PTC Technical Specialists E-Newsletter

Date: November 30, 2005

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PTC Product Focus

Cabling Design Option

With Pro/ENGINEER Cabling Design, PTC has given it's users an easy way to create and route wires or cables in their Pro/ENGINEER assemblies. This functionality has eliminated the need to create prototypes of products in order to determine wire lengths for a wide variety of customers. Wires and cables can be grouped into harness parts and these harness models can be flattened out in order to create fully detailed nail board drawings.

The Cabling Design Option can be implemented with different levels of complexity. Users can choose to use the routing capability to create simple representations of wires, cables, and hoses or they can choose to link up the cabling environment with PTC's schematic capture tool, Routed Systems Designer, to automatically route wires or cables based off a wiring diagram.

The Cabling Design Option has now been included in Pro/ENGINEER's high end seats named the Flex 3c. Due to this fact many of the users reading this newsletter may already have access to this functionality. If you do have access, try out cabling by pulling up an assembly, and selecting **Applications**, **Cabling** from the pull down menus. This will bring up the Cabling menu.

Routing cables can be done by following four steps. These steps are:

1. Create a harness
2. Read in or create spools
3. Designate connectors
4. Create and route wires or cables

To create a harness, choose **Harness** from the cabling menu. This will allow the user to create or activate an existing harness. A harness is a part file in Pro/ENGINEER. It will show up in the model tree and get saved with a .prt extension. However, these harness parts can not be retrieved by themselves outside the context of an assembly. They can be used to group wires or cables

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together for downstream deliverables. An assembly can contain multiple harness parts. To create a harness simply select **Create** and give the harness a name.

The next step is to read in or create spool files. A spool is a definition for a wire or cable that contains thickness, minimum bend radius, color, and any other parameters the users would like to track for a particular type of wire. The idea is that you can create a spool to use to route individual wires just like it would be done on the shop floor. To create a spool select **Spools** from the *Harness* menu. Select **Create** from the *Spools* menu, set the type (wire or cable), and give the spool a name. This will bring up the Electrical Parameters dialog box shown below.
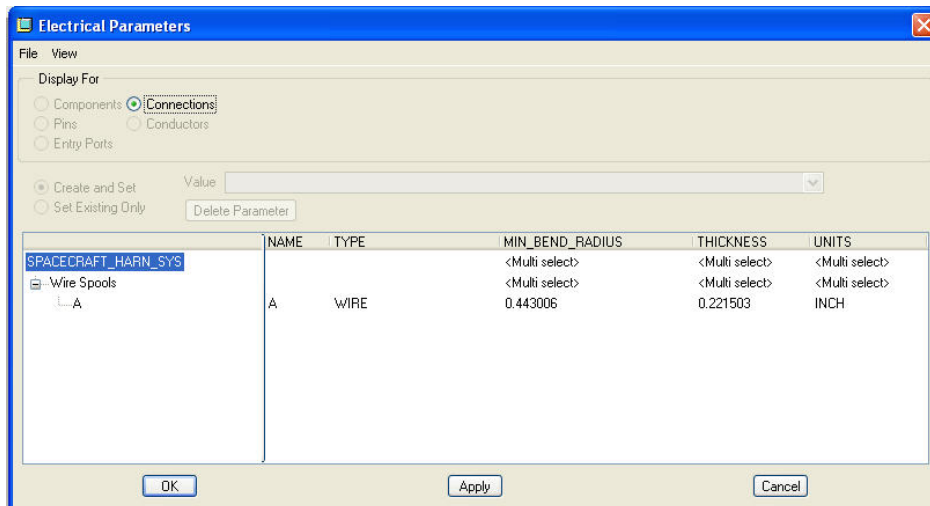
Fill out the cells for minimum bend radius and thickness. Additional parameters can be added to the spools by selecting **View, Columns** from the dialog box's pull down menus. This will bring up a dialog box to select many different electrical parameters or create user defined parameters. Add or create parameters as needed.

