Customer PTC E-Newsletter
Date: 4/9/03

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B) Oracle for Windchill : Understanding the Basic Database Structure

Tips of the Week: **A)** Associative Shrinkwrap Using Envelopes

B) Oracle for Windchill : Avoiding & Fixing Tablespace Fragmentation

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Upcoming Events & Training Schedule: Events & Training Schedule

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PTC Product Focus

Pro/Engineer ModelCHECK

ModelCHECK is an add-on for Pro/ENGINEER that will help improve the quality of CAD data and reduce the number of duplicate designs that are created in an organization. ModelCHECK will identify and provide assistance in repairing problems that make models difficult to re-use or share. Using patented technology, ModelCHECK will monitor a user's design and recognize when it starts to resemble something that has been created in the past. This means that the existing design can be re-used rather than re-created, saving both time and money.

The Challenge

Downstream Reusability

Many companies are now using Pro/ENGINEER for more than just creating models and drawings. They are reusing 3D solid models directly in downstream applications such as rapid prototyping, analysis, mold design, NC, Bill of Materials and visualization. Each of these applications is sensitive to *how* the Pro/ENGINEER model was originally constructed. Standard design techniques and 'start part' components (datums, views, and parameters) are defined by companies for Pro/ENGINEER models to ensure that these downstream applications can effectively use the models.

Design Reuse

When designing a parametric solid model, there are different techniques that can be used. In order for models to be effectively used and modified by other designers, standard best practices should be followed. This allows users, other than the original designer, to easily and predictably change models and create design variations.

Modeling Best Practices

Over 30,500 companies worldwide rely on Pro/ENGINEER and related products. During their use of Pro/ENGINEER many of these customers have developed a number of 'standard' best practices and approaches.

The Solution

ModelCHECK evaluates Pro/ENGINEER parts, assemblies, and drawings to ensure that they meet specific modeling standards and design practices.

Facilitates cost-effective re-use of models
Just as a spell checker in a word processing application helps a writer maintain the quality of a text document, ModelCHECK helps to maintain the standards of existing models as designers add new features to them.

Eliminates development of duplicate designs
ModelCHECK powerful new patent pending Shape Indexing technology helps manufacturers significantly cut costs by preventing model or feature duplication. As the user builds a new model, ModelCHECK constantly searches for similar models in the company's system. If it finds one, it tells the user where to find it and displays the two models for comparison.

Early problem detection and resolution
If ModelCHECK detects an exception to a standard, it provides online help and other tools to fix the problem and can automatically make corrections.

Easy customization and configuration
ModelCHECK can easily be customized to perform any combination of more than 130 checks to ensure that Pro/ENGINEER users build models that meet accepted standards, utilize proper modeling techniques and fulfill specific manufacturing requirements
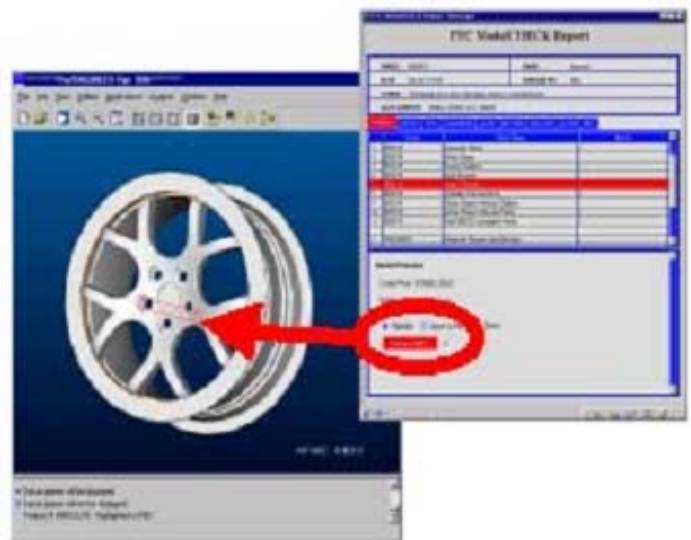
Automated tracking and reporting

