Customer PTC E-Newsletter
Date: April 30, 2004

PTC Product Focus: **A)** Reverse Engineering Extension - RESTYLE

                                  B) Windchill 7.0 What's new with Visualization

Tips of the Week: **A)** 1. Creating a Multi-Configuration BOM Report Table

                                      2. Using the Associative Topology Bus to achieve backwards
                                         compatibility between Pro/E Wildfire and 2001

                                  B) Windchill 7.0: How to include reference and describing documents in the ProductView structure for visualization and enhanced batch printing

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PTC Product Focus

Pro/ENGINEER® Reverse Engineering Extension™ (RESTYLE)

Enable engineers to create accurate, modifiable Pro/ENGINEER models from 3D scan data quickly while capturing design intent and maintaining critical geometric data.

Today's reverse engineering solutions tend to lose any design intent that may be included in a surface quilt. For example, imported surfaces may be tangent at the adjacent boundaries but these attributes are lost on import so they will not be recognized by downstream applications. Restyle will capture critical design intent and this is accomplished inside of Pro/ENGINEER ensuring any subsequent changes are seamlessly propagated throughout the design.

Obtaining the Competitive Advantage Using RESTYLE

      Many companies have some of their most successful products represented in a physical medium only. These physical representations do not have the flexibility to be modified or improved quickly and efficiently and because of this companies suffer major losses in productivity. Designing an existing product from scratch or wasting time on unnecessary design iterations is undesirable to say the least in today's market place. Restyle fills the need for a product that not only creates high quality, manufacture-able surfaces from scanned data but also significantly automates that process. Simplify the workflow, reduce training time and allow the designer to focus on the task at hand.

Restyle allows customers to reverse engineer or design products to fit existing physical models as well as thrive in industries where mass customization is required. Restyle has combined proven technology from Raindrop Geomagic with Pro/ENGINEER.

Restyle contains cutting edge tools to work with and refine point cloud and polygonal data ensuring optimum results in subsequent phases of the process. These tools reduce noise and/or the total number of points to speed up processing time without sacrificing the level of detail in the original design model. Triangles may be deleted or refined in order to eliminate any error that may have occurred during the wrapping process without compromising surface integrity or detail.

Once an acceptable facet model has been created, simple surfaces such as cylinders and cones or complex surfaces can be created with ease. Surfaces can be projected to fit facet data or boundary surfaces created from curves sketched on the facet model. Once a surface is created the full set of Pro/E tools are available for surface analysis and to check for any deviation between the surface and facet model. Surfaces may also be edited to change properties such as surface type (NURBS/BAZIER), U/V parameters, etc. Surfaces can be manually manipulated in a free form or numerical manner for fine-tuning the final model. The result is a fully modifiable, highly customizable Pro/E surface representation of a product that fits right the first time.

The Advanced Functionality of the Reverse Engineering Extension Includes:

Point Cloud Refinement
- Point cropping
- Noise reduction
- Automatic deletion of outlying points
- Point sampling (Random, uniform and curvature sampling)
- The ability to fill holes in scanned data

Curve Creation
- Curve on facet
- Curve on surface
- Section
- From Edge
- Analysis Extremum
- Analysis Isoline
- From border
- From sharp edges
- Through points

Facet Modeling
- Facet filling operations
- Cleaning operations
- Decimating operations
- The ability to refine facets
- The ability to relax facets
- Automatic removal of webbing

Surface Creation
- Plane
- Cylinder
- Cone
- Revolution
- Extrusion
- Four curves rectangle
- Three curves triangle
- Surface from net
- Three curves rectangle
- Surface from box
- Four points rectangle
- Four points cross
- Two curves rectangle
- N curves loft

- Removal of facets as singles, groups or user defined cropping
- Make manifold operations

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PTC Product Focus

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Windchill 7.0 What's new w/ Visualization

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Tips of the Week

## Creating a Multi-Configuration BOM Report Table

Creating a Multi-Configuration BOM Report Table

In a Pro/E Drawing, an assembly drawing format can be modified so that the BOM Report Table can reflect multiple configurations of an assembly based on family table instances of an assembly or based on separate assemblies.

The following example shows how to create additional columns in a table to reflect the multiple configurations:

- Open an assembly drawing format and zoom in on the BOM Table.

