



- PTC Product Focus:** A) [Multi-Site Pro/INTRALINK](#)  
B) [Windchill Workgroup Manager for Pro/ENGINEER](#)
- Tips of the Week:** A) [Pro/LOCATE](#)  
B) [Creating Pro/E Template Content for New CAD Docs](#)
- Announcements:** [Most Recent Announcements](#)
- Upcoming Events & Training Schedule:** [Events & Training Schedule](#)

## PTC Product Focus

### Multi-Site Pro/INTRALINK

[Click Here To View](#)

[Back To Top](#)

## PTC Product Focus

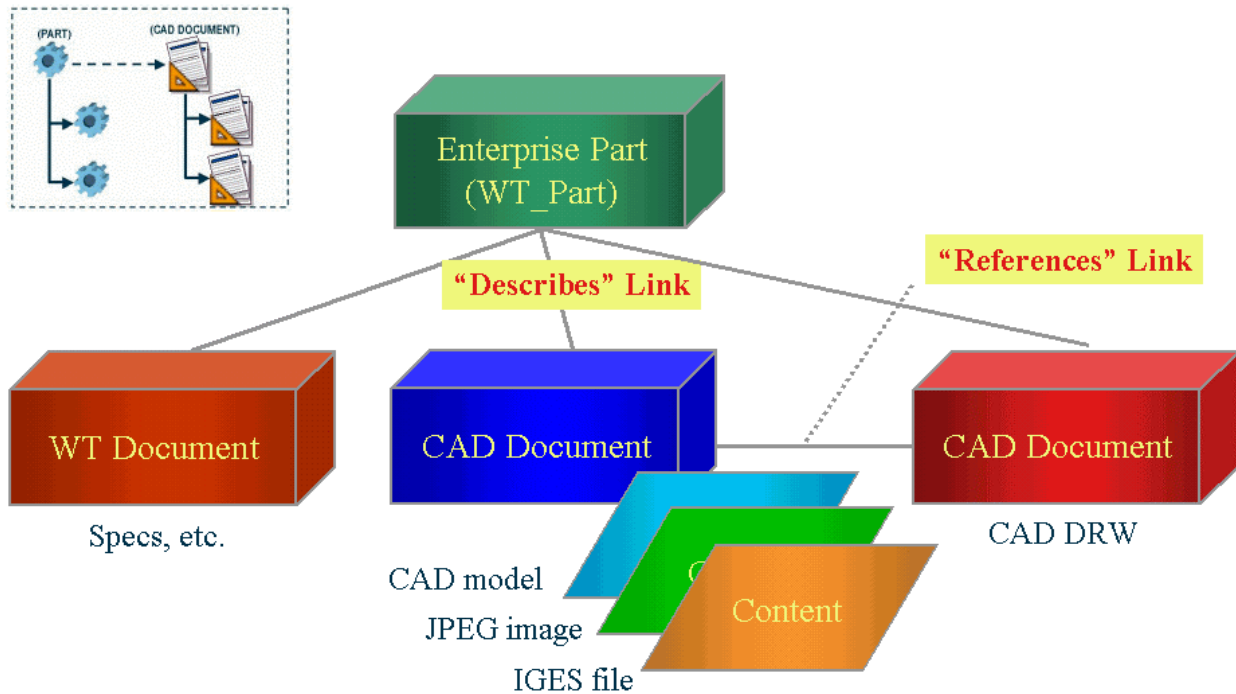
### Windchill Workgroup Manager for Pro/Engineer

Windchill CAD Integrations – CAD to CAD Collaboration

Your company's engineers work in Pro/ENGINEER, but your outsourcing partners work in AutoCAD and Solidworks. With Windchill CAD integrations, also referred to as the Workgroup Managers, users can now access product information, review work in progress data or make engineering changes from within the Windchill environment, no matter what their CAD package. The Windchill Visualization allows users to share files and collaborate on them together, without the need for native design content.

The Workgroup Manager for Pro/ENGINEER enables Windchill users to directly manage product modeling and engineering information created using Pro/ENGINEER. You can build enterprise-level product information models by creating association links between Pro/ENGINEER objects (CAD models, CAD assemblies, and Drawings) and Windchill-managed enterprise objects and product structures. Windchill-managed enterprise objects can contain additional information used in the definition of the product information model (such as technical specifications, process plans, routing documents, and engineering change orders). See Figure 1 below.

The Workgroup Manager for Pro/ENGINEER ensures engineering data security and automatically makes this information available for use by other applications in the Windchill enterprise.