Creating spools should be a one time creation process. Select **File, Write** from the dialog box to save the file to disk. This will create a .spl file. These files should be placed in a network accessible directory. A config.pro option of pro_spool_dir can be used to point to this directory for future use. Hence, users could simply read in existing spools for new harnesses.

The next step is to designate connectors. Any Pro/ENGINEER part or assembly can be a connector. However, the model should contain coordinate systems to represent where the wires or cables will be attached. These coordinate systems should have the z axis point in the direction the cable will attach. To designate a connector, select **Components** from the *Cabling* menu. This will bring up the *Cable Components* menu. Select **Designate** from this menu and select a part or subassembly. The user can read in a parameters file for the connector or select the default parameters. Hit the return key to accept the default parameters.

The component is now designated. The user can now define information for the electrical connection points. These points are called Entry Ports and are represented by coordinate systems. Select **Entry Ports** from the *Mod Conn* menu and select a coordinate system on the designated model.

Multiple coordinate systems can be selected by holding down the <ctrl> key. Once all the coordinate systems are selected, select OK or use the middle mouse button to accept the coordinate systems selected. If desired, enter in an internal length for the ports. This length will be added to the overall length of any wire or cable routed to the port. This allows the user to create a coordinate system at the edge of the part and not inside the part where the wire will terminate.

Finally, select a port type of either Wire, Round, or Flat. These options control the packing of the wires at the port. A Wire type will only allow one wire to be attached to the port. A Round type will allow multiple wires to attach to the port and group the wires in a circular or round pattern. A Flat type will allow multiple connections and group the wires in a flat line along the x axis of the coordinate system. Select a round type for a simple example. Select **Done/Return** from the *Cable Components* menu to finish the connector definition.

The last step is to create and route the wires or cables. This is really a two step process. This was down so that the user can create many different wires or cables and then route them all together. To create a wire, select **Feature, Create, Wire** from the *Cabling* menu and give the wire a name. Then select the spool to use form the *Spool Names* menu.

To route the wire just created, select **Route** from the *Cabling* menu. This will bring up a number of menus to select what to route. The easiest way to select existing wires to route is to use the search tool. Select the binocular icon from the top tool bar and use the search tool to select the newly created wire from the list as shown below.
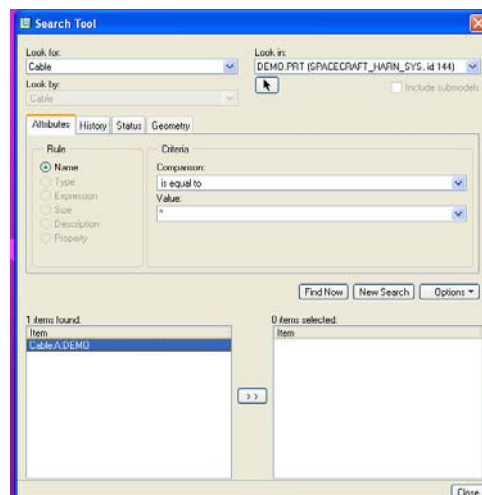
Select the wire from the list and use the >> to move it to the selected list. Select **Close** from the search tool dialog box. Select **Done Sel** from the *Cable Sel* menu to begin routing.