- Edit the BOM Table to add a Column or multiple columns if multiple configurations are needed. If a Quantity column already exists add the new columns adjacent to the existing quantity column. Select Table, Insert, Column and select a location on the existing table.
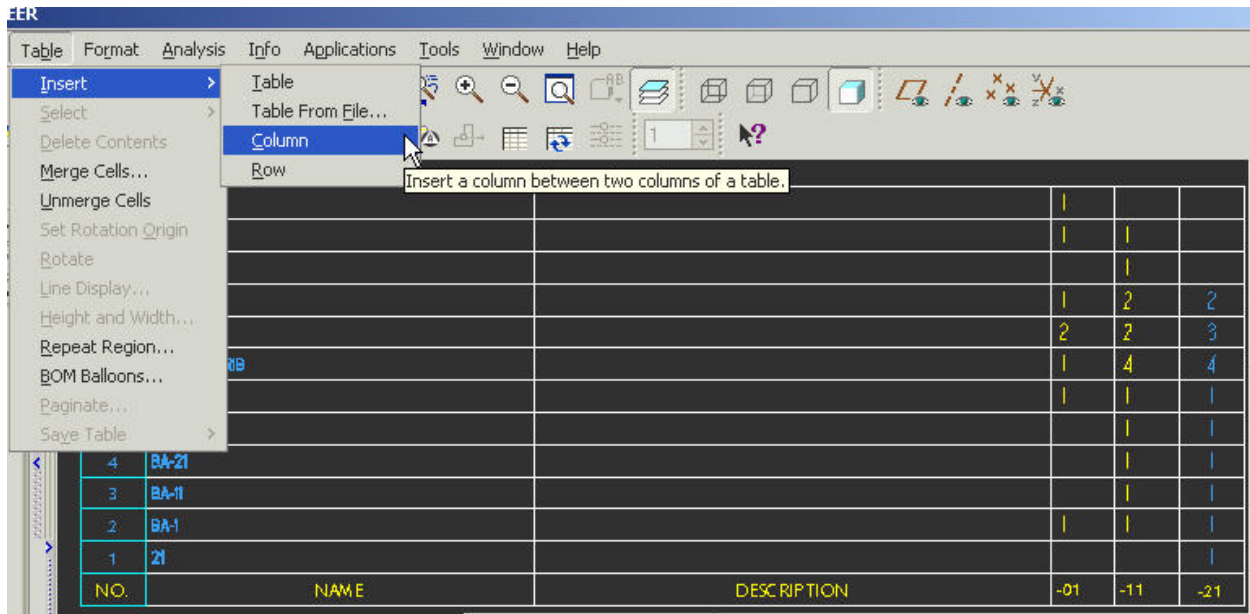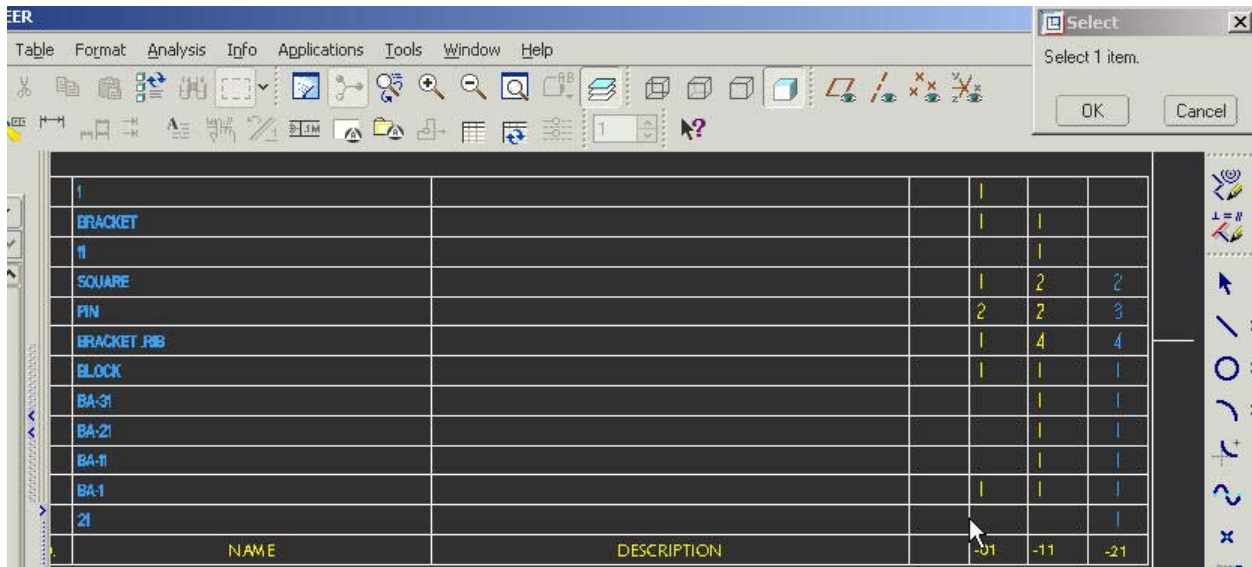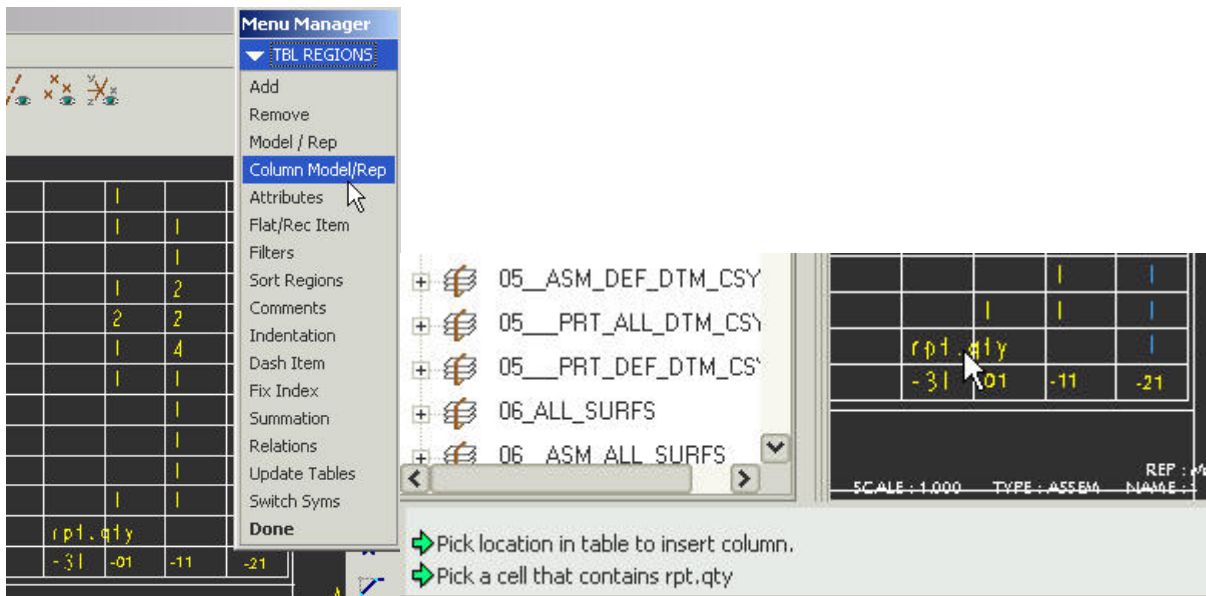