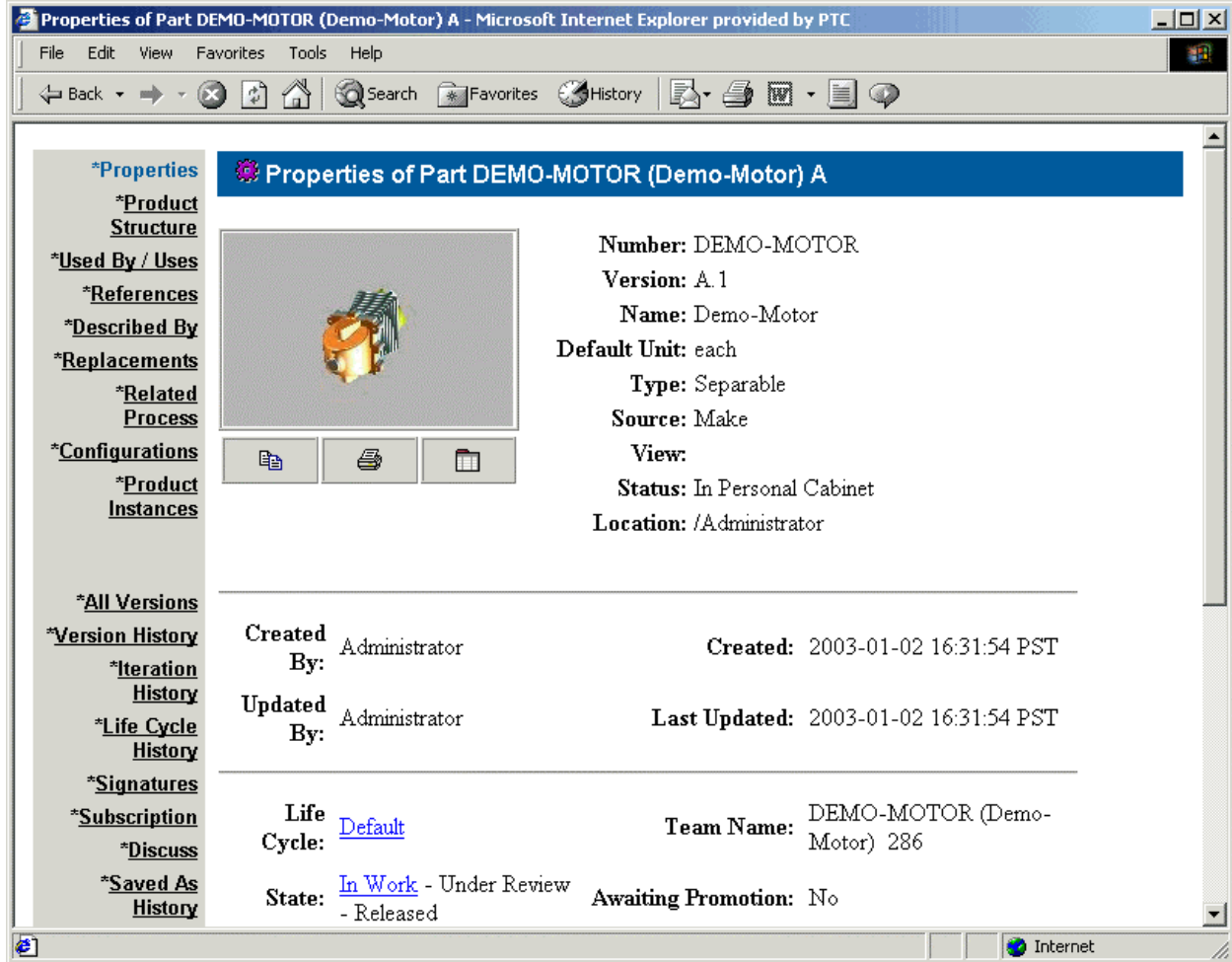


**Figure 1 - WTPart & CAD Document Links**

You can use Workgroup Manager for Pro/ENGINEER to:

- Manage broadly defined engineering data and concepts, such as parts, assemblies, form, function, and fit.
- Perform design automation integration by sharing a common manipulation mechanism with Windchill. This allows you to manage Pro/ENGINEER model creation, updating, and association with enterprise parts.
- Revise WTParts and CAD documents through the Workgroup Manager for Pro/ENGINEER user interface.
- Manage heterogeneous information from multiple product design authoring systems.
- Work with custom WTParts that have enterprise-specific behaviors.
- Recognize configuration-based life-cycle states, baselines, and effectivities using standard Windchill configuration management capabilities.
- Move configuration-managed design into engineering.
- Open an existing or start a new Pro/ENGINEER session from the Workspace Browser.

- Share product information throughout your enterprise.



**Figure 2 - Windchill Properties Page for a Part**

[Back To Top](#)

## Tips of the Week

### Pro/LOCATE

[Click Here To View](#)

[Back To Top](#)

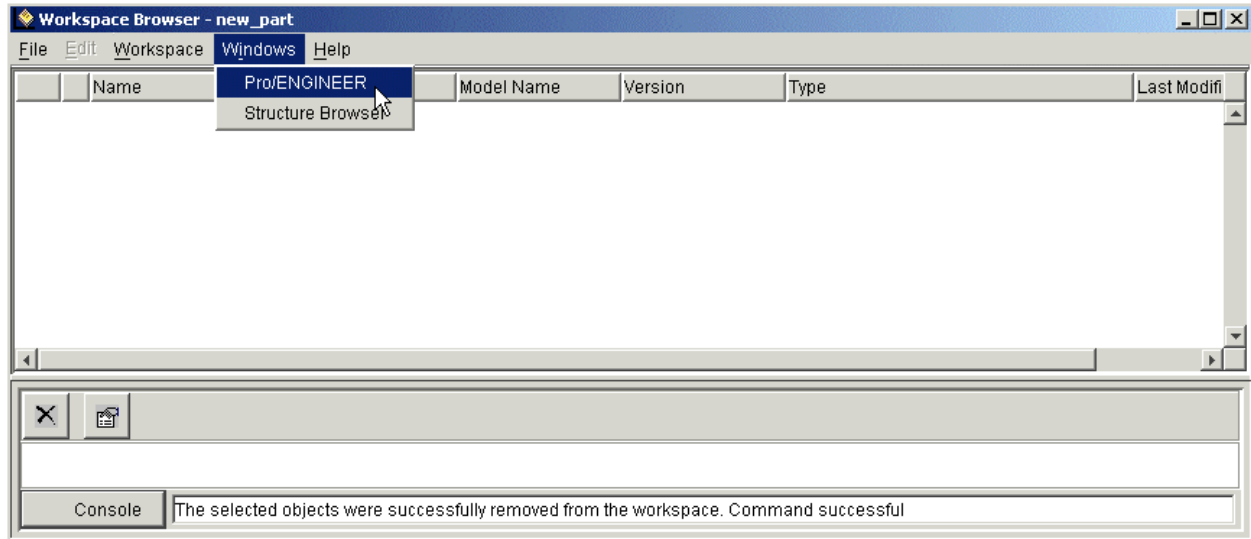
## Tips of the Week

### Creating Pro/E Template Content for New CAD Docs

The Workgroup Manager for Pro/ENGINEER (WGM ProE) will allow you to create new CAD Documents, consisting of Pro/ENGINEER parts, assemblies, and drawings. It has been standard practice to use start parts, assemblies, and drawings when creating Pro/ENGINEER data. The Workgroup Manager for Pro/ENGINEER uses templates to define the creation of new Pro/ENGINEER data. The following describes how to get your start parts configured as the standard templates when creating new CAD Documents for Pro/ENGINEER objects. You will need administrative privileges in Windchill and access to the Windchill server machine to configure the CAD templates.

To have your start parts reside within Windchill for usage when creating new CAD Documents, the start parts must be reside in Windchill. You will need to create a shared domain and folder to store the template documents. From the Windchill Administrator page, create a domain. Then create a shared folder in this shared domain for placement of the template parts.

Start the Workgroup Manager for Pro/ENGINEER. Open or create a Workspace. Once inside the workspace, start Pro/ENGINEER.



