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PTC Product Focus

Pro/ENGINEER Wildfire 2.0, Why Upgrade?

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PTC Product Focus

ProductView Composer Option

ProductView Composer allows you to create and deploy interactive assembly, disassembly, inspection and maintenance sequences of critical processes to product lifecycle stakeholders. Composer sequences combine process steps and parameters with powerful 3D graphics, allowing end users in manufacturing, QA and maintenance to quickly understand how to carry out procedures on your products. Composer authors can organize ideas and options to support deferred design decisions and flexibility throughout the design, manufacturing and support processes. Composer is an optional module for ProductView Standard Edition visual collaboration tool, Composer complements your existing Windchill® Product Development System environment.

Composer allows you to import and edit existing sequences from Pro/Process for Assemblies, for rapid dissemination to the enterprise. Or Composer will allow you to visually capture the process of assembling, disassembling, inspecting or servicing a product in ProductView. You create a script, which is a sequence of steps that represent particular actions in the overall process. You can define instructions, camera angle, variants (scenarios), and other parameters for each step.

A script can contain sequences representing assembly or disassembly. Sequences provide ways to group steps together. Sequences can contain steps and scenarios. Instructions and parameters

can be defined for any node in the Composer tree. Each step can contain a key to specify the action (capture) and camera angle.

You can use Auto Run to automatically step through each of the keys, displaying the process. Auto Run plays the current Sequence in the order depicted in the tree. This allows you to smoothly transition views from one key to the next in the Sequence.

Composer Interface

The tree view displays a Script node, which can contain steps and scenarios. Each Step represents an element in the sequence. Steps are played in the order in which they appear in the tree. Steps can contain key(s) and instructions. Keys are where you define the camera angle and components, along with the action taken, such as translate, assemble, etc. Nodes in the tree can be used to group items such as steps or sequences. You can drag nodes in the tree to reposition them in the sequence.

The Notes tab lists the instruction files available in the sequence. The Scripts tab lists the Composer files contained in the assembly that is open in ProductView. Parameters are shown at the bottom of the Composer window. Parameters contain name/value attributes that can be edited.

Defining Scenarios

In addition to defining the basic steps in the sequence, you can also create Scenarios, which are customized steps or sequences that are based on specific alternatives that affect the way the process occurs. For example, the assembly process might vary based on elements such as the type of disk drive. In this case, you would specify scenarios for each different disk drive. Then, when the assembly takes place, the person can specify the type of disk drive and the assembly sequence for that variant will apply. Scenarios contain steps, sequences, or other scenarios.

Composer Benefits

- Reduce the time required to understand product assembly sequences, reduce time in training, and provide better supporting information
- Reduce time and money spent creating and printing engineering drawings for signoff processes and for use in manufacturing
- Gain early visibility of manufacturing processes based in CAD data that is still Work-In-Progress minimizing the time and costs associated with any Design For Manufacturing changes
- Reduce chance of rework by graphically representing manufacturing, maintenance or inspection sequences
- View Composer digital instructions without the Composer module, i.e., shop floor users can view instructional information using ProductView Standard Edition (The Composer option is only needed for editing Pro/Process for Assemblies imported instructions or Authoring/Editing new Sequences or Scenarios.)

For more details about creating digital instruction with ProductView Composer, please visit the Product area of ptc.com or contact your PTC Account Manager.

