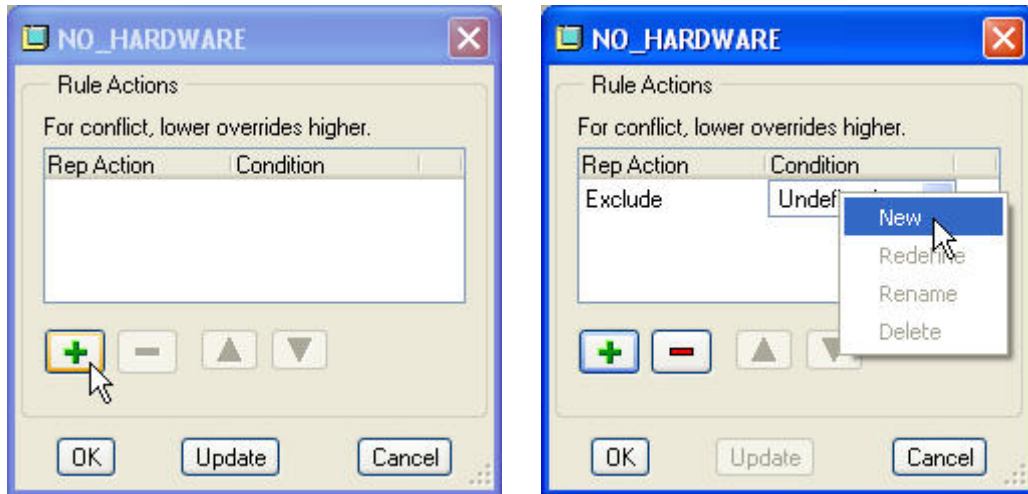
Similar to creating rule-based layers, the Advanced Assembly Extension in Pro/Engineer Wildfire gives you the ability to create rule-based Simplified Reps. This means you can define a simplified rep that automatically updates itself when new parts are added to the assembly, all based upon user-defined criteria.

As an example of this, we will look at a steering column assembly. We want to create a simplified rep that excludes all the hardware so we get better performance when working on the design. Furthermore, we want to be sure that any hardware added in the future will automatically become a part of this simplified rep.

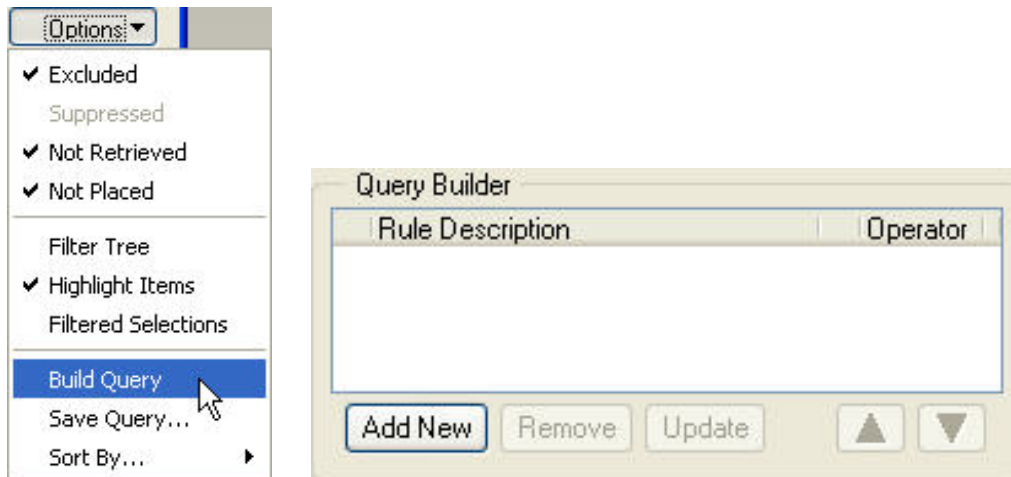
To accomplish this, the first thing we will do is to create a simplified rep called **No_Hardware**. Then with the rep selected in the **View Manager** dialog box, we select **Edit, Redefine**. Then in the **EDIT: NO_HARDWARE** dialog box, make sure the **Exclude** tab is selected and press the **Setup Rule Actions** icon.



At this point, the **NO_HARDWARE** dialog box will come up, and we press the + button to add a new condition to the definition of our simplified rep. A new line will appear in the box, defaulting to exclude parts that match our rule criteria, and with an undefined condition. We simply right-click the undefined condition and select **New** to create a new condition, and then press enter inside the condition box to bring up the **Rule Editor** dialog box.