**Figure 3 - Starting Pro/ENGINEER from Workspace**

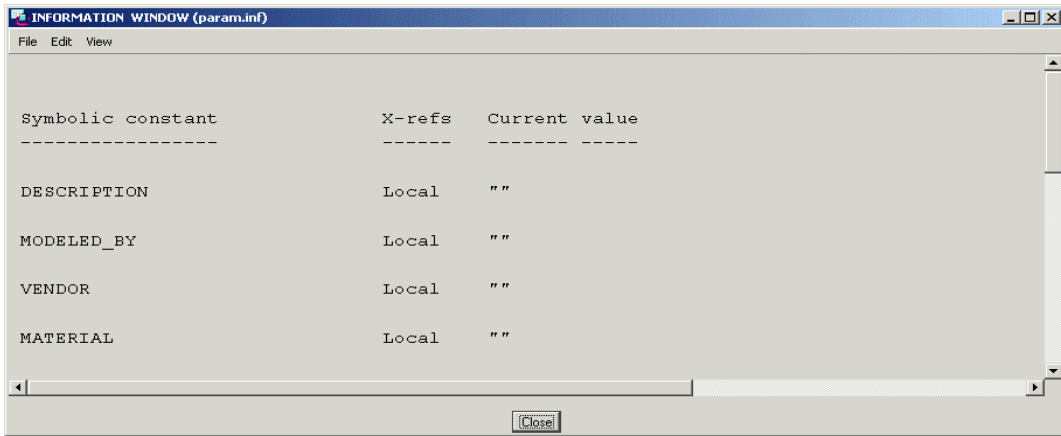
When you are in Pro/ENGINEER, either create or open your start part in this linked session of Pro/ENGINEER. Add necessary datum features, notes, parameters, geometry, etc. All this information will be saved as the template and used when creating new CAD Documents for Pro/ENGINEER parts within the Workgroup Manager for Pro/ENGINEER. It is recommended that you name the Pro/ENGINEER models as the following:

- Start Part ⇒ Template.prt
- Start Assembly ⇒ Template.asm
- Start Drawing ⇒ Template.drw

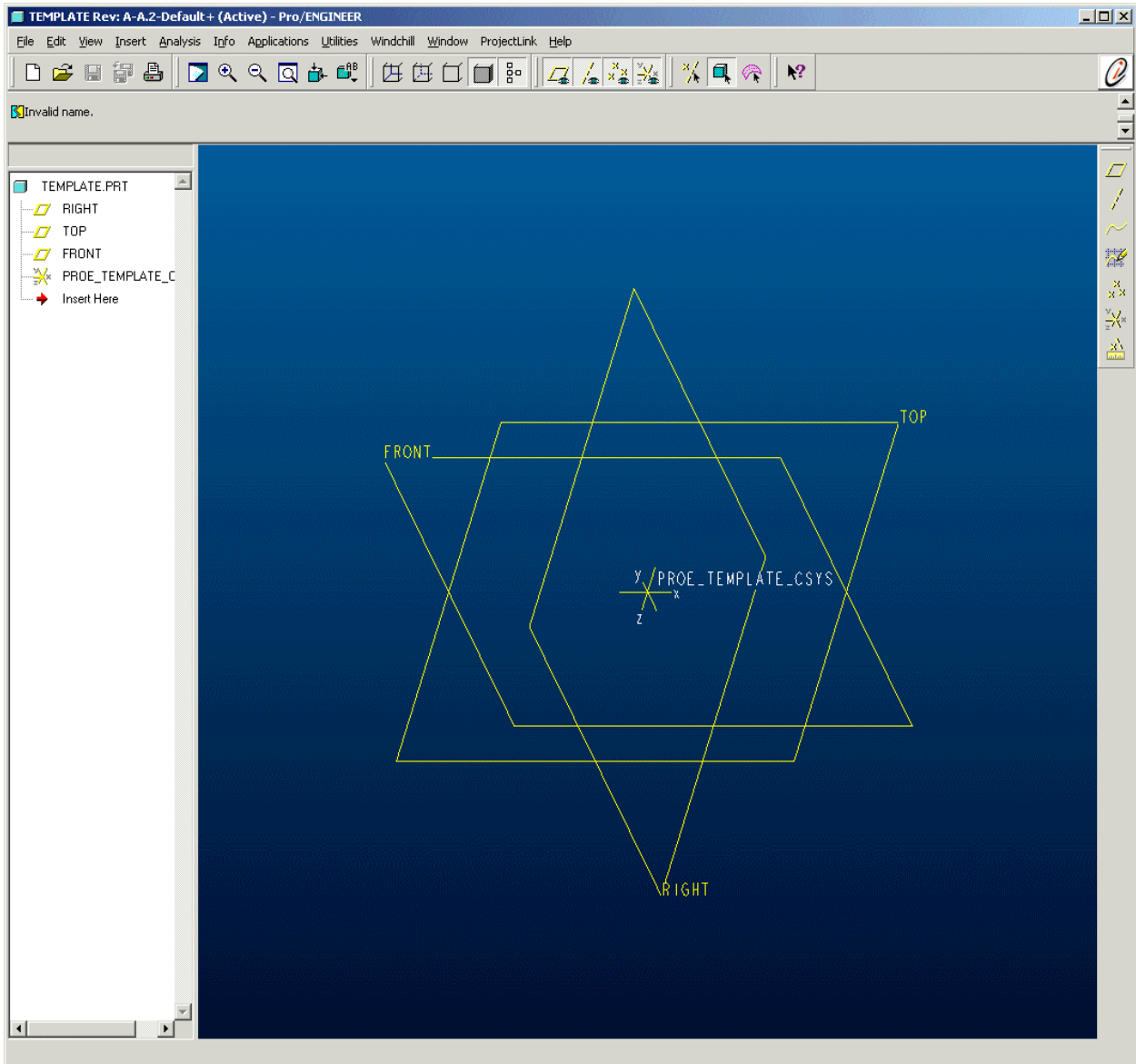
If you want Pro/ENGINEER parameters to show up on the properties page of both wtparts and CAD Documents in Windchill, you will have to map Pro/ENGINEER parameters to Windchill Instance Based Attributes (IBAs). Windchill supports only the following type or Pro/ENGINEER parameters.

- Integer
- Real
- String
- Yes/No

All the parameters you want to map must be **designated** in Pro/ENGINEER.