ModelCHECK continually builds a database that can be used to track sort and graph the standards exceptions it finds and how frequently they occur. Management can then analyze that data to better understand the training needs of a company's Pro/ENGINEER users

New with Pro/E Wildfire

ModelCHECK reports are completely overhauled to improve ease of use and add functionality

- Reports are displayed directly in the Pro/E User Interface
- Batch reports are still displayed in a Web browser
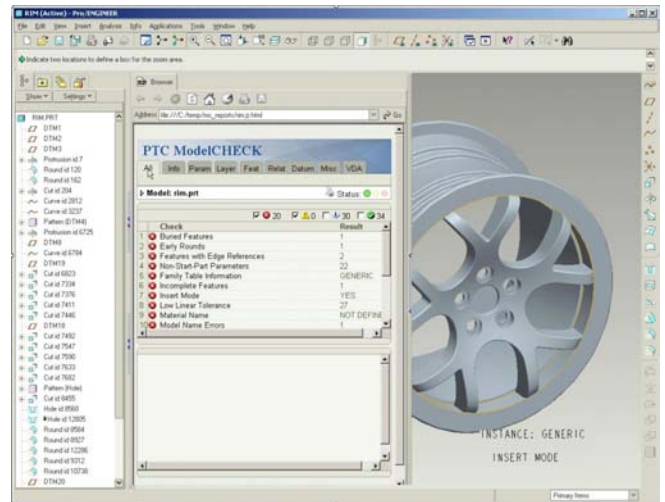
Model Status Report displays the status of the model based on defined rules (number of errors and warnings found in the model)

Green – Model is OK

Yellow – Model is OK, but user should be cautious. It would be a good idea to investigate the problems and make repairs, but it is not mandatory

Red – Model is bad and the user should investigate further and make immediate repairs

Integration with Pro/INTRALINK

- Shape Indexing information is stored in Pro/INTRALINK and ModelCHECK searches for duplicate models in the Commonspace
- Fast preview technology is integrated into Shape Indexing to allow users to preview duplicate models directly from the ModelCHECK report without opening them into Pro/ENGINEER

Improvements to batch mode

- New Distributed Pro/BATCH is completely overhauled and ModelCHECK is integrated into it
- Batch processes can be distributed across several machines
- Many more items can be automatically corrected
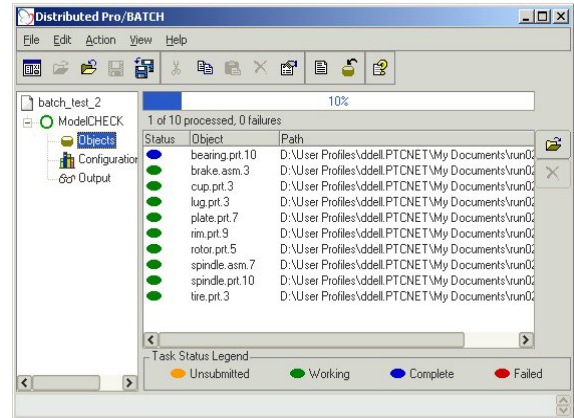
Other Enhancements

- Use part types to condition file (pipe, cable, Sheetmetal.)
  - Automatically run different configurations for different part types
- Drawing items outside of the drawing boundaries
  - Enhance to check for all items in drawings
- Report whether a family table has been verified
  - Good pre-PDM check
- Dependent copied features
  - Identify copied features and their type
- Circular references
  - Report and identify circular references
- External dependencies
  - Report and identify external dependencies
- Drawing note case
  - Check that all notes in a drawing are uppercase
- Extra items on layers
  - Items that are on a layer that they should not be on
- Datums blanked on layers
  - Check that all datum features are on layers that are blanked
- Drawing view scale
  - Verify that all views have a scale assigned to them
- Launch ModelCHECK from a Pro/TOOLKIT application
  - Functionality added to Pro/TOOLKIT to run ModelCHECK from a program
- New auto-correction and highlighting capabilities
  - Launch the global reference viewer for checks that require it
  - Highlight @O dims
  - Update density
  - Delete incomplete features
  - Reapply standard .dtl to a drawing
  - Resume erased views
  - Change accuracy type and value
  - Highlight min and max tolerances
  - …. And more