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Tips of the Week

Understanding the Search Tool in Pro/ENGINEER Wildfire 2.0

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Tips of the Week

Model Feature Dimensions Are Only a Double-Click Away

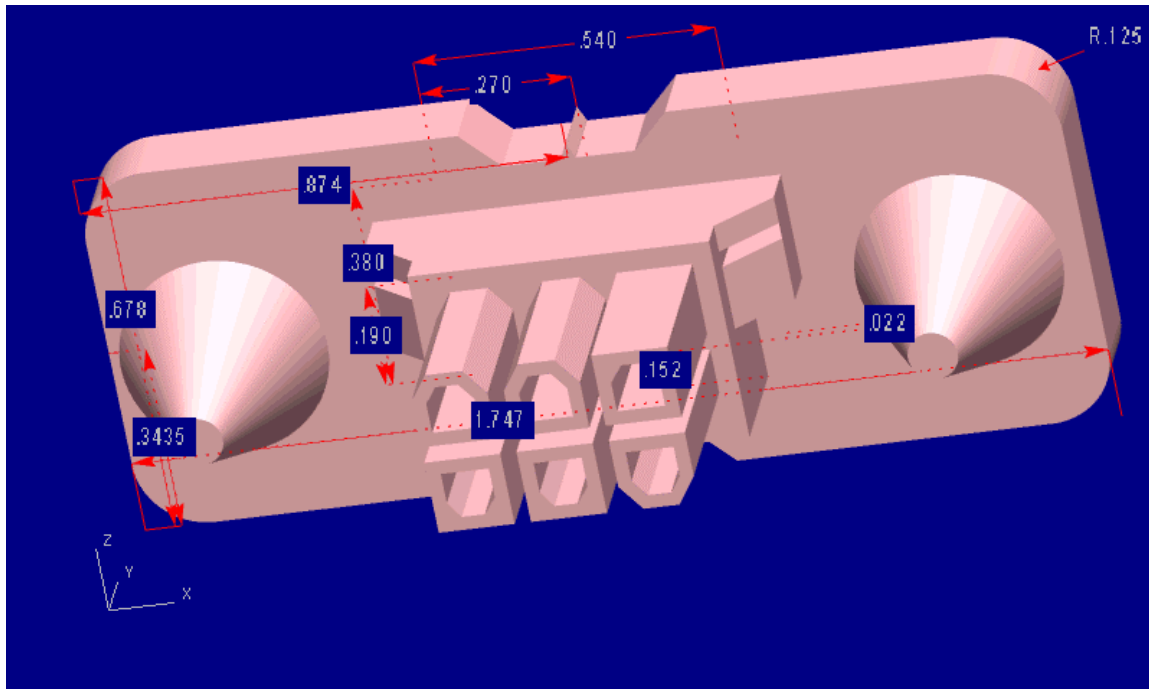
ProductView lets you view the annotations that are associated with features. These annotations are part of the file that is brought into ProductView from Pro/ENGINEER, UG, SolidWorks and I-DEAS NX. You can view, but not create, these annotations using ProductView. Feature-based annotation can include dimensions, surface finish notes, geometric tolerance, and 3D notes. (You may need to adjust the recipe file to publish this information. See Recipe editing notes below.)

To view feature-based annotation, follow these steps:

NOTE: When the data is initially loaded all of the feature dimensions will be displayed by

default. To hide them select all of the components in the view . Use the **RMB Menu > Properties** to display the components property panel. Click the Layers Tab. Uncheck the DIM Layer and click close button.

1. To activate select feature mode, choose **Edit > Select > Feature (Shift+T)**. You must enable feature selection before you can create feature-based annotations.
2. Press Shift to highlight faces or curves that have feature information associated with them. This highlighting is similar to measurement snap modes or pick targets.
3. Click to select the feature you want to annotate.
4. Click on the feature again to display the feature-based annotation (Double-Click). You can click to toggle this display on and off. The types of features that appear are defined using the context menu or preferences.



The selected Annotations and View state can be saved as an annotation set and can be accessed during collaboration sessions.

Edit > Component > Clear Feature Based Annotations will remove them from the view.

5. (Optional) To adjust the types of annotations that appear, right-click to display the context menu. Check the annotation type(s) that you want to appear when feature selection is enabled. Feature annotation types include dimensions, surface finish notes, geometric tolerance, and 3D notes. Note: You can specify the default feature-based annotation types using the Feature Based Annotation page in Preferences.

The context menu indicates the current type of annotation selected, and this is a toggle. You can click to select a different annotation type or keep the type as is. Annotation information will reflect the current selection. For example, when a 3D note is attached to a feature, you can right-click that feature to display a 3D note context menu. Once a feature is selected, you can still toggle each annotation type using the context menu; however, only the annotation types you specify in Preferences will appear by default.

