

PRODUCT DESIGN & DEVELOPMENT LAB

Specialization Program
PLM Windchill

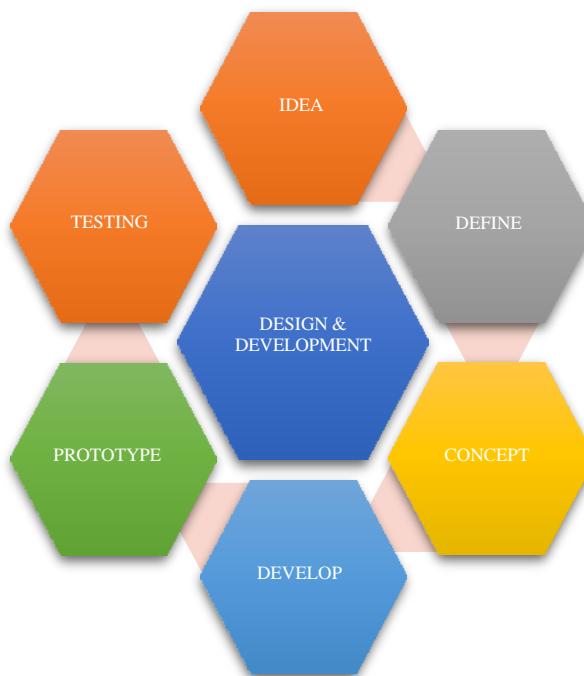
STUDENT MANUAL

• DesignTech
Technology for designing the future



PRODUCT DESIGN AND DEVELOPMENT LAB

- In this lab we use Creo for imagine, design, create, and innovate your products better with Creo, 3D CAD/PLM software, and solutions for product design and development.
- This lab allows you to visualize your imagination in a virtual environment and which can be further developed and optimized. PTC Creo software package is used to achieve this and which helps you to save time, money and effort to develop new concepts and bring them into reality. In any design cycle, product design and development is the critical stage to bring life to any conceptual model.
 - In this lab you will be learning basic modeling of parts (solids, sheet metals and Class – A surfaces), Assembly, Drafting, creating of different tools and dies for manufacturing special parts and also MathCAD to create report of your mathematical calculations related to your design.





- **The list of courses offered,**

S. No	Name of the Course	Duration
1	Creo for Design	108 Hours
2	Creo for Industrial Engineers	72 Hours
3	Specialization program for Tool and Die design	40 Hours
4	MathCAD	40 Hours

WINDCHILL

- Manufacturers have never needed real-time information sharing, dynamic data visualization, and the ability to collaborate more than they do today. With easy, secure data access for multi-disciplinary and geographically-distributed teams, quality-focused processes, and a data driven approach to manufacturing, Windchill is elevating how product development gets done.
- Windchill's open architecture enables easy integration with other enterprise systems, including IoT, providing a solid foundation for a product-driven digital thread. PTC's PLM system provides comprehensive out-of-the-box functionality and highly configurable role and task-based apps. Expand self-service access of traceable product data to non-experts who don't typically use PLM, while avoiding over-customization and complexity.



➤ Courses offered:

S. No	Name of the Course	Duration
1	Specialization Program PLM Windchill	50 Hours



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Day 1 Introduction to the Windchill Environment

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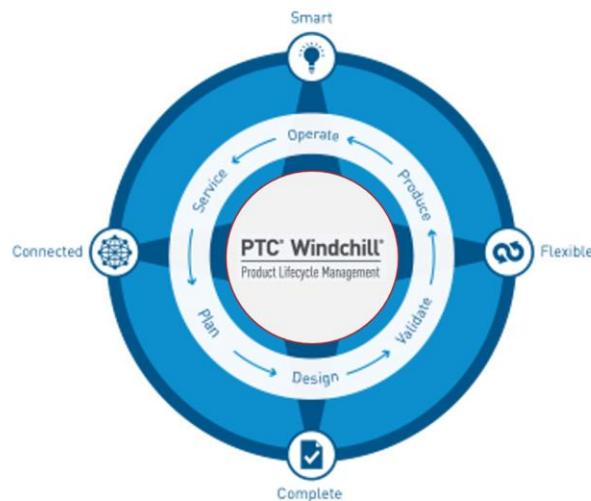
Introduction to Windchill Environment

- Product lifecycle management software allows geographically separated, multi-disciplinary units to strategically collaborate with partners and customers using advanced, updated product data.
- Windchill PLM is the base for the digital thread, producing supply chain coordination and business continuation. Data governance and traceability given by product lifecycle management empowers organizations to drive down expenses, quicker time to market, and achieve the most significant levels of quality and compliance.
- Develop a base for a wide variety of new revenue and cost-saving opportunities with PTC Windchill out-of-the-box applications. Seamlessly integrate our PDM & PLM system with ERP, MES, CAD, AR, IoT technologies, and more.