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PTC Product Focus

Oracle for Windchill : Understanding the Basic Database Structure

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Tips of the Week

Associative Shrinkwrap Using Envelopes

Introduction

An envelope is a special kind of part created by the user to represent a pre-determined set of components (parts and sub-assemblies). Substituting it in a simplified representation to replace the components it represents uses it. Envelope parts are generally created with simple geometry and take less memory than the components they represent. Envelope parts, when substituted in simplified representations reduce memory usage, while still giving the user the ability of representing the geometry of the replaced components. The envelope can be made to look similar to its components using the different options offered during envelope creation. You can use envelopes to select components in an assembly for a simplified representation.

Beginning with 2001, you can create envelopes from shrinkwraps, and you can use zones to select components and create envelope parts.

You can use envelopes in multiple simplified representations, and display them in any representation by using the visibility option. You can also create an envelope while a simplified representation is active, without the entire Master Representation being in session.

Pro/ENGINEER stores envelopes as individual part files. When it retrieves a simplified representation containing envelopes, it only retrieves the assembly file and the envelope part files. Envelope parts do not appear in the assembly BOM. They appear in the Information window along with information concerning the part geometry and the list of reference components. Envelopes must have both geometry and a (non-empty) list of reference components.

Example

Here is an assembly of an automobile transmission. It consists of 423 unique parts and takes about 226 MB of Ram to load the Master Rep. into memory. The goal is to simplify the assembly reducing the amount of memory used thus facilitating working with the model.
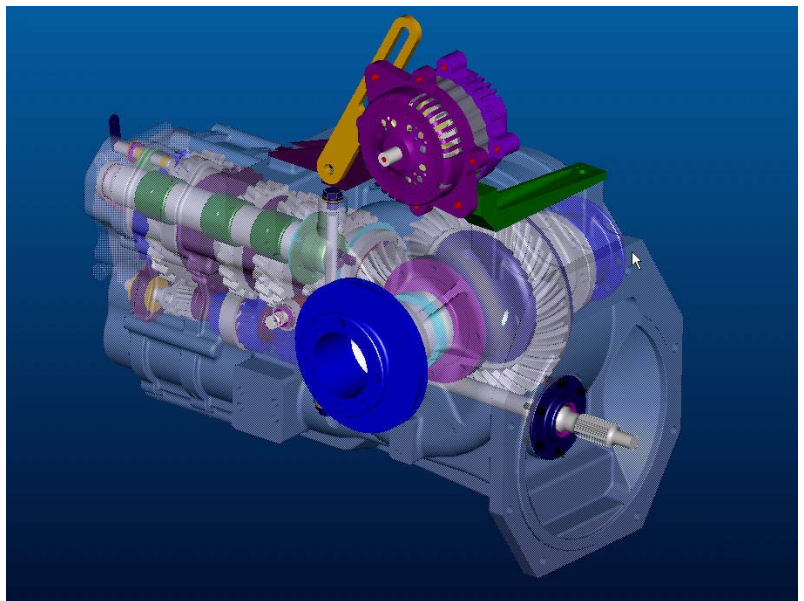
It is best to work with subassemblies. In this case a simplified represntation is required of this subassembly. Currently it is not required in the master assmbly of the transmission so it will be substituted with and envelope. To create an envelope, use the **Envelope** command in the **ASSEM SETUP** menu or **DESIGN MGR** menu as shown below.
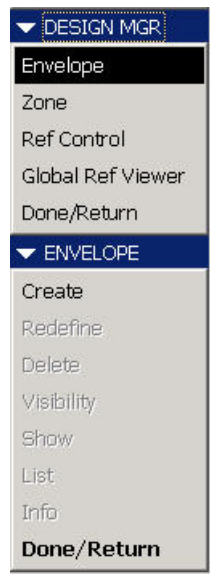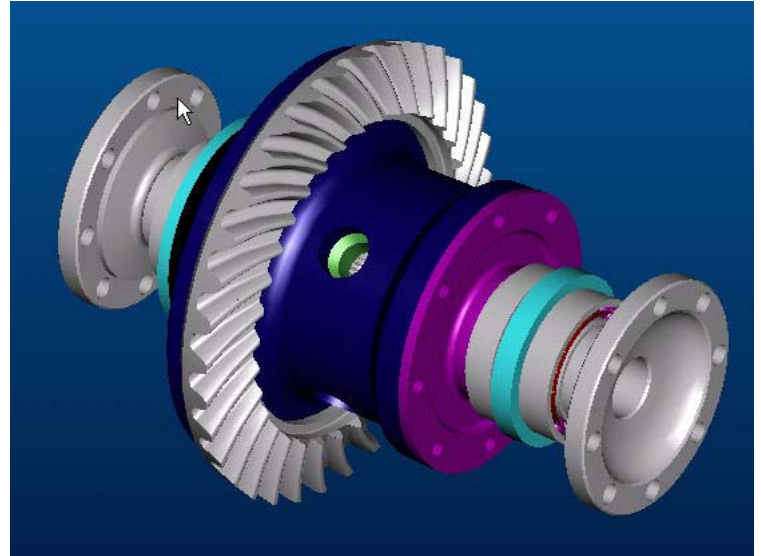