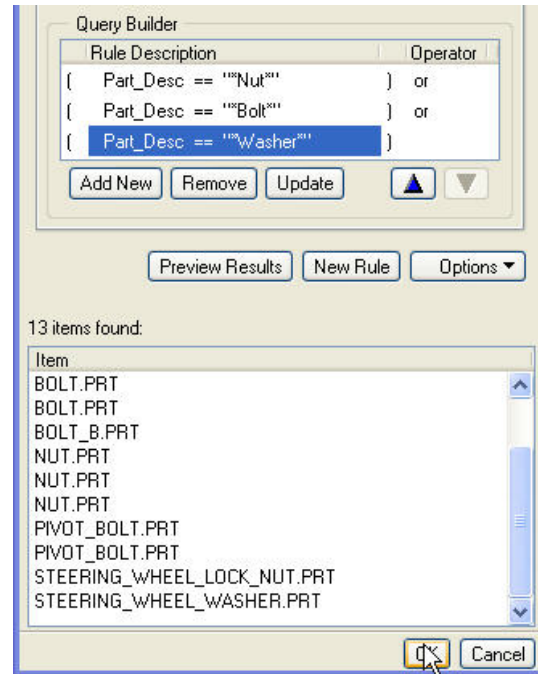
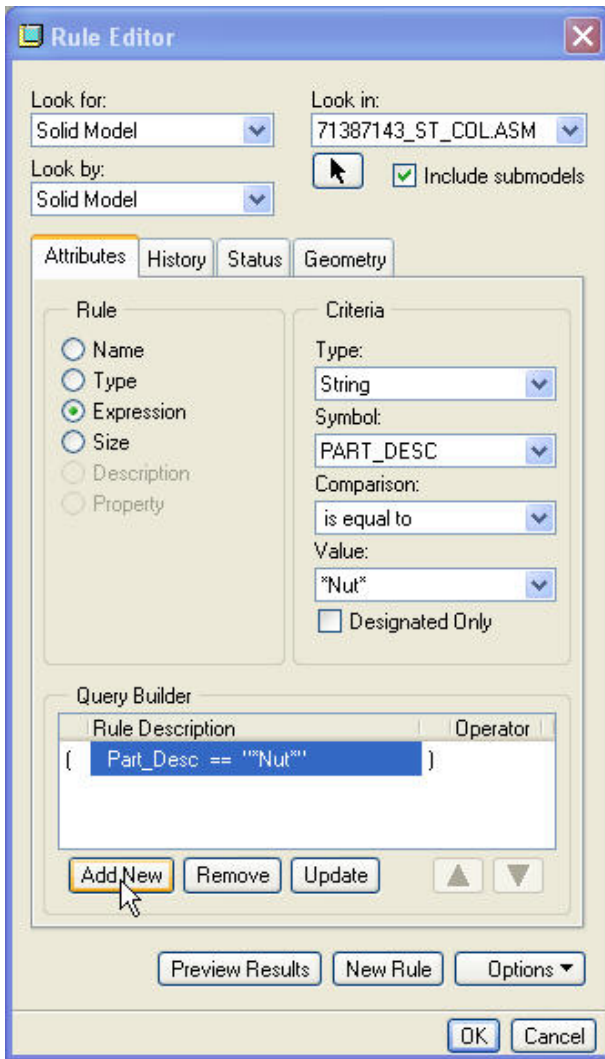


If the **Query Builder** is not shown in the **Rule Editor** dialog box, click on the **Options** button at the bottom of the dialog box, and select **Build Query**.



The next step is to create the actual criteria we will use to find all the hardware in our assembly. Unfortunately, we do not have a **PART_TYPE** parameter that has a unique entry for hardware components, so we will have to create our rule based upon the **PART_DESCRIPTION** parameter.

We will accomplish this by searching for all solid models that have the words “nut”, “bolt” and “washer” in their description. To find all parts with the word nut in their description, we set up the **Rule Editor** dialog box as shown in the following figure, and press the **Add New** button. We then repeat this procedure changing the **Value** field to “*Bolt*” and “*Washer*” to complete our rule definition for this simplified representation.



After the *Query Builder* information is complete, we select **Preview Results** to verify that we correctly defined our rule. Notice that in the preview box we see several parts in this assembly that are nuts, bolts and washers. This means we have correctly defined this rep, and we can press **OK** in the *Rule Editor* dialog box to complete our rule. Finally, after pressing **OK** on all subsequent dialog boxes, we have successfully created our rule-based simplified representation.

An alternate way to do this exercise is to create a PART_TYPE parameter in each part (you can include this parameter in your start parts) and classify them as hardware, electrical, sheetmetal, etc. Then creating rules becomes easier and more accurate, as you do not rely on any consistency in part names or descriptions.

As one last note, if any parts are subsequently added to our assembly that have the words nut, bolt or washer in their description, they will automatically be excluded from this simplified rep.

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Tips of the Week

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Upcoming, 2006 Your local Pro/Engineer User Groups
<http://www.ptcuser.org/rugs/>

June 4 - 7, 2006 Dallas, Texas
PTC/USER International Conference
<http://www.ptcuser.org/>

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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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