Introduction of Windchill

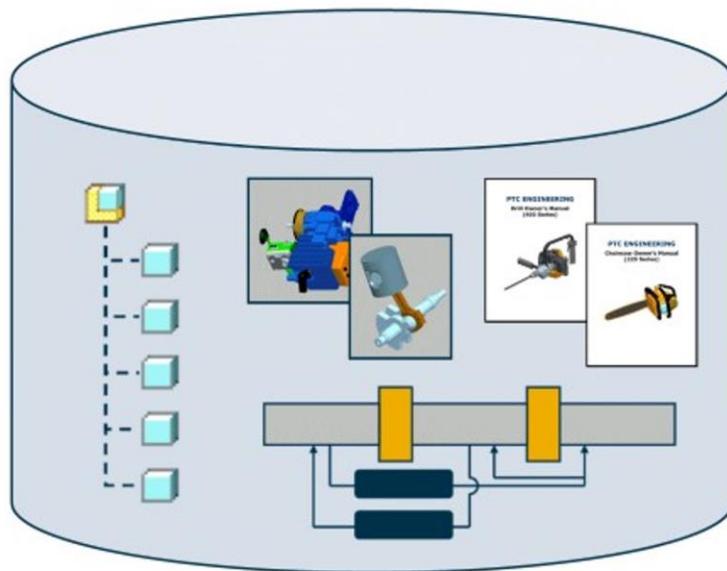




- PTC Windchill is a product lifecycle management (PLM) application suite that leverages consolidated view of product information through multi-system data.
- It's a systemic enterprise wide approach to maintaining product and process quality throughout the entire product lifecycle.
- PTC Windchill PLM software provides a complete functionality dimension to help organizations capture product structures from computer-aided design tools, transform them into full engineering bill of materials, to manufacturing bill of materials, to service bill of materials all while retaining the linkages between different perspectives on the product. Windchill solutions break down organizational barriers, allowing teams to work faster and more accurately all while reducing time-to-market and cutting costs.
- The Windchill Product Lifecycle Management system can help streamline your product development and service information processes. It's a system that can help your organization create better products, with more variations, faster, at a lower cost.



- As a manufacturer, your pressures are not only continual – they're mounting. Industry is changing, customers' requirements are evolving. And supply chain is constantly adjusting to compete in today's environment. In the middle of all this, more is being demanded of you and your entire organization.
- Windchill helps with exactly that. As an integral component of PTC's Product Development System, Windchill manages all product content and business processes throughout the product and service lifecycle. And it has a robust, high-performing architecture to help you today – and to prepare you for tomorrow's uncertainties.
- Windchill provides a rich systematic approach for creating, configuring, managing and reusing product structures and associated content, such as CAD files, documentation, requirements, manufacturing information, service information, part/supplier data, calculations and illustrations.
- Windchill can also play an important role in increasing your company's competitiveness by allowing continuous improvements and automation of business processes and procedures.



- Windchill is a collaborative environment, enabled with today's latest Web technology, and has been designed to enable you to develop and manage data more effectively and efficiently.
- It provides your entire enterprise with a common information-sharing mechanism that stores and manages information, manages information evolution, controls access to information, and provides collaboration tools.

Introduction of Windchill Environment

- Windchill Is collaborative data management environment
- It stores Information
- Manages information in storage locations
- Manages information evolution
- Controls access to information
- Provides Collaboration tools
- Company's intellectual capital is often complex, multi-faceted, and can contain hundreds of relationships. Windchill is designed to manage this information and all of its required relationships.
- Consequently, to understand any specific feature of Windchill, you must also have an understanding of what its role is in the larger picture.