After naming the Envelope (above) the menu on the left asks what parts are to be included. To select every part in the subassembly use the **All** command and **Done** to finish.

The following window will appear as shown here. Select **Surface Subset Shrinkwrap** method. Enter in a descriptive name and choose the Quality level. Also note the other options available. For more information on ShrinkWrap visit the following links.
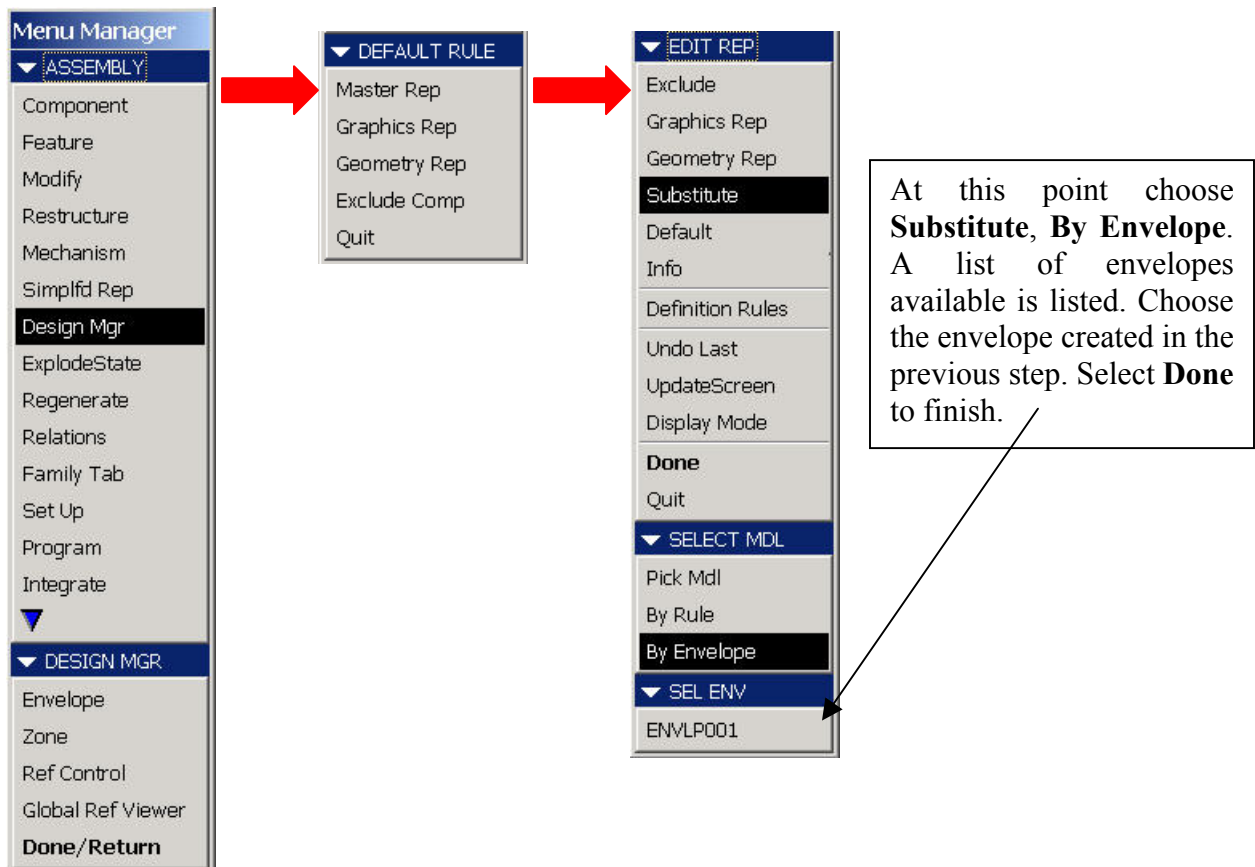
About ShrinkWrap Models