- On the top or bottom row of the new column, depending whether the table is ascending or descending, add text which describes the configuration (-11, -21, -31, etc). Select the block in the column, press the Right Mouse Button and select Properties. Enter the desired text. Select OK on the Note Properties window.
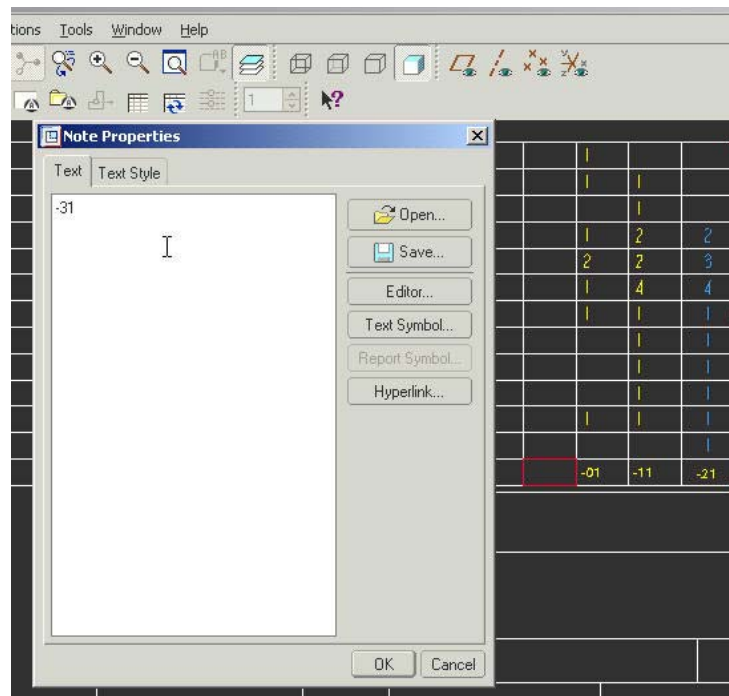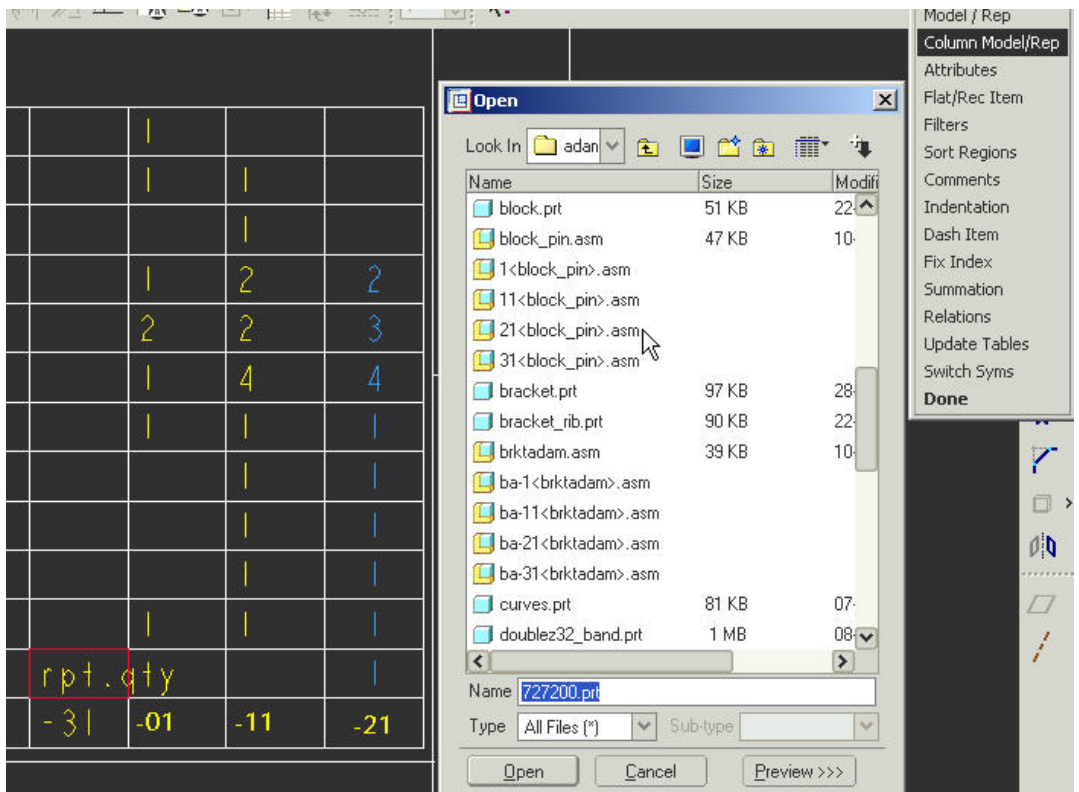