- It is useful for following:
- Store Information
- Manage Information
- Control Information
- In any company, the product development process generates a tremendous amount of intellectual property.
- To reduce time-to-market, a team must share the data and collaborate on product design.
- Without the ability to control product information, you find multiple developers using different variations of the same content, which can result in overlapping or inconsistent designs.
- Windchill PDMLink solves this problem by storing master data in a secure area where you can assure its integrity and monitor, control, and record all changes.
- While securely stored, you can distribute the master data freely to users in various departments for input, review, or reuse.
- When a change is made to the data, Windchill PDMLink stores a modified copy of the data, signed and dated, in a secure area alongside the old data, which remains in its original form as a permanent record. In addition to providing change control
- management, Windchill PDMLink enables you to manage a product's release cycle as well as its configuration.



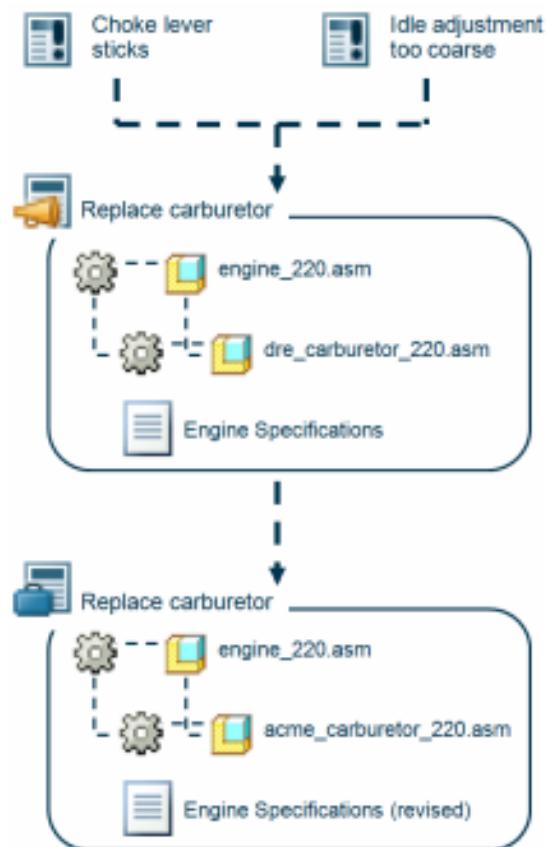
- Data Management objects store and manage information and content
 - Parts and End items
 - Cad Documents
 - Dynamic Documents
 - Documents
 - Link
- 
- One of the main purposes of Windchill is to store and manage information and content. To do this, Windchill uses what it calls a Business Object. Because your data is often complex, you may need to store a number of different types of information.
 - Windchill has several types of objects to handle these different types of information and enables you to build relationships between them
 - Parts and end items are used to build product structures. They do not store data or design files, but reference other objects that do. They can also reference each other in a hierarchy called a product structure.
 - CAD document objects store files developed in a CAD authoring tool. Windchill displays the appropriate icon based on the file type stored as primary content.
 - Dynamic Document objects store files authored in Arbortext Editor. If the primary content is an SGML file or fragment or an XML file or fragment, Windchill displays the leaf icon. If the primary content is a graphic file used in an SGML or XML document, Windchill displays the graphic icon.



- Document objects store and manage content. While document objects typically store and manage documentation files, there is no limitation to the types of electronic files a document object can store as content.
- Links store URLs to Web pages both inside and outside of the Windchill system.

Change Management objects

- Change management objects store and manage information related to change management
- Problem Report(PR):
 - Open to users
 - Captures potential problems and enhancements
- Change Request(CR):
 - Initiates the change process
 - Collects the affected objects
 - Drives the decision to pursue change
- Change Notice(CN):
 - Executes the changes
 - Manages new objects and new objects versions



- Change Management Objects are used in Windchill's change processes. Their purpose is to gather information about a product issue or suggested enhancement, collect the affected product objects, such as parts and documents, and route the information through a series of business processes where

the issues are evaluated, decisions are made, and the product changes promulgated. Each change object has a specific purpose:

- The problem report (PR) can be completed by anyone who would like to suggest a change or report a problem with a product. Typically, problem reports are used to document problems submitted in testing, in the field, or by customers.

A problem report:

- Is open to most users.
- Captures potential problems and enhancements.

A change request (CR) can then be used to gather common problem reports and submit a formal request for a change. Alternatively, a change request can be initiated without any problem reports. A change request:

Initiates the change process.