This will bring up the *Cbl Route* menu. The default routing option will be selected in black as **Entry Port**. This means that any entry port defined can be used as a starting point for the wire or cable. If the user does not want to start at an entry port, any of the other routing options can be selected. Select an entry port and notice a green target will be placed on the port as shown below.
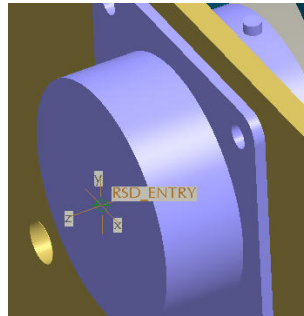
This green target is the "from" point for the route. Notice that the route option is changed to Free in the *Cbl Route* menu. This routing option allows the user to mouse pick a point on any surface to route the cable. Select a couple points for routing on any surface. Experiment with other routing options to see how easy it is to route wires and cables.
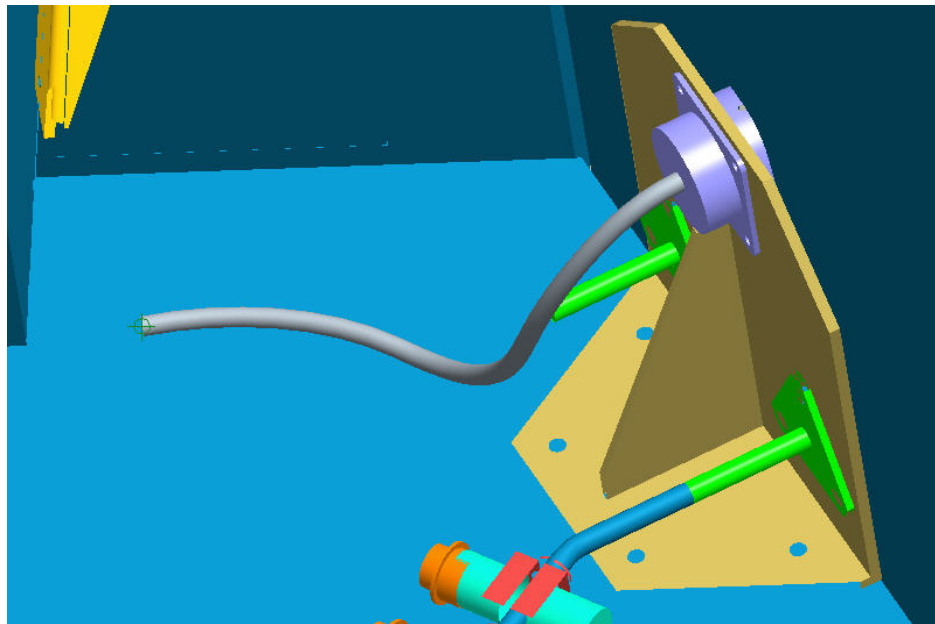
More Information on Cabling Design and other Routed Systems Products

To find out more about Pro/ENGINEER's Cabling Design Option visit our Website.

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Introducing Windchill Archive

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Tips of the Week

Part Level Simplified Representations

Many Pro/ENGINEER users' are familiar with and regularly use Pro/E's capability to create simplified versions of complex assemblies using simplified representations. However, many users do not realize that similar functionality exists to simplify parts as well. Not only is this capability useful when designing a complex part, but these simplified parts can also be used as a substitute to the fully detailed part in an assembly simplified rep. These substitutions are effective ways to represent cutouts, create zones, and show only portions of models needed for a particular design task. For this tips section I'll show a variety of part level simplified reps on the model shown below.
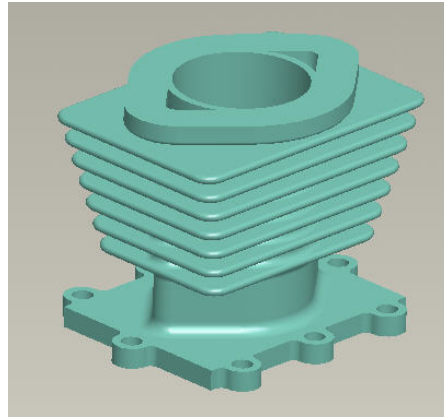
Pat level simplified reps are created by using the view manager just as assembly simplified reps are created. To get started, pull up an individual part and select the view manager icon. This will bring up the View Manager dialog box. This dialog box is nearly the same view manager dialog box as in assembly mode. The only difference will be that the Style and Explode tabs will be missing since these tabs do not apply to individual parts. The View Manager dialog box is shown below.
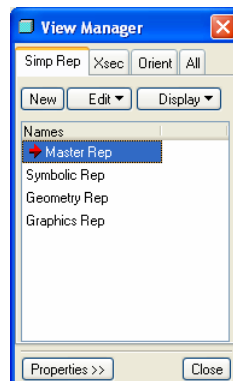
Select the Simp Rep tab near the top of the dialog box. By default, there will be four reps that already exist in the model as shown above. These reps are the master rep, symbolic rep, geometry rep, and graphics rep. These are simplified representations that are created automatically when saving a part file. Refer to PTC help for a description of each of these reps.