- On the next row, up or down, add a report symbol which will define the information to populate the column. Select the block in the column, press the Right Mouse Button and select Properties. Select Report Symbol and then select RPT and QTY from the tables displayed. Select OK on the Note Properties window.
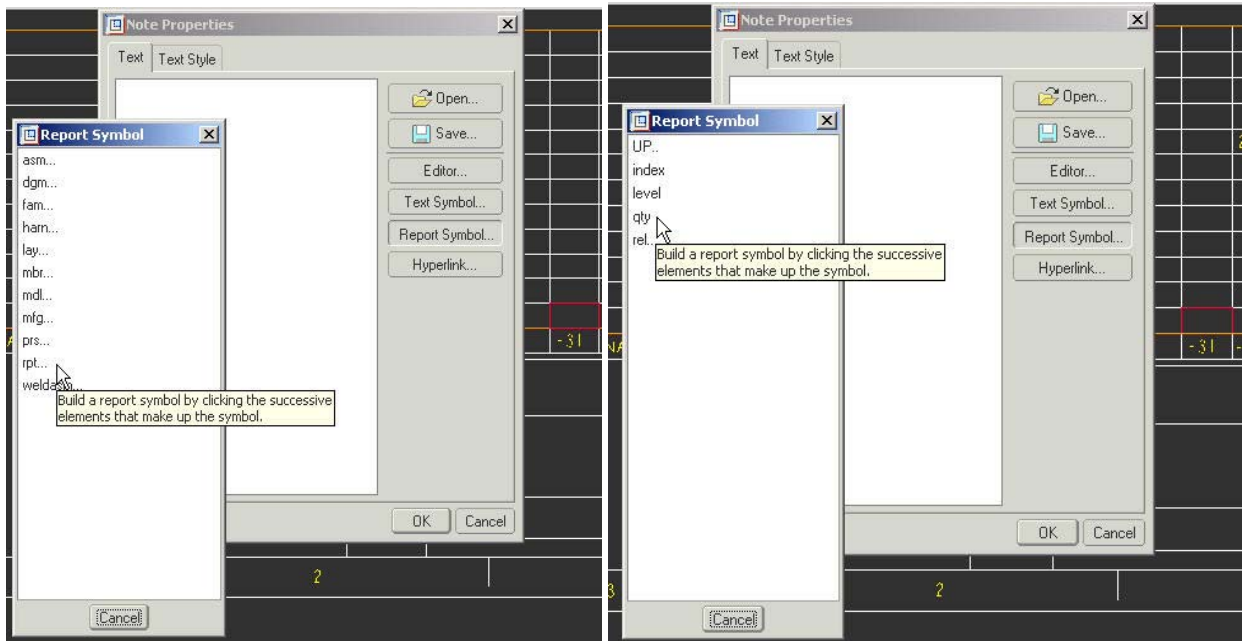
- Use Repeat Region Switch Dims to see the report symbols in the table.