- Collects the affected objects.
- Drives the decision to pursue change.

Finally, a change notice (CN) can be used to assign tasks that complete the implementation plan and implement the changes.

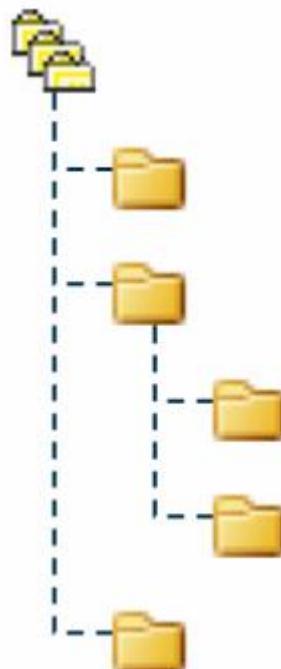
A change notice:

- Executes the change.
- Manages new objects and new object versions.

All of the change objects can be associated to one another, and to the Windchill parts and documents that they are affecting.



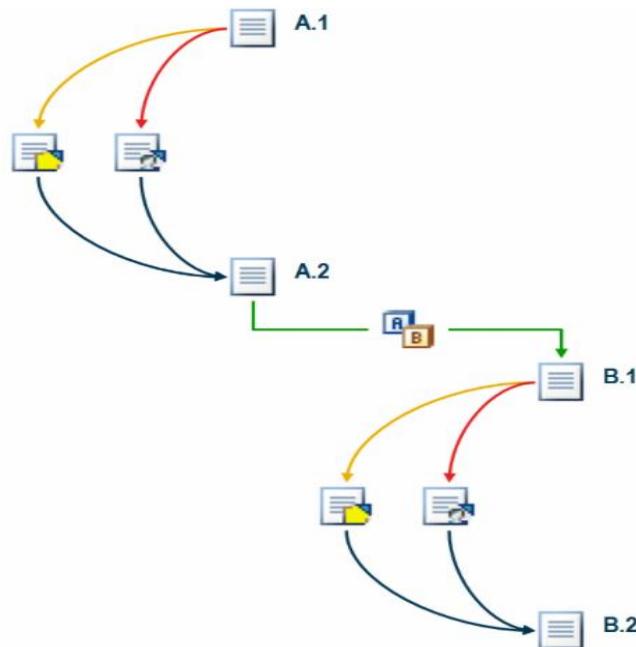
- Objects are stored in context.
- Products store objects related to one product or deliverable
- Libraries store objects related to one other, but not to one specific product



- When you create objects, you also decide where to store them. Typically, this is based on the context in which the object is used.
- Products represent storage for information related to a specific product or deliverable.
- Libraries are more general. A library is the logical storage location for objects that are related to one another but not to one specific product. Typically, libraries store information that must be shared across the organization.
- Within a context, information can be further organized into a folder structure. However, in addition to just providing organized storage, contexts also provide the capability to apply administration to the objects they store. For example, rules can be established to specify which users can access and/or modify information within a context. Your administrator must develop a context strategy that balances administrative requirements with logical information storage.



- Modification are tracked and controlled using revisions and iterations.
- Version:
- Revision
- Iteration
- Check out
- Check In



- Windchill uses an attribute called version to track and control object modification. An object version is composed of two elements: the character to the left of the decimal is the revision, and the character to the right of the decimal is the iteration. The out-of-the-box versioning scheme configured for all Windchill data objects uses a letter for the revision and a number for the iteration. Your administrator may assign a unique
- versioning scheme for each data type based on the best practices for your company or industry.

To modify a data object, you must check it out. This locks the object from modification by any other user and signals to other users your intent to modify the object. The checkout action creates two copies of the object:

- A working copy that you can modify.
- A checked out copy that others can still access

Once you have made your changes, you check the object back in. The system uses the working copy to generate a new iteration of the object. The new iteration is now available to other users for modification.

