Essential Windchill CAD Tasks

Windchill is a Product Lifecycle Management (PLM) platform that provides numerous functions in a product development environment including:

- Computer Aided Design (CAD) data management.
- Bill of Materials (BOM) Management.
- Change Management.
- Configuration Management.
- Manufacturing Process Planning.
- Supplier Management.
- Access to viewables across the enterprise without needing the native authoring software installed locally.

Windchill can be accessed from a standalone web browser or from the embedded browser within Creo. To log into Windchill you will need your Windchill username and password.

The following list contains the 10 essential tasks that a person needs to know when performing CAD data management tasks in Windchill with Creo Parametric. The actions can be initiated from the Creo Parametric interface or embedded browser as indicated.

1. Register Server (usually a one-time task). This connects Creo to Windchill and can only be performed from within Creo.

   File > Manage Session > Server management > Server > Register new server

2. Create New Workspace. Workspaces are your own personal local areas to view and modify CAD documents.

   a. Creo: File > Manage Session > Server management > Workspace > New

   b. Browser: Navigator (narrow left strip): Browse > Product icon > expand desired context > Workspaces > New icon

Creo Windchill – https://www.creowindchill.com

3. Browse. This is used to locate objects when you are unsure of the necessary attributes to allow searching, or when you simply want to investigate what objects are available.

    a. Creo: Navigator (left panel) Common Folders > Windchill Cabinets

    b. Browser: Browse button in Navigator, and select Products or Libraries

4. Search. This is the most effective method for locating objects in the Commonspace (Windchill vault).

    a. Creo: Search tab in Navigator

    b. Browser: Search bar at top of each page

5. Add to Workspace (when you want to open, view, measure, print, etc.).

    a. Creo: File > Save (for new objects) or File > Open (for objects already in vault)

    b. Browser: RMB on object and

6. Check Out (when you want to make changes to an object and lock it to prevent others from making changes to it).

    a. Creo: File > Check Out

    b. Browser: Check the object's box and Check Out Icon

7. Check In (when you want to share changes with everyone by adding modified objects back to the Commonspace).

    a. Creo: File > Check In

    b. Browser: From the Workspace page, check the object's box and Check In icon

8. Update Workspace (get the latest changes from other users to the CAD documents in your Workspace). This can only be performed from the embedded workspace in Creo, and should be done at the beginning of every session.

    From the Workspace in the embedded browser: Update icon

9. Synchronize Workspace (get the latest changes to the object's metadata, such as name, folder location, etc.). This should be done at the beginning of every session.

    From the Workspace in the embedded browser: Tools > Synchronize

10. Delete Workspace. For proper housekeeping, Workspaces should be created at the beginning of a new task and deleted when the task is complete. Old Workspaces with many (several thousand) objects can become unstable and lead to a corrupt cache.

    a. Creo: Tools > Server Manager > Workspace > Delete

    b. Browser: My Workspaces page > Delete icon ✕

For questions and comments, please contact Dave Martin at dmartin@creowindchill.com.
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