- To complete the column, select Table, Repeat Region and Column Model/Rep. Select the block in the new column of the table which contains RPT.QTY. The system will then prompt the user to browse for and select the instance of the assembly or new assembly to be represented by this column. Select Confirm.

Model / Rep
Column Model/Rep
Attributes
Flat/Rec Item
Filters
Sort Regions
Comments
Indentation
Dash Item
Fix Index
Summation
Relations
Update Tables
Switch Syms
Done

Open

Look In adam

Name	Size	Modifi
block.prt	51 KB	22
block_pin.asm	47 KB	10
1<block_pin>.asm		
11<block_pin>.asm		
21<block_pin>.asm		
31<block_pin>.asm		
bracket.prt	97 KB	28
bracket_rib.prt	90 KB	22
brktadam.asm	39 KB	10
ba-1<brktadam>.asm		
ba-11<brktadam>.asm		
ba-21<brktadam>.asm		
ba-31<brktadam>.asm		
curves.prt	81 KB	07
doublez32_band.prt	1 MB	08

Name 727200.prt
Type All Files (*)   Sub-type

Open   Cancel   Preview >>>

rpt.qty

-31	-01	-11	-21

1.

NO.	NAME	DESCRIPTION	-31	-01	-11	-21
12	1					
11	BRACKET					
10	11					
9	SQUARE				2	2
8	PIN			2	2	3
7	BRACKET RIB				4	4
6	BLOCK					
5	BA-31					
4	BA-21					
3	BA-11					
2	BA-1					
1	21					

## Using the Associative Topology Bus to achieve backwards compatibility between Pro/E Wildfire and 2001

Using the Neutral file export and import format along with the Associative Topology Bus (ATB) capabilities users can transfer data back to 2001 format from Pro/E Wildfire and maintain an associative link for updating the parts to account for subsequent changes made to the parts in Pro/E Wildfire.

The following example shows how this would work:

- Add "TOPOBUS_ENABLE   YES" to your config.pro file.
- With a session of Pro/E Wildfire active, Open a part and select File, Save a Copy and select Neutral file under Type.
- Select OK to export the data as a Neutral file.
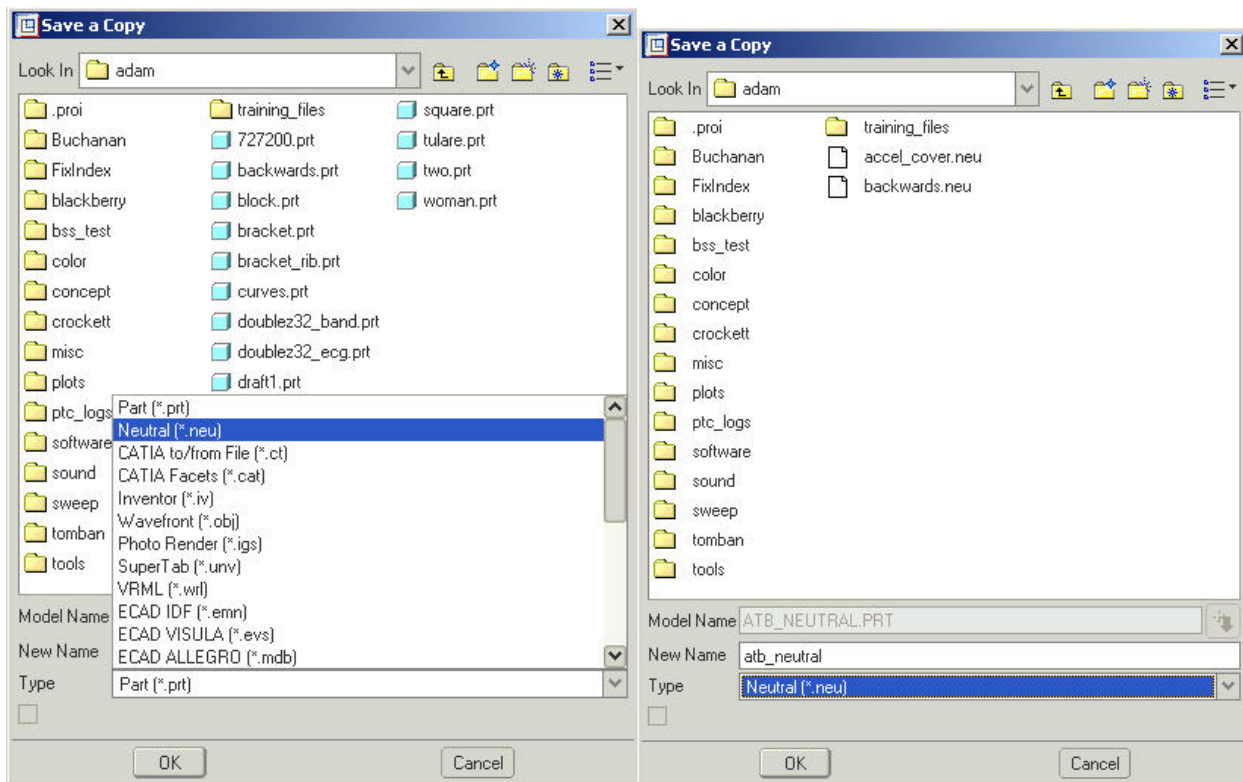
- Select a Coordinate System or accept the Default to complete the export process.
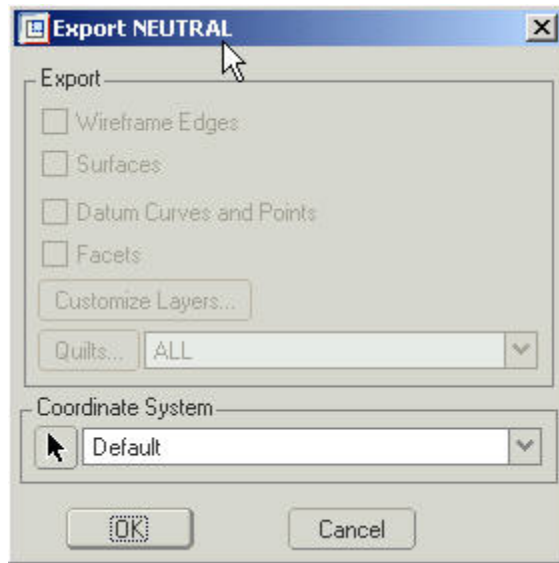
- Within a session of Pro/E 2001, select File, Open and set Type to Neutral.
- Highlight the file on the list and select Open.
- From the Import New Model dialog box, select Part or Assembly and select OK to complete the import.
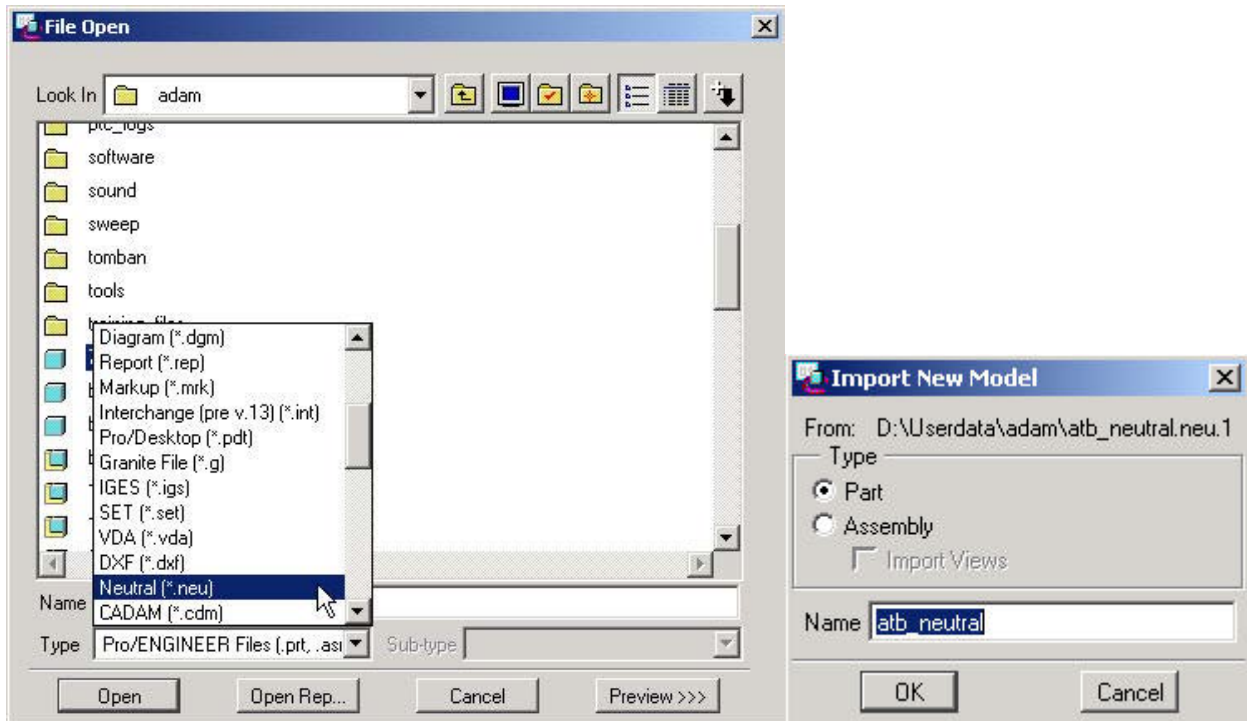
- Return to Pro/E Wildfire, Open the original part, make a change and Save the part.