Your business rules may preclude you from altering information past a certain development state. In this situation, you must use a New Revision action to generate a new object version before making any changes. As part of the revision process, Windchill applies business rules and administrative settings in generating the new object version. These include, but are not limited to:

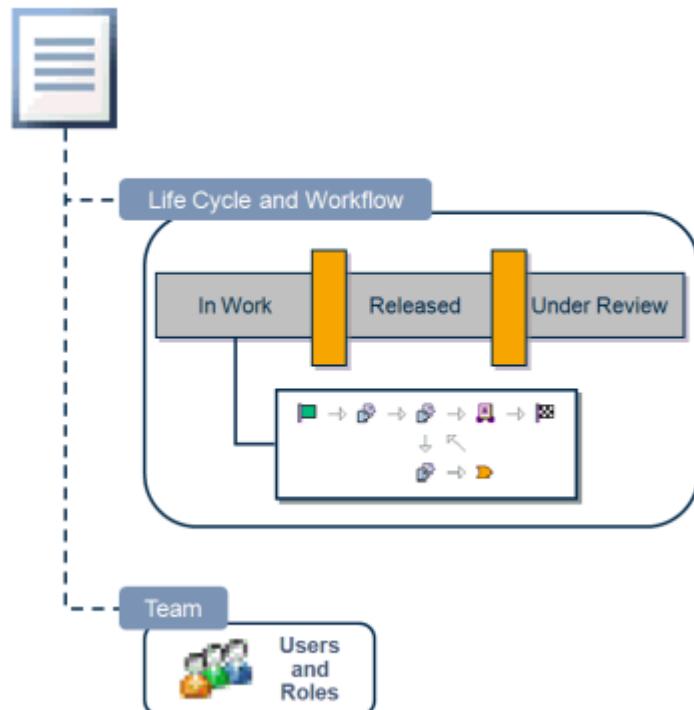
- Incrementing the Revision number or letter to the next value in the version sequence.
- Resetting the iteration number or letter to the beginning of its sequence.

- Setting the life cycle state. Typically, the administrative rules designate a state with access rules that enable modification using check out and check in operations



Process Controls

- Process controls automate many of your business process
- Automated Process Components
- Life cycles : transitions
- Work Flows
- Teams



Windchill can also automate many of your business processes. The process management tools are extremely flexible and can be configured to represent your company's best development practices. Windchill uses a combination of objects to conduct process management:

- Life Cycles
- Workflows

Teams A life cycle establishes the stages or states that an object, such as a document, can exist in. For example,

your company has decided that documents can only exist in one of these three states: In Work, Released, or Under Review. The gray boxes in the diagram represent these three states

Each of these states (and the gates that follow them, shown above as yellow boxes) can use a workflow process to automatically route development and review tasks to the appropriate people. As major milestones in the development are completed, the workflows route the document to the appropriate state



- Access controls determine your ability to perform actions on certain objects.
- Types of Permissions
- Full Control
- Read
- Modify
- Create
- Revise
- Delete



The information that you create and store in Windchill is the valuable intellectual capital of your organization. Consequently, your administrators provide you with the appropriate access controls. Access controls determine whether you can read, modify, create, delete, or revise each type of object in Windchill.

If you determine that you are not able to access an object or perform an action, such as creating a document, your permissions are probably set to prevent that action.

You can be granted access through domains applied to storage contexts, through life cycles, or through security policies applied directly to folders or individual objects.

Any documents, Part files you can give access to everyone or particular people according to requirements.



Collaboration Features



- Almost every aspect of Windchill is designed with collaboration in mind. However, there are tools that specifically foster collaborative communication:
- You can use discussion forums to discuss design information. Discussion forums are message boards that you can subscribe to so you can be notified every time they are modified.
- Windchill's visualization tool, Creo View, enables you to view and mark up data even if you do not have the native application used to generate the content.
- You can also schedule and host Web-based meetings about Windchill objects.



Logging On to Windchill

- Log on Process:
- Open your web browser
- Type URL
- Log on
- User Name
- Password

Sign in to access this site

Authorization required by <http://dtcoedws017.gttc.com>
Your connection to this site is not secure

Username:

Password:

Sign in **Cancel**

- Double-click the Internet Explorer icon on your desktop.
- Click the Windchill Server link in the Internet Explorer Favorites bar to access Windchill.
- Type areed in the User name field.
- Type ptc in the Password field.
- Click the OK button to log on to Windchill.
- Windchill authenticates your user name and password, and you are logged on to the system



The header of the Windchill User Interface. It features a dark grey background with a yellow and red circular logo on the left. In the center, the text "Understanding the Windchill User Interface" is displayed in white. On the right side, there is a white circular button containing the "DesignTech Edu" logo and the word "Empirical Learning" below it.

