

# PTC® Windchill® Tips and Tricks

PTC Windchill is an integral component of PTC's Product Development System, managing all product content and business processes throughout the product and service lifecycle. PTC Windchill offers a central repository for all product and process information that reduces rework and improves time-to-market by effectively capturing, configuring, and managing all product information during every step of the product lifecycle.

By analyzing performance in the earliest phases of product development (where decisions have the most impact and are the least disruptive), it enables customers to bring better performing, more profitable products to market faster. Customers can use PTC Windchill to track and manage quality throughout the product development lifecycle—ensuring that product quality and reliability are aligned with the requirements set for them over the course of a product's development and use. Finally, PTC Windchill helps users streamline key product development and service information processes, which ultimately will help them achieve their company's business objectives.

PTC Windchill works closely with PTC Creo® for CAD Data Management, with the former serving as a PLM tool and the latter providing authoring capabilities. Customers come to PTC Windchill and PTC Creo with varying degrees of experience. Some long-time users think they know the products inside and out, and others are completely new to the software. Regardless of your level of experience, PTC Windchill solutions offer numerous ways to perform any given task in conjunction with PTC Creo—some of which are obvious, some of which are not.

The following are tips and tricks most users don't know how to implement. By having an arsenal of several ways of performing common tasks, you can select the method that works best for you.

## Tip 1—Externalizing Simplified Representations

If you're working on an assembly for, say, a tractor with thousands of parts and gigs of data, you may want to work with a subset of that assembly (e.g., the body or the power train) to remove complexity and improve system performance. PTC Creo offers a capability called Simplified Representations that allows you to look at a subset of an assembly. By default, however, PTC Creo saves the Simplified Representation as part

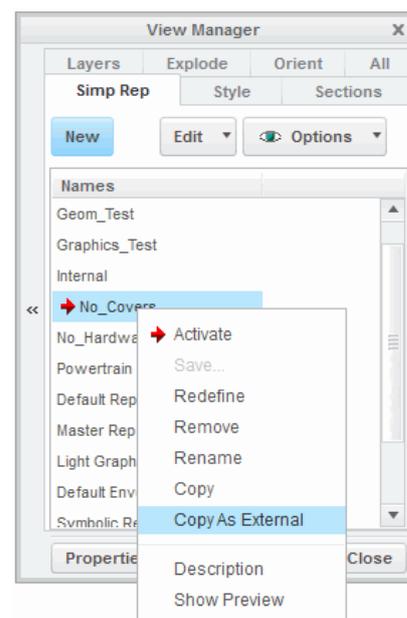


Figure 1: Use the View Manager to create an External Simplified Representation.

of the assembly file. This means it's hidden within the assembly and PTC Windchill doesn't know anything about it. As a result, when you want to find a Simplified Representation again, you can't simply do a search to find a list of all simplified representations and pick the one you want. You have to browse through the File Open menu to find what you're looking for, and then open it in a specialized manner.

To make Simplified Representations easier to find, consider externalizing them. Externalizing a Simplified Representation makes it a searchable PTC Windchill object.

To create an External Simplified Representation, go to the View Manager, right-click an existing active Simplified Representation and select 'Copy As External,' as shown in Figure 1.

PTC Creo will prompt you to enter a name. Provide a name that you'll be able to distinguish when searching for the Simplified Representation in PTC Windchill (e.g., "drill\_400\_no\_covers" as shown in Figure 2). The saved External Representation will appear in your workspace browser with its own specific Object Type icon.

With an External Simplified Representation, you can create a Contributing Content link between it and the assembly's owner-linked PTC Windchill part. This allows the Simplified Representation to provide additional information about the PTC Windchill part. After uploading the External Simplified Representation, you can use the Edit > Auto Associate workspace feature to generate this link. You can then check the External Simplified Representation into PTC Windchill. To make the Contributing Content association, you go to the Auto Associate Menu, select the External Simplified Representation you're looking for, and choose "Contributing Content" as the Association Type. You can even create a folder in PTC Windchill specifically for External Simplified Representations.

Figure 4 shows all of the Contributing Content for the Drill 400 part.

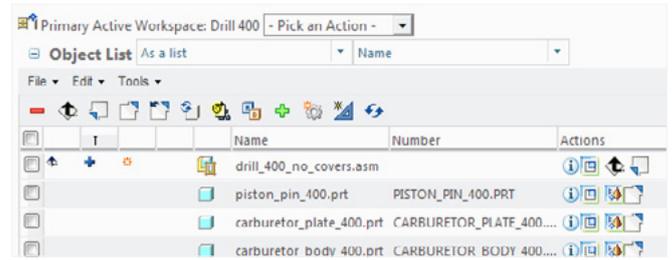


Figure 2: The saved External Representation appears in your workspace browser.