This tip will focus on creating three different types of reps. Select the New button from the View Manager dialog box and give the rep a name. Either select Edit, Redefine from the buttons or use the right mouse button (RMB) pop up menu to select redefine. This will bring up the Edit Method menu as shown below.


This menu allows the user to set the attributes about the part level simplified reps as well as create three different types of simplified reps. The three types of part level simplified reps are features, work regions, and surfaces. We will take a look at each one of these types of reps. Let's start by looking at the Features rep. This type of rep will allow the user to exclude features from the model. This functionality is different from simply suppressing features due to the fact that parents of features can be excluded from the rep without requiring the user to suppress the children. Select Features from the Edit Method menu. The Feat Inc/Exc menu will appear. By default the menu will be set to allowing the user to select the features to exclude. To change the default, the Attributes menu pick from the Edit Method menu can be used. For this example the default of including the features and selecting the features to be excluded will be used. Select a few features to exclude. Select the Update Screen pick to update the part with the selected features excluded.

Another way to select a set of features is to use the search tool. This tool can be used to select all features of a certain type. This is a great way to create a simplified rep that excludes all the rounds from a model. With the Exclude menu pick selected, select the binoculars icon to launch the search tool. The Search Tool dialog box will appear. Select the Type radio button and use the criteria options to set the type equal to round as shown below.
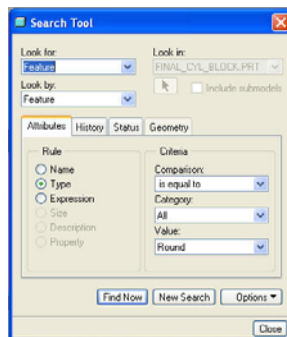

Select the Find Now button to find all the rounds in the model. The rounds will be listed in the left hand column under a title of items found. The user can select individual rounds from this list or multiple items by using the ctrl or shift key. All the items in the list can be selected by selecting ctrl A. Once the desired rounds are selected, use the arrows button to move the items from the found list to the selected column on the right hand side of the dialog box. Close the search tool and notice that the rounds are shown as excluded in the model tree. This search tool can be used in a number of ways to select a variety of features.

Once the features are selected to be excluded from the rep, select Done from the Feat Inc/Exc menu. This will bring the user back to the Edit Method menu. To finish off the rep, select Done/Return from this menu. The view manager dialog box will appear. To switch between the new rep and the master rep simply double click on the rep name from within the dialog box. The red arrow will indicate which rep is active. A simplified rep of the example model with no rounds is shown below.
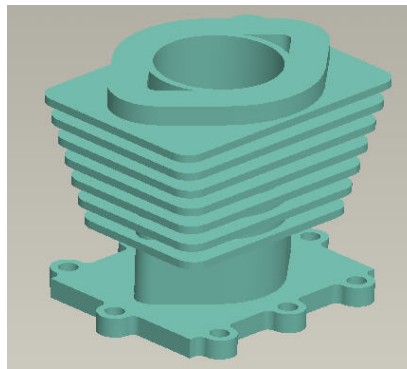

Let's look at the next type of part level simplified rep called a work region. A work region allows the user to create a cut or series of cuts to remove material from the model leaving only a section of the model to work on. This is a great technique to eliminate a portion of volume from large part models or to create cut outs to use in an assembly.