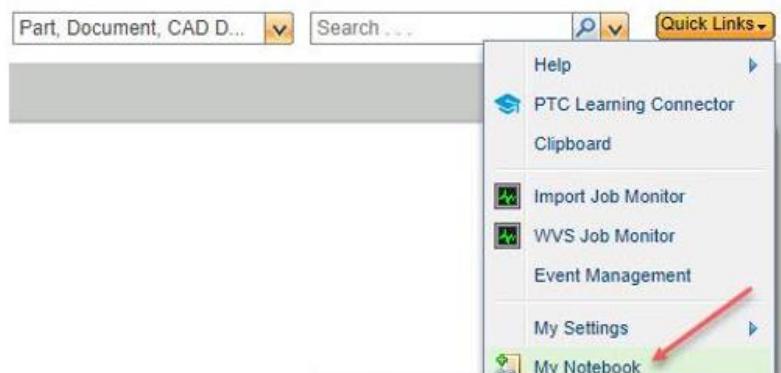
- The Windchill user interface is composed of three sections:
 - Header
 - Page
 - Navigator Panel
- Windchill system has a simple and consistent layout, that enables you to quickly navigate and work with your enterprise information. The Windchill interface is composed of three sections:
 - The Header contains a welcome message, the ability to perform a simple search, and a series of useful tools available from the Quick Links drop-down menu.
 - The Page displays information associated with your navigation. Your work consists of: reviewing data or performing actions on objects and information presented in the tables and forms available on the Windchill pages.
 - The Navigator Panel provides access to advanced search utilities and the ability to browse contexts to which you have access.

- The Home page provides information that is specific to you



- Click the Home icon to access the Home page.
- Click the Customize button to expand the Customize menu.
- Select the Data Monitors option to add the Data Monitors table.
- Select the Saved Reports option to add the Saved Reports table.
- Select the Reports option to add the Reports table.
- Select the Discussions option to add the My Discussions table.
- Select the Subscriptions option to add the Subscriptions table.
- Select the Notebook option to add the My Notebook table.
- Select the Meetings option to add the My Meetings table.
- Select the Workspaces option to add the My Workspaces table.
- Select the Packages option to add the Packages table.
- Move the Checked-Out Work table to the top of the Home page by clicking the Checked-Out Work link at the top of the page and dragging it from its current position to a position left of the Packages link.
- Move the Updates table to the top of the Home page by clicking the Updates link at the top of the page and dragging it from its current position to a position left of the Checked-Out Work link.
- Move the My Tasks table to the top of the Home page by clicking the Tasks link at the top of the page and dragging it from its current position to a position left of the Updates link.

- The quick Links menu enables you to access commonly used tools, preference and information.



- The Quick Links menu enables you to access commonly used tools, preferences, and information. The Quick Links menu is located in the upper-right corner of the Windchill User Interface. The Quick Links Menu contains the following:
 - The Help option opens the Help menu, which provides access to a variety of Help tools and utilities.
 - The Clipboard option enables you to cut, copy, and paste objects and URLs in the Windchill system.
 - The Import Job Monitor option enables you to track each imported Microsoft Excel spreadsheet that contains parts and product structures.
 - The WVS Job Monitor option opens the WVS Job Monitor page from which you can view the status of Visualization Publish jobs.
 - The Event Management option enables workspace users to view and manage system log messages generated from their authoring or modeling applications.
 - The My Settings option opens the My Settings Menu, from which you can set your Preferences, view and update your Profile, and manage your Windchill calendar.
 - The My Notebook option opens the My Notebook table, which displays links and files related to your work.
 - The Software Downloads option opens the Software Downloads page, from which you can install supporting and integration software for your Windchill implementation.
 - The Email Page option opens the E-mail Page utility, which enables you to send an e-mail referencing the currently viewed Windchill page.
 - The Copy Page option enables you to copy the current Windchill page to the clipboard.
 - Use the Navigator to search for information and navigate the Windchill environment. The Navigator is composed of two tabs, Search and Browse.
 - The Search tab provides access to a Search History list, a Saved Searches list, and the Advanced Search utility. The Browse tab provides a series of icons that enable you to navigate to all of the contexts to which you have access.
 - A Recently visited option on the browse tab provides quick access to contexts you have recently accessed. Initially, the Navigator is collapsed to enable maximum screen area for the Windchill pages.