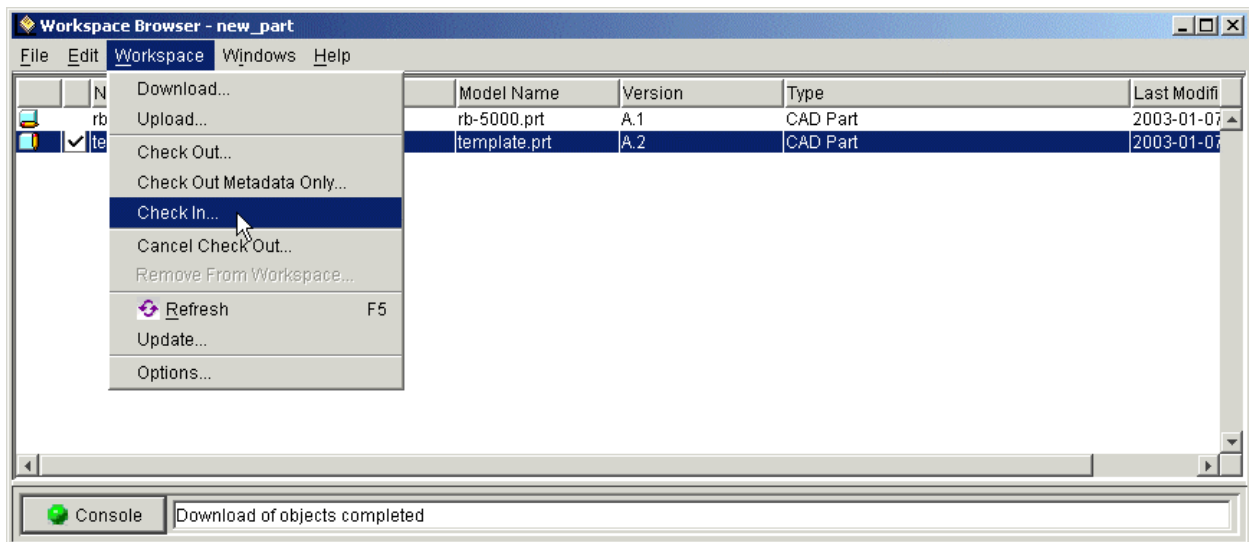
**Figure 4 - Pro/ENGINEER Parameter Information Window**



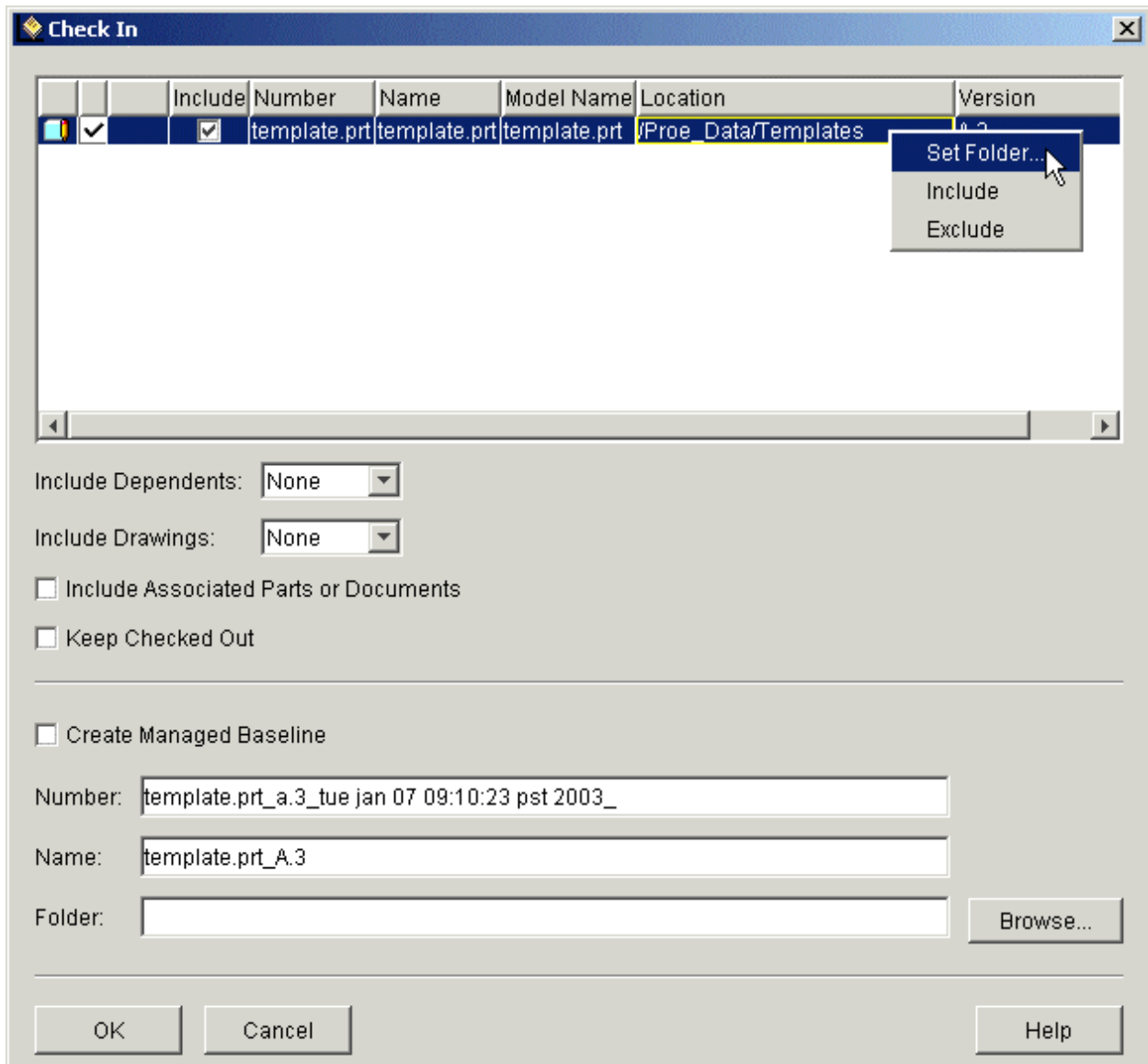
**Figure 5 - Pro/ENGINEER Template Part**

Once you have completed the construction of your start part, you need to upload it to the Workspace Browser. You complete this by selecting on the Windchill command in the Pro/ENGINEER Toolbar. Select **Workspace > Upload** to create the template CAD documents and **Check-In** the new CAD documents. Verify that the working folder (as specified in the working folder parameter) has write permission to ensure download or upload functionality. The **Workspace Browser** creates a subdirectory in the working folder with the same name as the workspace created by the user. This directory must have default write permission. If you change the directory permission, the **Upload** and **Download** or **Check In** and **Check Out** commands may not function correctly. Users who check out and download objects into the `epm.new.object.folder` must have access privileges. The Windchill folder that corresponds to the mapping for `epm.new.object.folder=<folder_path>` is defined in the `profileadapter.ini` file.