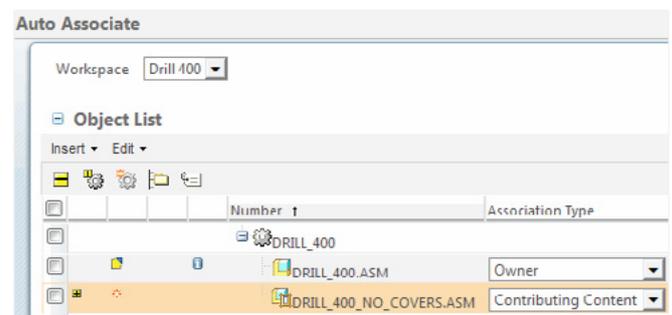


Figure 3: A Contributing Content association allows you to link the External Simplified Representation to a PTC Windchill part to provide additional information.

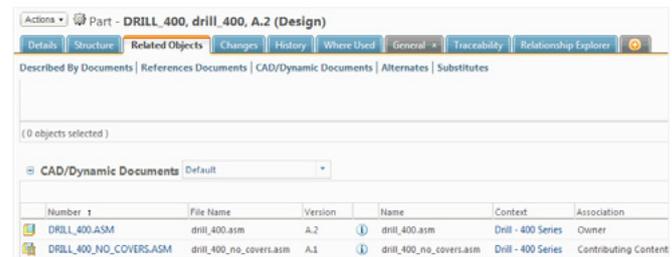


Figure 4: A list of Contributing Content for a particular part

When you want to find a particular External Simplified Representation, you can directly search/browse for and open it from PTC Windchill. The system will only download components within the External Simplified Representation to your workspace, as shown in Figure 5.

By creating External Simplified Representations, not only can you view a subset of a large assembly for ease of use and better performance, but you can also find the Simplified Representations you're looking for easier and faster.

## Tip 2—Auto Associating CAD files

When checking a CAD file into PTC Windchill, you can choose to associate it with a product structure — which can be either a new or an existing product structure. You have two options for associating the CAD file with an existing product structure:

- The Auto Associate check-in option
- The Edit > Auto Associate workspace feature

Most people use the Auto Associate check-in option (which is accessed through the check-in wizard) because it's quick. When you want to check-in your CAD document, you simply select the Auto Associate option. PTC Windchill will then look for an existing PTC Windchill part of the same name. If the part is found, PTC Windchill associates it with your CAD document. If the part is not found, a new part will first be created, and then associated with your CAD document. The trouble is that you're unable to preview the PTC Windchill part that PTC Windchill associates the CAD file with. To make sure that PTC Windchill has associated the CAD file with the correct part, you have to query the part and look at it. In addition, this option only gives you an "Owner" link.

The Edit > Auto Associate workspace feature (which you select from the Workspace Browser) has two advantages over the Auto Associate feature. It provides:

- A preview of associations between PTC Windchill parts and CAD documents so you can review the proposed associations on a case-by-case basis prior to check-in.
- More direct control over associations and CAD-driven PTC Windchill part attributes while maintaining default settings for ease of use. You can choose from any of the five possible standard link association types: Owner, Content, Image, Contributing Content, and Contributing Image, as shown in Figure 6. You can also manage any exceptions to the default associations.

Once you have verified that you have the desired associations, you can check-in.

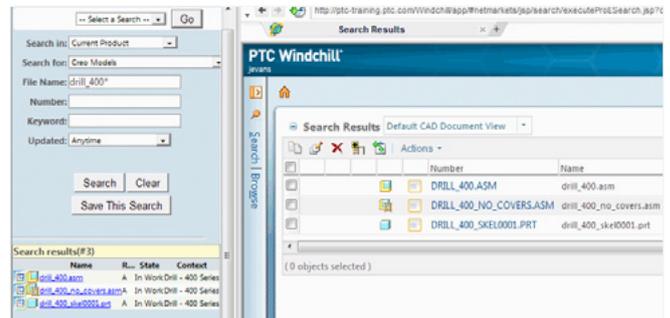


Figure 5: Search for Simplified Representations from the search window.

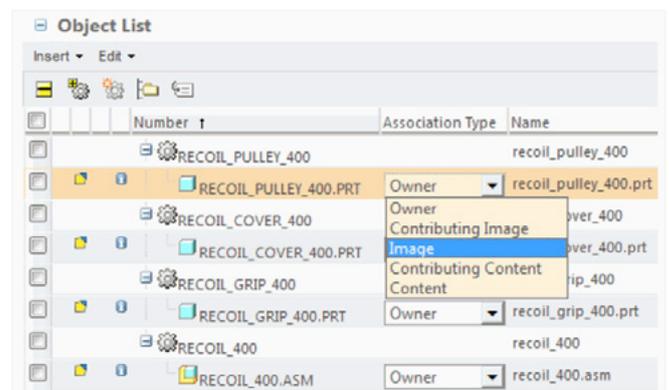


Figure 6: The Edit>Auto Associate workspace feature lets you choose from five link association types: Owner, Contributing Image, Image, Contributing Content and Content

By using the Edit > Auto Associate workspace feature, you save yourself the trouble of having to iteratively change the association between your CAD file and the PTC Windchill part if the program doesn't make the auto association you hoped it would.