- You can click the Expand icon to expose the Navigator, or you can use the Search or Browse links at the top of the Navigator to quickly access the desired Navigator tab. The navigator normally collapses when you make a selection that changes the main page. When you need to keep the Navigator open while.
- working in the main page, you should pin it using the Pin Navigator icon. If you later need more space in the main page, collapse the navigator using the Unpin Navigator icon.
- A drag handle on the far-right border of the Navigator enables you to adjust the width of the Navigator. Finally, a Home icon in the upper-right corner enables you to quickly access your Home page.



Using the Breadcrumb Trail

The screenshot shows the Windchill interface with a dark header bar. On the left is a yellow circular icon with a red dot. To its right is the title "Using the Breadcrumb Trail". On the far right is a red circular icon containing the text "DesignTech Edu" and "Empirical Learning". Below the header is a breadcrumb trail bar with a house icon followed by the path: Products > Chainsaw - 220 Series > Engineering.

- The Breadcrumb trail displays the navigation hierarchy of the table or object you are currently viewing.

- 
- The breadcrumb trail, located to the right of the Home icon on the Windchill page, displays the navigation hierarchy of the table or object you are currently viewing. This includes the object's context and any folder directory structure or workspace used to store that object.
 - You can view the full length of longer folder paths by cursoring over the breadcrumb trail. Folders and contexts displayed in the breadcrumb trail also serve as links to their respective information pages and lists in Windchill.



Navigating Products and Libraries

The screenshot shows the Windchill interface with a dark header bar. On the left is a yellow circular icon with a red dot. To its right is the title "Navigating Products and Libraries". On the far right is a red circular icon containing the text "DesignTech Edu" and "Empirical Learning". Below the header is a breadcrumb trail bar with a house icon followed by the path: Products > Chainsaw - 220 Series > Engineering.

- Products and libraries store and manage product information:
 - Details
 - Folders
 - Workspace
 - Packages
 - Team
 - Tasks
 - Change Monitor
 - Discussions
 - Reports
- The Product and Library options on the Navigator display information about the products and libraries to which you have access. A Product represents storage for information related to a specific



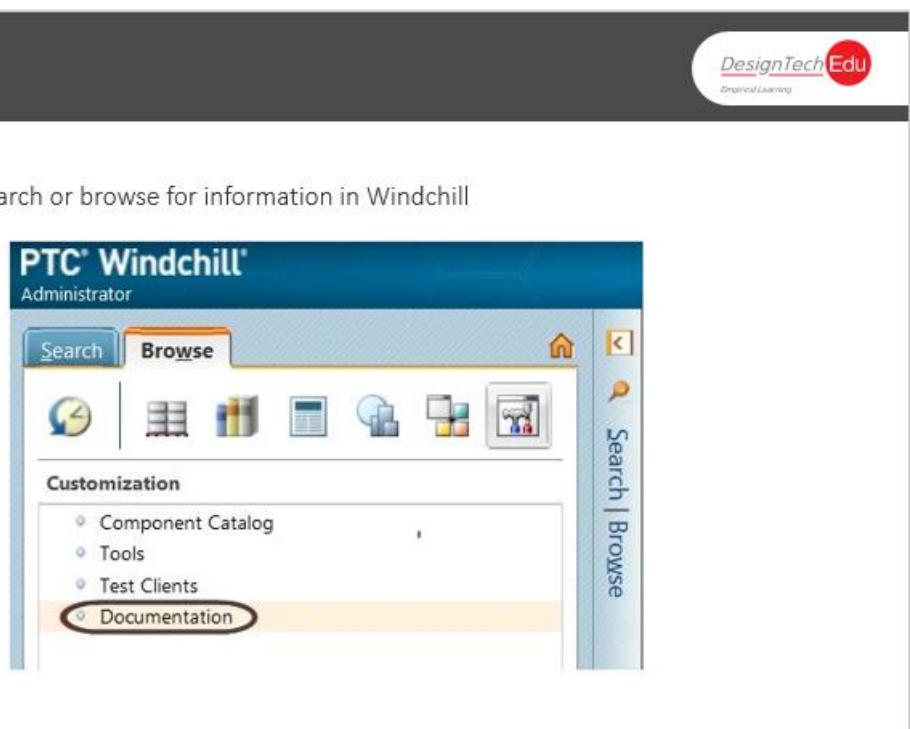
product or deliverable, where a library provides a place for storing and providing access to general business information that is not centered on a specific product or deliverable.