Feature Based Annotation Preferences Page

To specify preferences for feature-based annotation, use the Feature Based Annotation page in the Preferences dialog box. Remember that you can use ProductView to view, but not create feature-based annotation. This information is taken directly from the file as it came from Pro/ENGINEER, UG, I-DEAS, SolidWorks.

1. Right-click in the viewer and choose Preferences.
2. Click the Feature Based Annotation page.
3. Check the annotation type(s) to specify the default types of feature-based annotation that will appear when feature selection is enabled and you make a selection:

- Dimensions
 - Geometric Tolerances
 - Surface Finish Notes
 - 3D Notes
4. Check Display Feature of CAD annotations if you want feature-based annotations to appear for CAD files.

Annotation information will reflect the current selection. For example, when a 3D note is attached to a feature, you can right-click that feature to display a 3D note context menu. Once a feature is selected, you can still toggle each annotation type using the context menu; however, only the annotation types you specify in Preferences will appear by default.

To display the annotations created on the selected feature, select the component that contains the feature.

Using the Recipe Editor to enable Feature Publishing

How To Publish Graphical Entities

By default, the CAD adaptors publish solids, quilts, curves, and wireframe information. Additional information such as 3-D notes, dimensions, datum curves, among others are not published by default, but can be enabled as follows:

- Run rcpedit at the command line.

Running the RCP Program

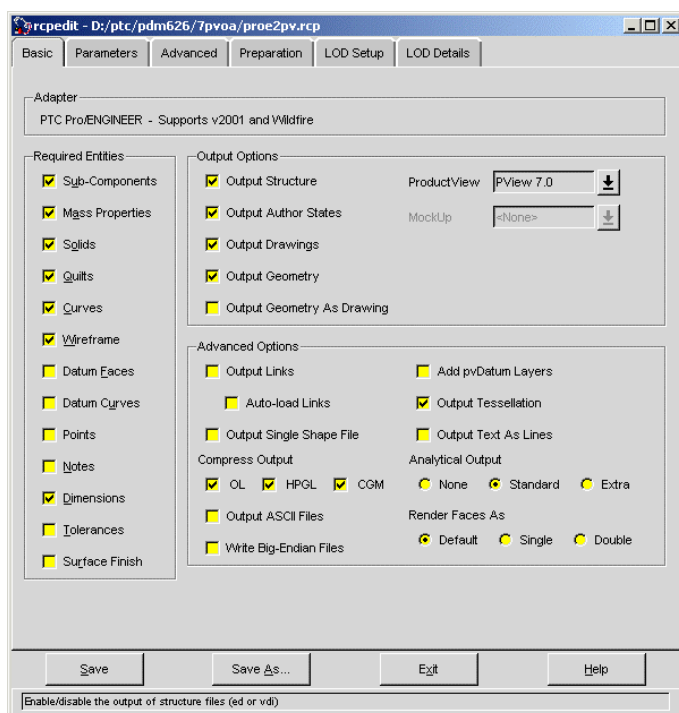
The recipe file is located in the directory where <cad>worker script was created. In that directory you should find <cad>2pv.rcp recipe file. This is the recipe file associated with the conversion process.

To access this file and run the RCP program, type the following command line:

```
<loadpoint>\bin\rcpedit <worker installation loadpoint>\<cad>2pv.rcp
```

Edit the recipe using the Graphical User Interface (GUI) that displays when the program opens. Click Save to save the default file. Click Save As to save a new override RCP file. Click Exit to quit the recipe edit

- On the Basic tab, select the various graphical entities you wish to convert. The defaults and Dimensions are shown selected (checked).



Note: In order to publish, then view additional feature-based annotations, select the additional settings in the required entities of the Basic tab of the recipe editor: Notes, Tolerances, and Surface Finish. Enabling these features allows the viewing of those annotations in the ProductView client.

- Save recipe file.

How to View Datum Information

By default, datum entities (including dimensions, tolerances, notes etc.) are written to special pvDatumXYZ layers. These are separate from any layers specified in the source CAD data.