- Select File, Save a Copy and select Neutral File under Type. Use the same name as the first neutral file which was exported.
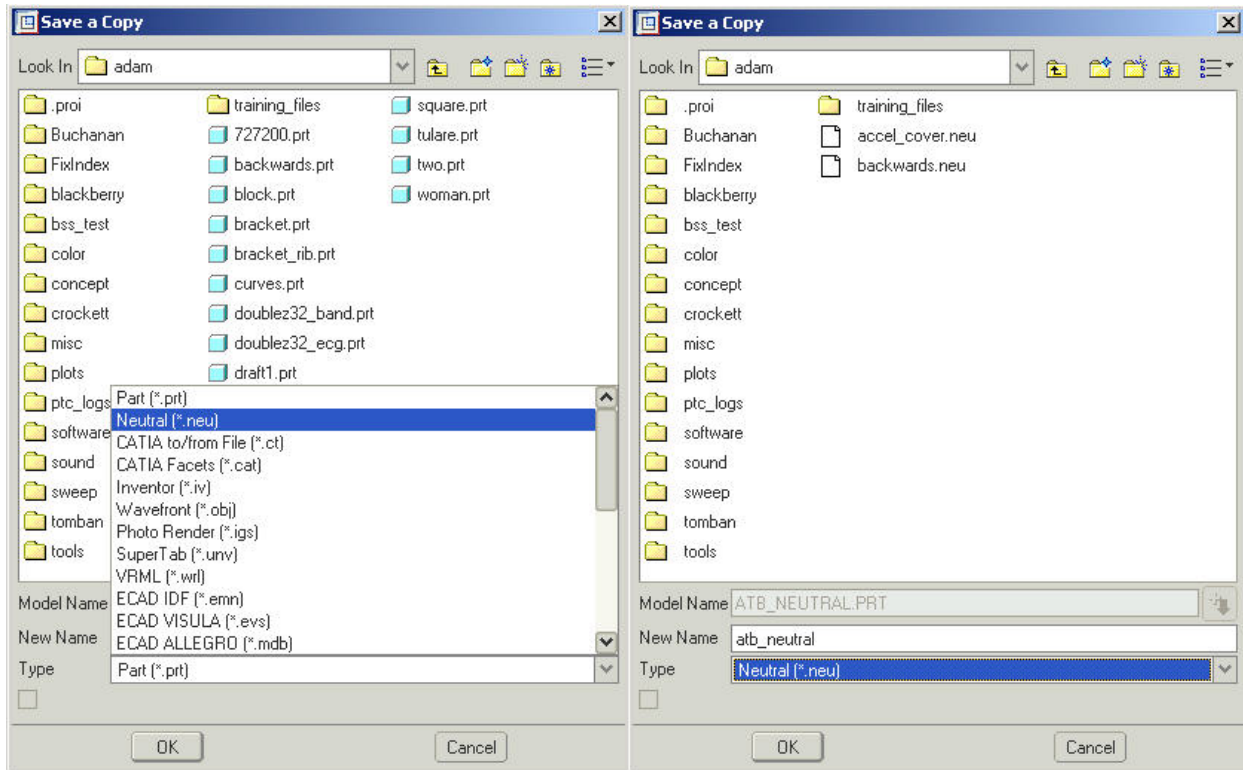
- Return to Pro/E 2001. With the imported part active is session, select File, Associative Topology Bus and Check Status. The Information Window will provide data informing the user if changes have been made to the model in Wildfire (ie: changes to the neutral file).
- Select File, Associative Topology Bus and Update All.

```
INFORMATION WINDOW (topo_bus.log.2)                              _ □ ×

File  Edit  View

===========================================================================
            TOPOBUS: Verifying outdated imported geometry (04/05/2004 11:32:31)
===========================================================================

Part : PRT0001
Source System                   : NEUTRAL
Full Path to Referenced Model : D:\Userdata\adam\atb_neutral.neu.3
                                    last imported         current
Source Sys Rev  of  Ref Model : 111                  145
File Date of Referenced Model : 04/05/2004 11:21:22  04/05/2004 11:32:18
Number of Surfaces            : N/A                  N/A