## Tip 3—Managing access to local cache folders

Some customers need to set up multiple projects each with their own corresponding PTC Windchill account. As a designer, you may not want to spend the time logging into both PTC Windchill and the OS each time you switch to a different account (e.g., to work on a different project). You can run into problems, however, when you use PTC Creo Parametric™ with PTC Windchill and attempt to login to PTC Windchill using different accounts with the same OS login.

The reason there may be problems? PTC Creo downloads your CAD data to a local cache. PTC has always required each PTC Windchill login to have its own local disk folder for the local workspace cache. If you're logging into PTC Windchill using different PTC Windchill accounts with the same OS login, you may be violating this PTC policy. In this case, attempting to login to PTC Windchill using an account that is different than the one last used to access the local cache, starting with PTC Creo 2.0 M080, will present you with the warning shown in Figure 7.

Note that if you use a separate OS login for each separate PTC Windchill login being used, you won't run into any problems because the local cache is established by default in the profile folder of the OS login.

If you do encounter this warning, you may be able to continue by clicking Ignore. However this is NOT recommended. If you proceed, there's a good chance you'll encounter more error messages.

To avoid this issue, you can create PTC Creo Parametric Startup Files (PSF) that will configure PTC Creo Parametric at startup to use different local drive locations for the local cache and settings files.

By default, PTC Creo generates a single PSF file during installation. This, and any additional PSF files, are located in the Creo Parametric <loadpoint>\Parametric\bin folder. If multiple PSF files exist, you're prompted to select a startup configuration when you start PTC Creo Parametric, as shown in Figure 8. One startup configuration entry is generated for each PSF file.

To create a new PSF file, copy an existing PSF to a new name (requires the .psf extension), then edit the PSF file. To name the new PSF file, change the text string entered for the value of DESC, as shown in Figure 9.

To specify a unique local cache area on a local disk in the PSF file, define the environment variable, PTC\_WF\_ROOT, to point to the desired local cache folder as shown in the highlighted line in Figure 9.

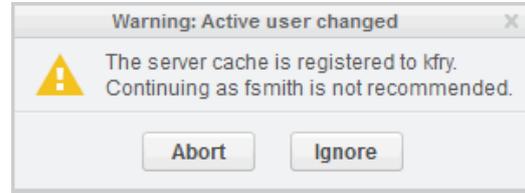


Figure 7: When you attempt to login to PTC Windchill using an account different than the last one used to access the local cache, PTC Creo presents this warning

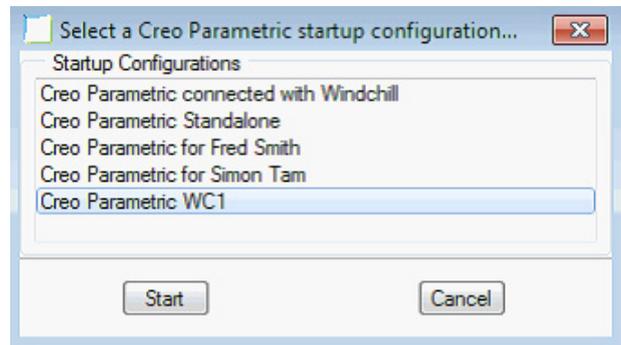


Figure 8: If multiple PSF files exist, PTC Creo prompts you to select a startup configuration

```
// PTC - PSF file: parametric
//
PRODUCT=Creo Parametric
DESC=Creo Parametric for Fred Smith
SHIPCODE=M080
CREO_COMMON_FILES=Common Files\M080
//
.
:
.

// USER - PSF
// Add User specific environment or run applications below here
ENV=DBG NCPOSI PAIH=%CREOAPP DIRECTORY%\bin
ENV=PTC_WF_ROOT=D:\ptc\PS_Creo_Server_Cache
RUN="C:\ptc\tools\restore-server-file.bat"
```

Figure 9: Editing a PSF file

In addition to editing the DESC line, you should create all other entries below the comment at the bottom of the file that instructs you to ‘//Add User specific environment or run applications below here.’ In general, the syntax is as follows:

- To specify the value of an environmental variable, use the ENV command
- Use the equal sign (ENV=) to change the existing value of the variable
- Use the plus sign and equal sign (ENV+=) to add a value after the existing value
- Use the minus sign and equal sign (ENV-=) to add a value before the existing value

By following this procedure, you’ll stay out of trouble and avoid any strange error messages.

## Conclusion

Some of these tips are not easily found in the basic handbook, so they are not widely known. They are representative of the types of hidden tips and tricks that interacting with instructors during PTC University training sessions will bring to light.

PTC University’s training and tools help individuals and organizations use PTC software more effectively. Whether you’re looking for specific training courses, ways to improve the proficiency of experienced users, or a personalized corporate learning program that meets your training schedule and budget requirements, PTC University can help you optimize your business through traditional classroom training, online courses, as well as leadership training and consulting.

PTC University enables you to make the most of your software investment through:

### PTC Product Knowledge—First Hand

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### Global Expert Network

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### Flexible Delivery Options

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To learn more about how our training can help your organization, please [contact us today](#). Our Training Advisors will be happy to help you select the right training option for you. Please also take a look at the [PTC University Website](#).

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