Reset the Master rep as active from the View Manager dialog box by double clicking on the Master Rep. Select to create a new rep as done before by hitting the new button in the dialog box and giving the rep a name. Use the right mouse button pop up to select redefine on the new rep name. This again will bring up the Edit Method menu. This time select Work Region from the menu.
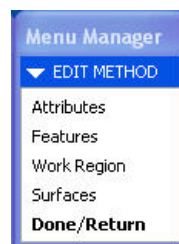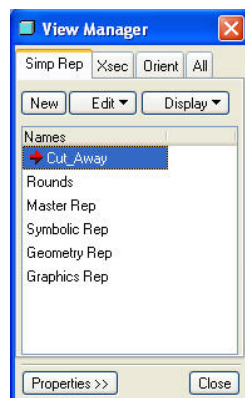
This will bring up the Solid Opts menu. This menu is essentially the old menu to create cuts from Pro/E 2001. Choose an option to Extrude, Revolve, Sweep, or any other desired creation method. Select Done from the menu. The dashboard will appear. Use the right mouse button pop up menu to define an internal sketch. Select a sketching plane and enter into the sketcher. Create a sketch to define the area to remove. An example sketch is shown below. This sketch will be extruded to create a work region of half the model.
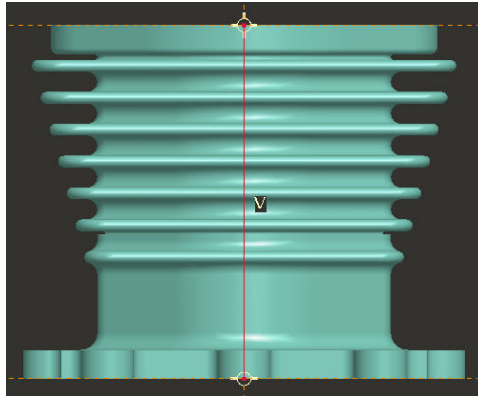

Once the sketch is created, use the dashboard to choose the extrusion options. For this example, the sketch was created on a center datum and extruded "Through All" in both directions. The options tab in the dashboard can be used to set the extrusion depths quickly and easily. Choose the green check mark from the dashboard to finish the cut. Then select Done/Return from the Edit Method menu to finish the new simplified rep. The view manager dialog box should now appear. The finished work region simplified rep is shown below.
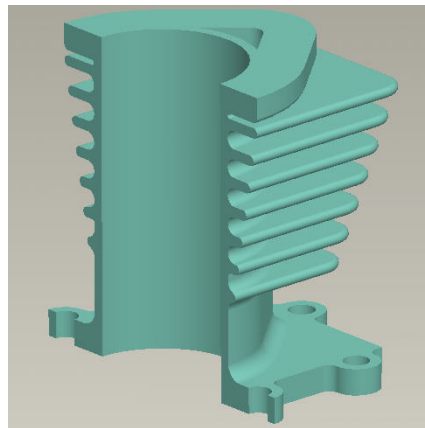
The final type of part level simplified rep we'll look at is the surfaces simplified rep. This type of rep allows the user to select a set of surfaces to be displayed for the part instead of the entire volume. These types of reps are great to use as substitutions in an assembly level simplified rep. A user can create a surfaces part level simplified rep that only contains the mating or critical surfaces of the model and substitute that rep to simplify a higher level assembly.

To start, open the View Manager and ensure that the Master Rep is set to active in the View Manager by double clicking on it (the red arrow should be pointing to Master Rep). Select the New button from the dialog box to create another new part level simplified rep. Give the new rep a name. Again use either the right mouse button pop up or the Edit pull down button to

select redefine on the simplified rep. The Edit Method menu will again pop up. This time, select Surfaces from the menu. The dashboard will appear allowing the user to copy surfaces. All surface selection methods are now available. Use the ctrl key and individually select a few surfaces. The shift key can also be used to select surfaces by either the surface and boundaries or loop methods. Once the surfaces are selected, select the green check from the dashboard to finish the surface copy.   The model should appear as only the selected surfaces. Select Done/Return from the Edit Method menu to finish the rep. An example surface copy simplified rep is shown below.
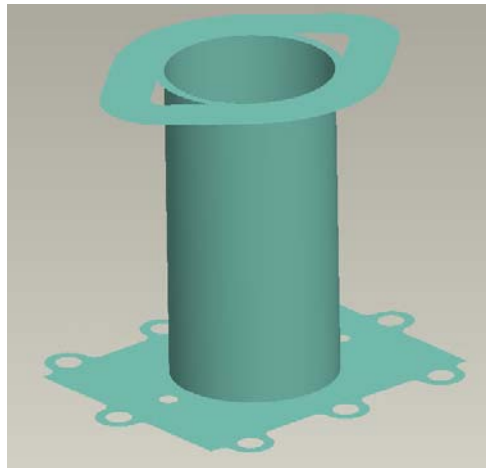