After you upload the template part, you will see the part in your Workspace Browser. The next step you must perform is to check the `template.prt` into Windchill.



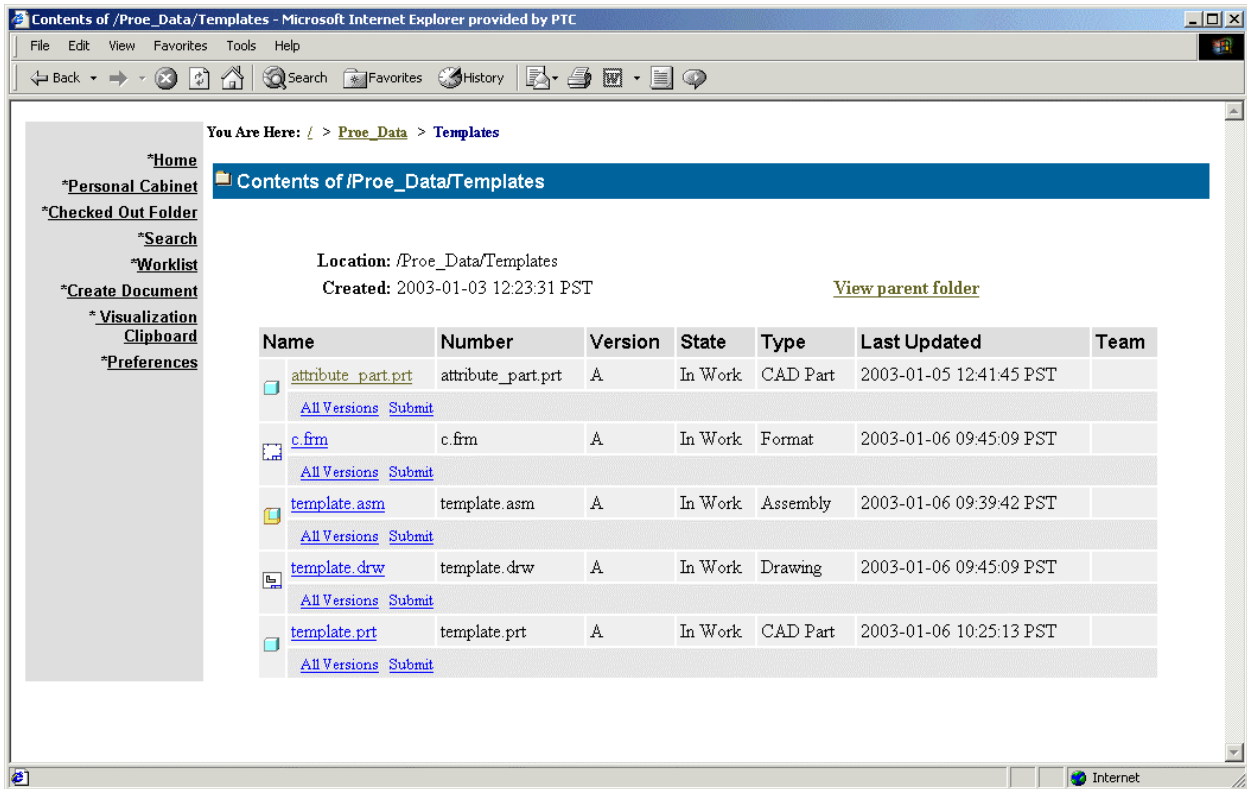
**Figure 6 - Check In of Template Part**



**Figure 7 - Check In Filters**

Repeat this procedure for your start assemblies and start drawing template.





**Figure 8 - Windchill Templates Folder**

Once all desired templates have been checked into Windchill, now you must map parameters to IBAs. The Workgroup Manager for Pro/ENGINEER supports Pro/ENGINEER data types for mapping parameters to Windchill IBAs, as shown in the following table. Pro/ENGINEER parameters must be **designated** to transfer their values to mapped Windchill IBAs.

| Pro/ENGINEER Parameters | Windchill Instance-Based Attributes (IBA) |
|-------------------------|---|
| Integer                 | Integer number                            |
| Real number             | Floating point number                     |
| String                  | String                                    |
| Yes/No                  | Boolean                                   |

The attribute mapping can be done in the two following ways:

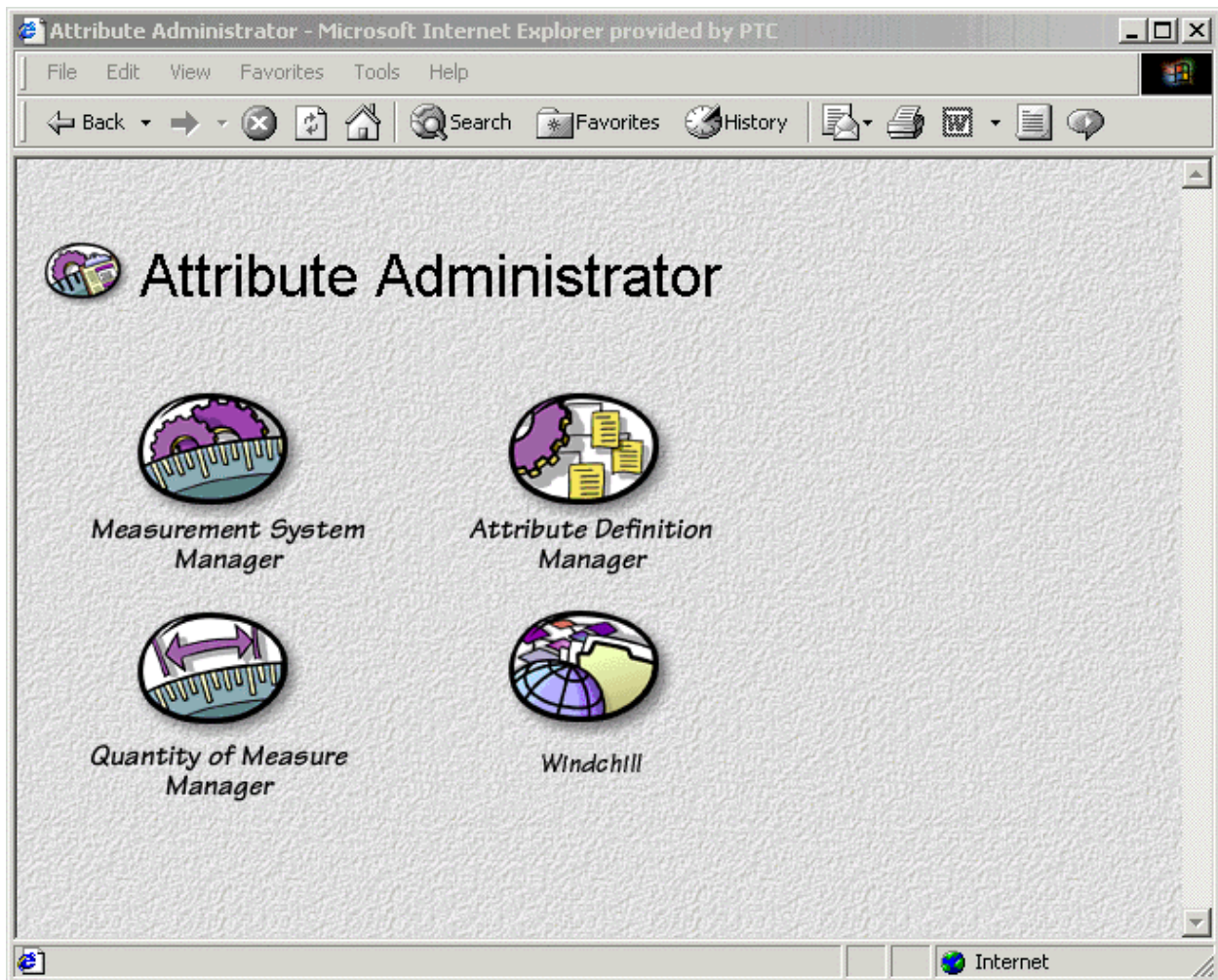
- Explicit parameter-to-attribute mapping

Explicitly map Pro/ENGINEER designated parameters with Windchill IBAs, the iba.properties file where the mapping is defined, must be created and modified.

- Implicit parameter-to-attribute mapping

Implicit parameter-to-attribute mapping occurs when there is an instance-based attribute in Windchill, with a name (all uppercase) identical to the name of a designated parameter in a Pro/ENGINEER model file. When the Pro/ENGINEER model file is uploaded into Windchill as content of a CAD document, the values of the Pro/ENGINEER parameter are transferred to the Windchill IBA.

To create IBAs, login to Windchill on the Home page, select the SiteMap link at the bottom of the page. Then select on Attribute Administration link. Then select Attribute Definition Manger to create Instance Based Attributes. (If you do not have access to these links, check with your Windchill Administrator.)



**Figure 9 - Attribute Administrator Applet**

Once all the attributes are defined, and then you need to map them in the iba.properties file. The iba.properties file is located on the Windchill Server machine at \$WTHOMES\codebase\com\ptc\prowt\proesvcs\util\iba.properties. Open the file a text browser and add Pro/ENGINEER name to be equal to the Windchill IBA as shown in the figure below. Once you have made changed this file, you will need to recreate the wmpref.jar.

Remake the jar file with the following command: Windchill wt.tools.Makejar jar=wmpref.jar

```
TextPad - [D:\ptc\wc626\Windchill\codebase\com\ptc\provw\proesvcs\util\iba.properties]
File Edit Search View Tools Macros Configure Window Help

iba.properties
wvs.properties

This file contains the attribute mappings for Pro/E designated parameters
and Windchill IBA's.

The key to the left of the '=' is the name of a Pro/E designated parameter. To the right
of the '=' is the name of the corresponding IBA definition in Windchill.
NOTE that the mapping is case sensitive. Pro/E parameter names must always be
in upper case.

On upload, the service will lookup a mapping for any designated object or
member attribute and, if the corresponding IBA definition exists, it will
assign the attribute value to the object being uploaded. If mapping for a
particular attribute does not exist, the adapter will try to use the upper case Pro/E
attribute name. If the definition cannot be found, a conflict will be created.

On download, the IBA name is mapped to the Pro/E Designated parameter name.

NOTE:
1. Do NOT mention the Attribute organizer in the IBA definition for Windchill
2. Always put the Pro/E designated parameter in "UPPER CASE"
3. Use "|" as the separator for hierarchical attributes
4. In an IBA definition of type Size|Length, Length will be treated as a sub attribute of S

The following are examples of invalid mappings:

length=Length
length=/Length
LENGTH=/Length
LENGTH=|Length

The following are examples of valid mappings:

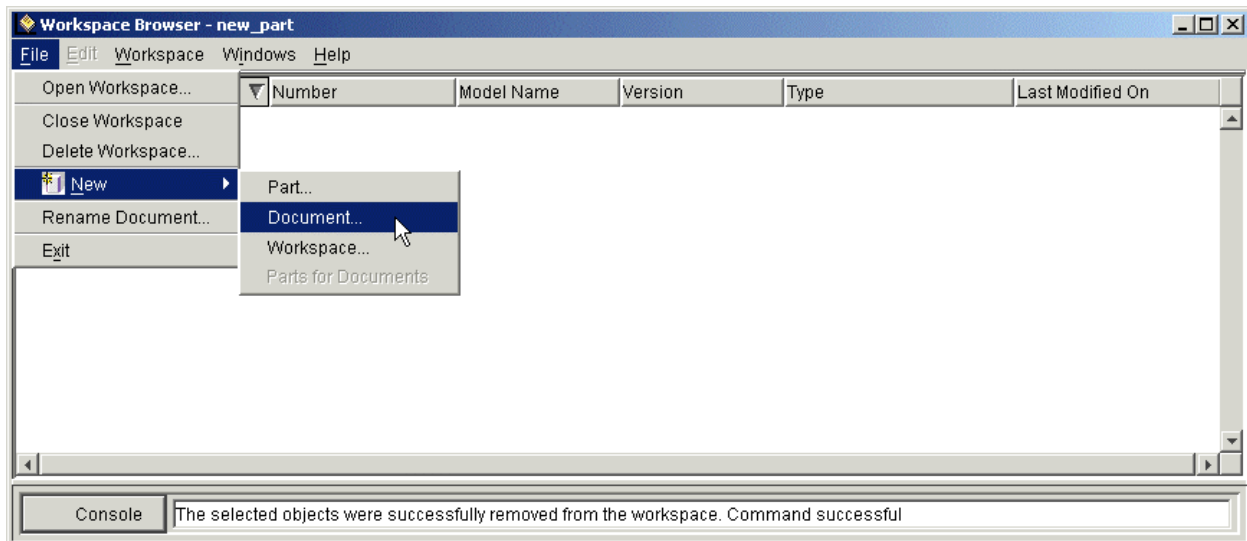
LENGTH=Length
DESIGNER=Author
CREATOR=Author
PROE_AUTHOR=AUTHOR

The following is an example of a valid mapping if the intent is to map to a sub attribute
Here "Size" is an attribute under an organizer. "Length" is a sub attribute of "Size"
LENGTH=Size|Length
MATERIAL=MATERIAL
VENDOR=VENDOR

Change the following for site requirements
PROI_VERSION=VERSION
PROI_BRANCH=BRANCH
PROI_MODIFIED=MODIFIED
```