To Create a ShrinkWrap Feature
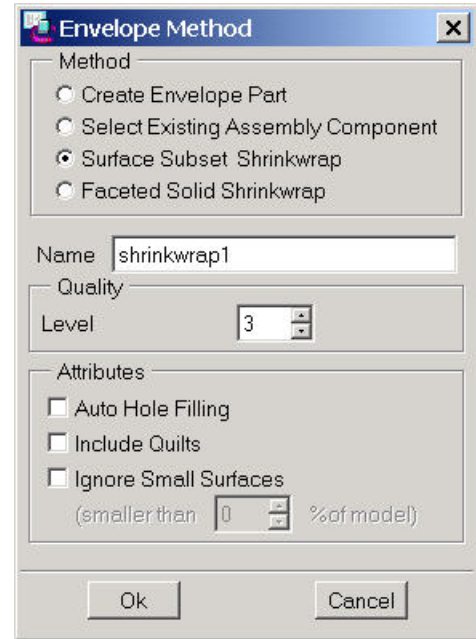
At this point the envelope is created and what remains is to create a Simplified Representation that will substitute the entire subassembly with the envelope part.

To create a Simplified Representation select **Simplfd Rep** from the main menu and then choose **Master Rep**. See the diagram below.

At this point choose **Substitute**, **By Envelope**. A list of envelopes available is listed. Choose the envelope created in the previous step. Select **Done** to finish.