If, however, you wish to keep the datum information on the original CAD layers they are assigned to or if you already have a pvDatumXYZ layer definition in your CAD data, you can disable the auto-creation of these pvDatumXYZ layers following the steps shown next:

- Run rcpedit at the command line.
- On the Basic tab, in the Advanced Options panel, **uncheck** the Add pvDatum Layers check box in the Output Options section. (See image above)
- Save the recipe file.

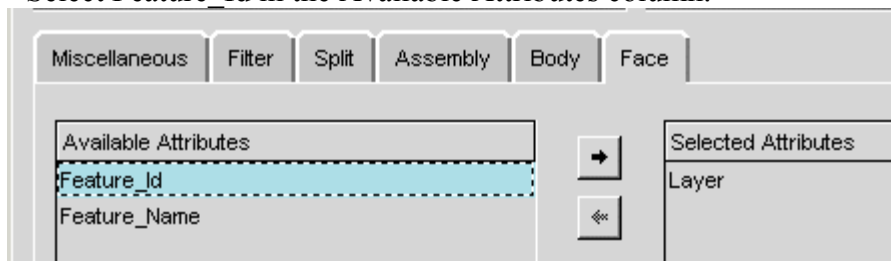
The ProductView client lets you view the annotations that are associated with features. These annotations are part of the file that is brought into the ProductView client for visualization. The annotations are not created in the ProductView client, but are viewed using this feature.

Feature-based annotations can include dimensions, surface finish notes, geometric tolerances, and 3-D notes.

Surface finish is a measure of the deviation of a part surface from its normal

value. In other words, the surface finish note specifies a value for the roughness of the surface. Surface finishes are abstracted from the CAD system. Enabling this setting allows the storing of the surface finish values in the converted .ol file. To enable this feature, follow the next instructions:

- Run `rpedit` at the command prompt.
- Select the Parameters tab.
- Select the Face tab.
- Select `Feature_Id` in the Available Attributes column.



- Click on the add button. This moves the selected item to the Selected Attributes column.
- Save the recipe file.
- Reconvert data.

ProductView Standard Edition Feature Based Annotation Preference Settings

To specify preferences for feature-based annotation, use the Feature Based Annotation page in the Preferences dialog box. Remember that you can use ProductView to view, but not create feature-based annotation. This information is taken directly from the file as it came from CAD.

1. Right-click in the viewer and choose Preferences.
2. Click the Feature Based Annotation page.
3. Check the annotation type(s) to specify the default types of feature-based annotation that will appear when feature selection is enabled and you make a selection:
 - Dimensions
 - Geometric Tolerances
 - Surface Finish Notes
 - 3D Notes
4. Check Display Feature of CAD annotations if you want feature-based annotations to appear for CAD files.

Annotation information will reflect the current selection. For example, when a 3D note is attached to a feature, you can right-click that feature to display a 3D note context menu. Once a feature is selected, you can still toggle each annotation type using the context menu; however, only the annotation types you specify in Preferences will appear by default.

To display the annotations created on the selected feature, select the component that contains the feature.

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Announcements

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PTC Tips & Techniques Newsletter Archives

Miss an issue! Can't find that awesome technique you read about? Fear not, you can click on the link below and go through our Customer PTC E-Newsletter archives.

[Click Here To Access](#)

It's better than finding the Covenant of the Ark!

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!

In addition to our Creation Tips & Techniques, here are some new events related to our Collaborate and Control solutions.

[August 19 - Tips & Techniques for Windchill ProjectLink Users](#)

[August 25 - An End User's Guide to Maximizing the Usability of Windchill Solutions](#)

Keep on the look out for additional events related to Windchill ProjectLink, Windchill PDMLink or Windchill in general coming soon!

Click below to see other regularly scheduled Tips & Techniques technical Webcasts that are designed to provide you with the most popular time-saving tricks that PTC solutions users of all skill levels will find useful. Get more out of your maintenance dollars!

[Tips & Techniques: Work Smarter Not Harder!](#)

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.ptcuser.org/rugs/>

June 5 - 8, 2004 Orlando, FL
PTC/USER International Conference
<http://www.ptcuser.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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