**Figure 10 - iba.properties File**

If you have not exited the Workspace Browser, do so now and restart the Workgroup Manger for Pro/ENGINEER. When you start the Workgroup Manager, you will automatically be prompted to download the new wmpref.jar file before the Workspace Browser appears. After downloading the new wmpref.jar file, open a Workspace Browser and select File, New, Document.



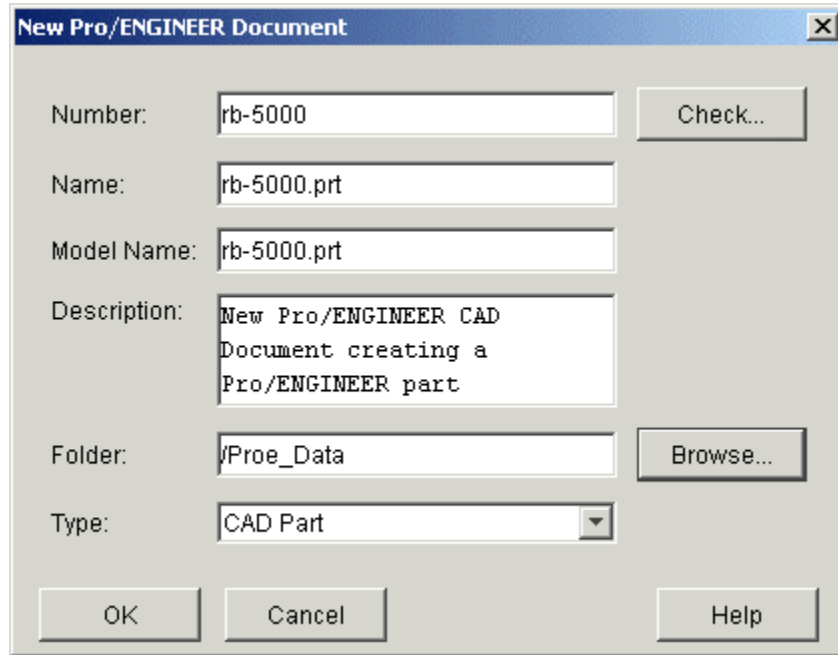
**Figure 11 - Create New CAD Document Menu**

You will be prompted to fill out the New Pro/ENGINEER Document dialog box to create the CAD Document. The new document is created with the document number, document name, Model Name, a description, document folder, and document type that you specified and checked out to your active workspace.

The following conditions apply:

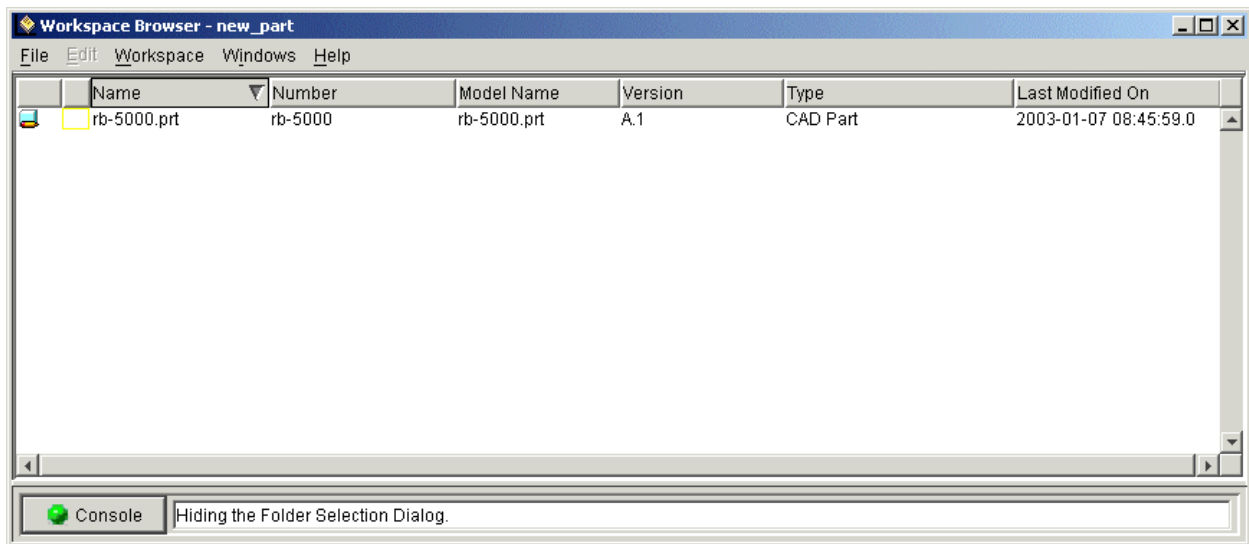
- Required fields are: a unique document number or a unique Model Name and the Windchill location in which to store the new CAD document.
- Enter a unique document number. Click **Check** to verify its uniqueness and length. CAD Document numbers are restricted by Windchill to 200 characters.
- You must specify a valid document type extension, such as .prt, .asm or .drw.
- If a template CAD document is specified in the newdocument.ini file, the newly created CAD document has a Pro/ENGINEER model file associated with it. Otherwise, the document is created without content.
- The **Name** field is not required. If not specified, the document name is the same as the document number.
- Enter a unique **Model Name** for the new CAD document.