Use the view manager to switch between these simplified reps. These reps can be valuable for working on or analyzing the individual part (they can be used in Mechanica for example). As mentioned before, another powerful tool for these part level simplified reps is to use them in assembly level simplified reps. To do this, pull up an assembly that contains a part with part level simplified reps.

Once the assembly is up, select the icon to launch the view manager. Create a new simplified rep and again use the right mouse button pop up menu or the edit pull down to choose to redefine the rep. This will bring up the Edit rep dialog box with three tabs across the top. Select the Substitute tab. The By Rep method will be selected by default. Select the part that contains part level simplified reps. A list of the part level simplified reps should appear as shown below.
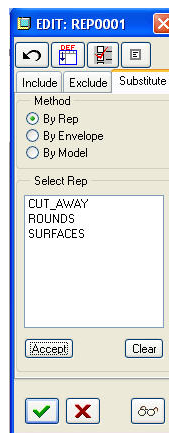
Choose the desired simplified rep and select the Accept button. The rep name should show up in the model tree next to the part name. Select the green check mark to create the rep. Multiple assembly level simplified reps can be created that reference different part level simplified reps through this substitution technique. A couple examples are shown below.
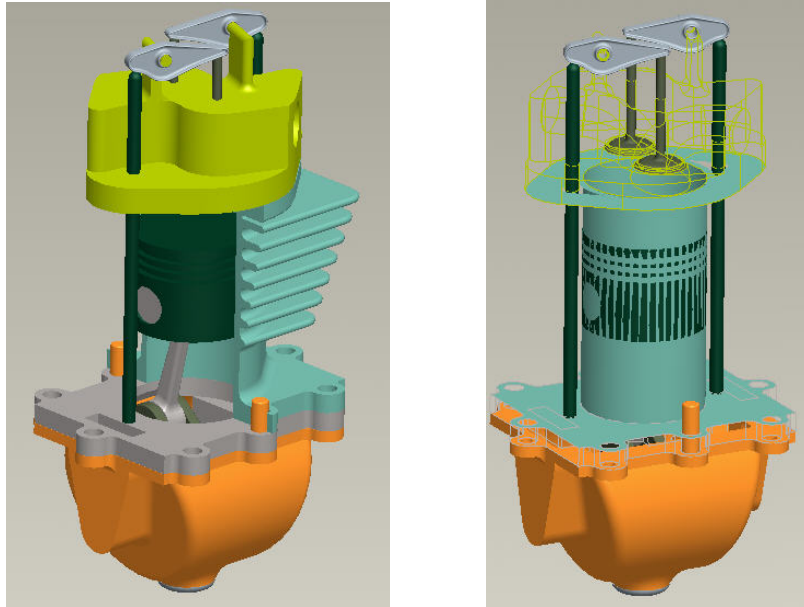
Give these part level simplified reps a try!

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Tips of the Week

How to Configure a Windchill Archive Server

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Announcements

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    - Conceptual and Industrial Design
    - Detailed Design
    - Simulation/Analysis
    - Production
    - Design Collaboration

- Windchill PDMLink

- Windchill ProjectLink

- Pro/INTRALINK

- PTC Online Tools

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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.

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Tips & Techniques: Work Smarter Not Harder!

Thinking About Pro/ENGINEER Wildfire? Check this out.

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

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, 2005   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/

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…

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- 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

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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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