                              [Close]
```
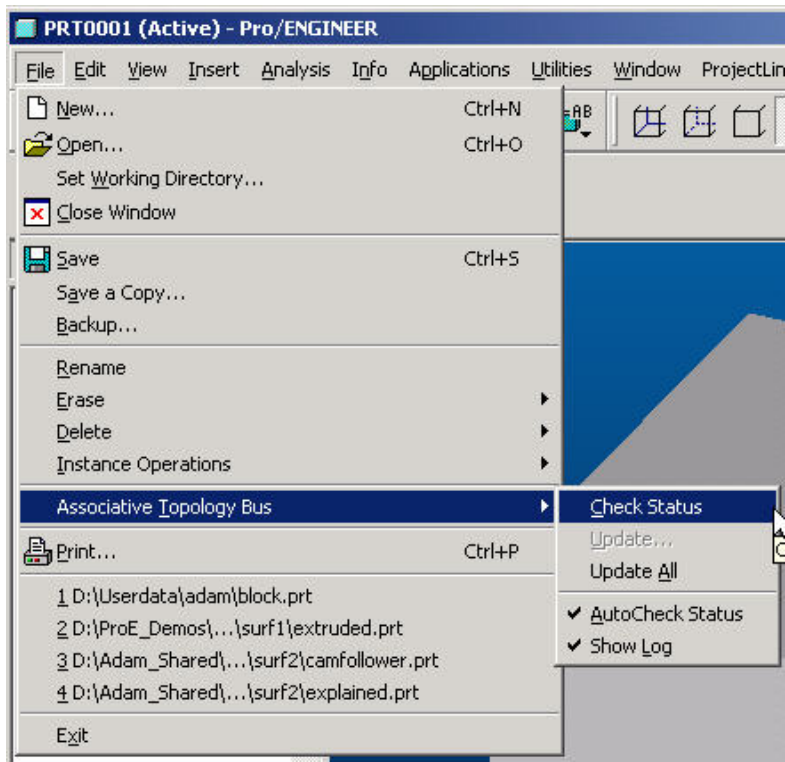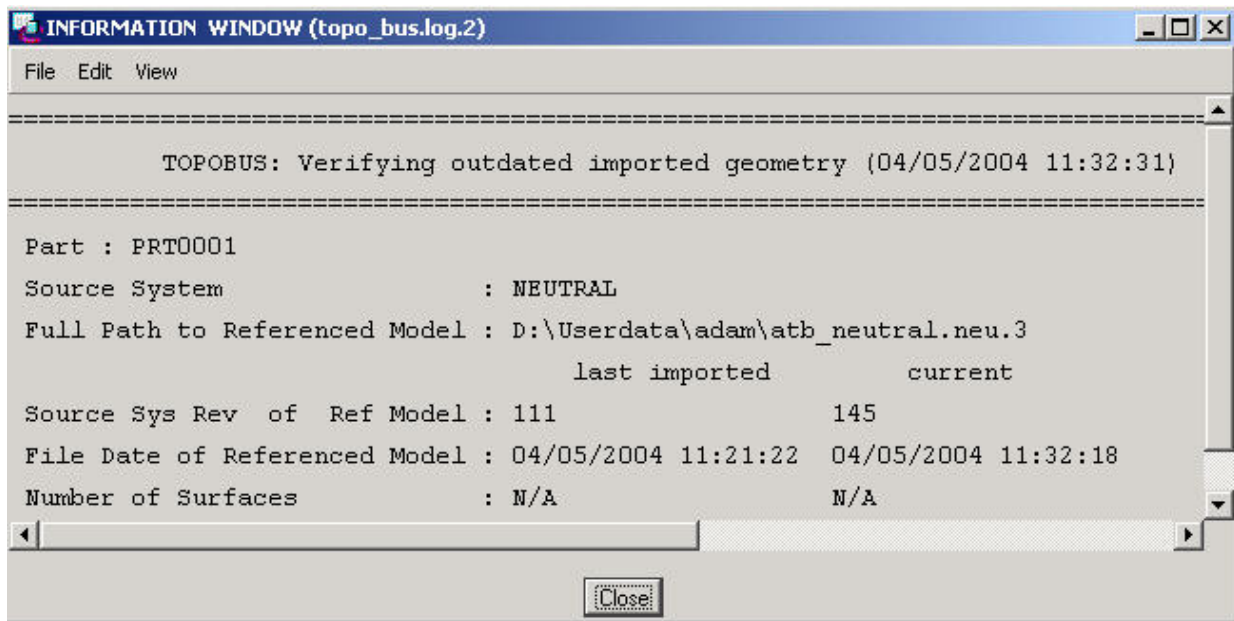
Note: This process bridges the gap when design is being done in Pro/E Wildfire but a vendor who is doing manufacturing or analysis is using Pro/E 2001 and has yet to upgrade to the new version.

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Tips of the Week

Windchill 7.0: How to include reference and describing documents in the ProductView structure for visualization and enhanced batch printing.

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Announcements

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!

http://www.ptc.com/appserver/it/icm/cda/template_lib/events/series.jsp?&im_dbkey=11442&icg_dbkey=141

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

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

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. 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.prouser.org/rugs/

June 13 - 16, 2004   Nashville, TN
                        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/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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