Returning to the main assembly of the transmission the sub-assembly can now be substituted for the Simplified Representation just created. The procedure is the same, as substituting for an envelope except the subassembly needs to be selected and a list of Simplified Reps. will be shown. The Simplified Representation called Rep0001, which is the envelope part, replaces the Subassembly 01-51200.ASM as seen below. The transmission doesn't look any different but by checking the memory usage the current Simplified Rep. Takes about 15 less Mb of RAM than the Master Representation.
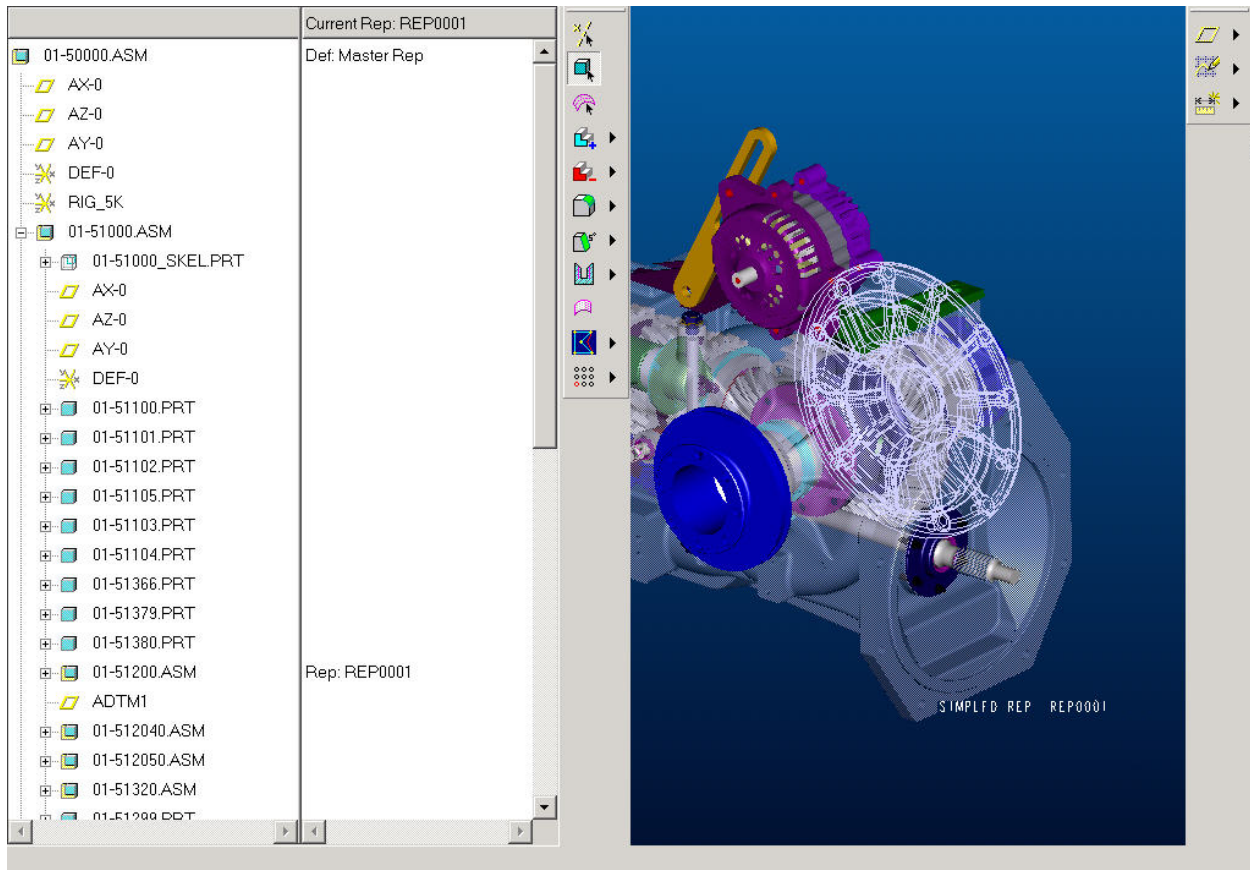
Conclusion

Envelopes are a great way to simplify very large assemblies while still keeping the geometry visible on the screen. In conclusion the following benefits can be derived from using this technique.

- Minimize memory usage
- Visualize very large assemblies
- Full associatively between the envelope and the part/assembly

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Tips of the Week

Oracle for Windchill : Avoiding and Fixing Tablespace Fragmentation

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Announcements

Special Hardware offers for customers updating to Pro/ENGINEER Wildfire

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

Click on the poster to gain access to the Product First Virtual Event resource center!! See the software and pick up a screen-saver or two.
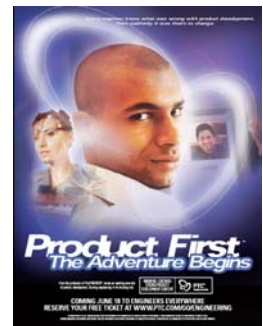
E-PROFILES IS HERE!!
We have been eagerly anticipating the debut of the new electronic version of Profiles Magazine and now it is here! This new web site will supplement the print edition of the magazine and will provide new useful features not feasible with paper media. e-Profiles will provide you with 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, 2003   Your local Pro/Engineer User Groups
                                  http://www.prouser.org/rugs/

June 8 - 11, 2003   Orlando, FL
                                  Pro/USER International Conference
                                  http://www.prouser.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/E 2001 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 products 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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