**Note:** The Model Name will be validated against Pro/ENGINEER naming conventions such as extension and length. Pro/ENGINEER and Workgroup Manager for Pro/ENGINEER will not allow Model Names to be more than 31 characters, excluding extension. Model Names must be unique in both Windchill and the current workspace.

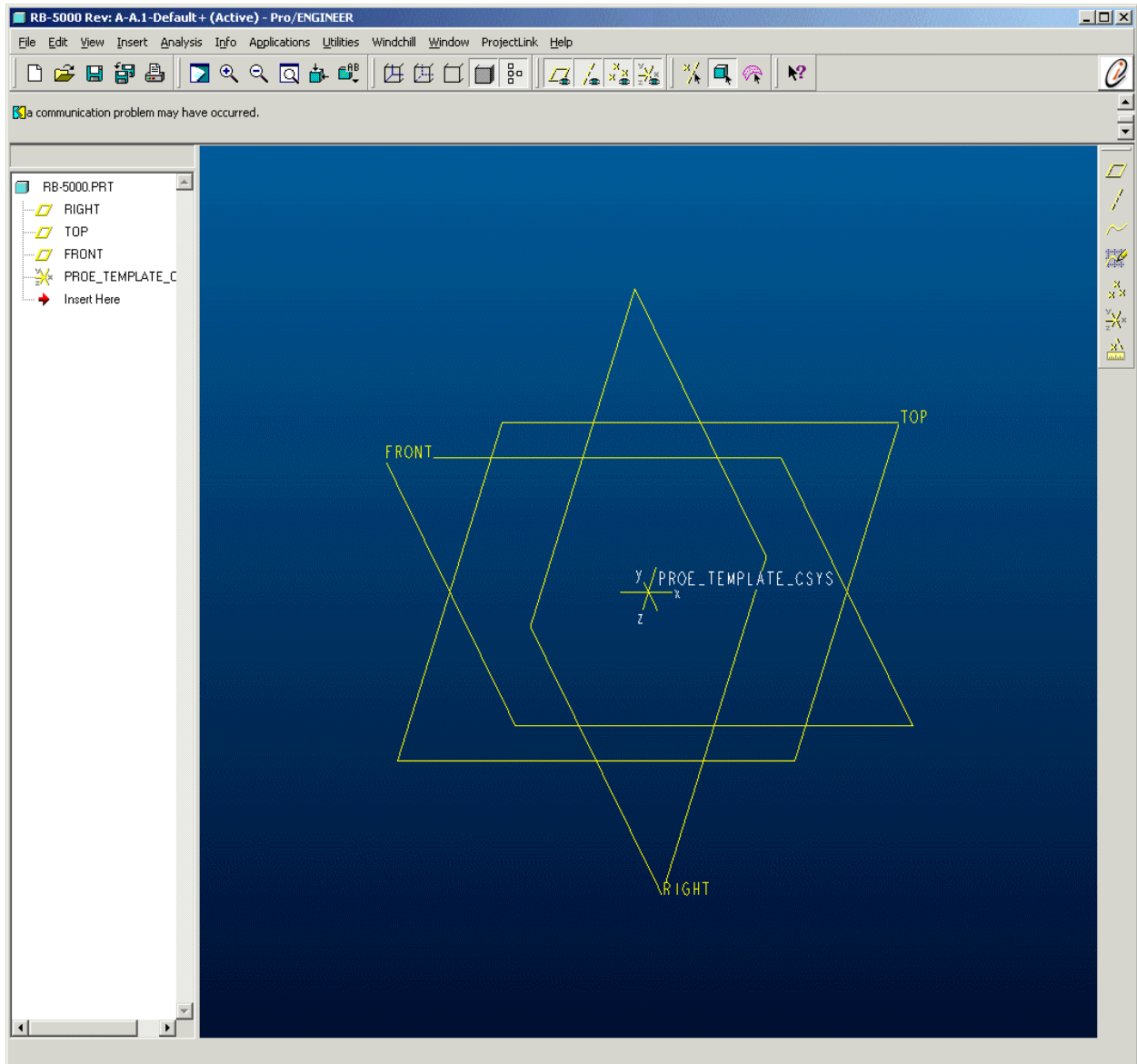


**Figure 12 - New Pro/ENGINEER Document**

The part now appears in your Workspace Browser as a new CAD Document. You can open this document in Pro/ENGINEER and begin designing your part. The new CAD Document contains all the information from the template you just created and configured to work with the Workgroup Manager.



**Figure 13 - Workspace Browser with New CAD Document**



**Figure 14 - New RB-5000.prt in Pro/ENGINEER**

[Back To Top](#)

## **Announcements**

### Customer PTC E-Newsletter Survey

For the past 2 years I've been sending you our Customer PTC E-Newsletter such as this one.

In an attempt to deliver the highest quality and create the most value we would like to use this survey to further our "Learning" Relationship with you, our Customers. Even if you are a first

time recipient of the newsletter, please feel free to share your comments on this one you're reading.

It will not take longer than a few minutes to fill out this quick survey as your input is invaluable and we greatly appreciate your input. Please click on the following link to begin:

[To Enter Survey, Please Click](#)

Please note it may be impossible to "Customize" the newsletter to meet every individual request however we will attempt to integrate as many ideas as possible.

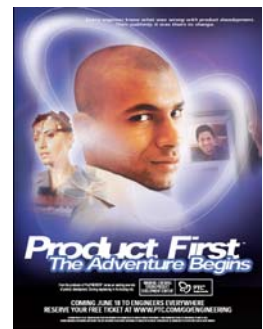
Thank-you very much for your time and feedback as we try to improve this value-added service.

## 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/>

[Back To Top](#)



## Upcoming Events & Training Class Schedules

Upcoming, 2003      Your local Pro/Engineer User Groups  
<http://www.prouser.org/rugs/>

Upcoming, 2003      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.

[Back To Top](#)