- The administrator for your implementation will set up products and libraries that correspond to your company's data, organization, and processes. Navigating a product or library begins by adding the context to the Navigator. This can be accomplished by clicking the View All link to access the Products or Libraries tables. From the Products or Libraries tables, you select the desired context.

Once a context is selected, additional information about the product or library is accessed by navigating a set of links below the context node on the Navigator:

- The Details link displays attributes and information about the context.
- The Folders link displays the contents of the current context in a folder structure.
- The Workspaces link displays all of your workspaces for the current context.
- The Packages link displays all of the packages for the context and delivery information for those packages
- The Team link displays all team-related information and actions for the team members of the current context.
- The Tasks link displays and manages tasks that are assigned to the context team.
- The Change Monitor link displays change information and status for the context.
- The Discussions link displays all discussion topics for the current context.
- The Reports link displays the reports available to run for the current context and the report results

Using the Navigator



The screenshot shows the PTC Windchill Administrator interface. At the top, there is a navigation bar with icons for Home, Search, and Browse. Below the bar, the title "PTC Windchill" and "Administrator" are displayed. On the left, there is a vertical "Customization" panel containing options like Component Catalog, Tools, Test Clients, and Documentation, with "Documentation" highlighted. The main area shows various icons for search, browse, and other system functions. A vertical sidebar on the right is labeled "Search | Browse".

- The Navigator enables you to search or browse for information in Windchill
 - Search:
 - Search History
 - Saved Searches
 - Advanced Searches
 - Browse:
 - Recently visited
 - Products
 - Changes
 - Libraries
 - Pin the Navigator

You can access change information and gauge the status of the change processes within your system using the Changes option on the Navigator. The Changes option exposes a series of selections that enable you to navigate to specific change information.

- The Issues selection displays all of the problem reports for which you are a team member.
- The Variances selection displays all variances for which you are a team member.

- The Change Requests selection displays all the change requests for which you are a team member.
- The Change Notices selection displays all the change notices for which you are a team member.
- The Change Monitor selection displays the Change Monitor page, which enables you to gauge change process status for various contexts.
- The Reports selection displays change reports available to run and the report results.

Navigate a Product context.

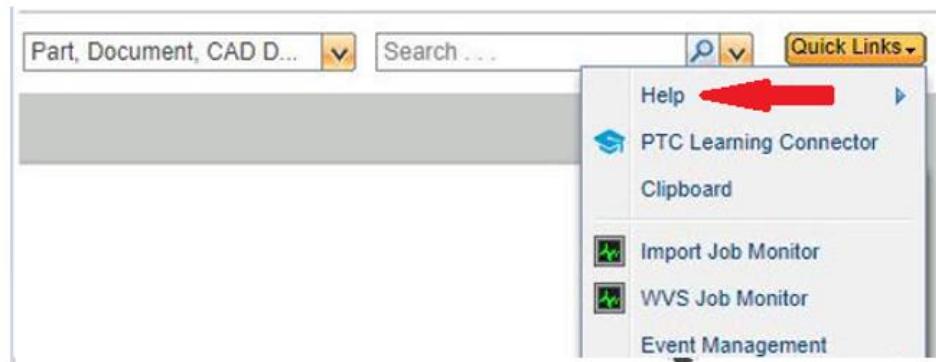
- Click the Expand icon to expand the Navigator.
- Select the Browse tab.
- Click the Recent Products icon.
- Click the View All link to access the Products table.
- Click the Chainsaw - 220 Series link to access the product
- Click the Browse link at the top of the Navigator.
- Click the Expand icon to the left of the Chainsaw - 220 Series node to expose the navigation options.
- Click the Details link to view the product information page
- Click the Discussions link to view the Discussions table.
- In the Discussions table, click the Expand icon to expand all nodes.
- Click the Reports link to view the Reports table

Navigate Changes.

- Click the Changes icon.
- Click the Issues link to view the Issues table.
- Click the drop-down menu to the right of the Issues label.
- Select All Open from the drop-down menu to view all open problem reports.
- Click the Change Monitor link to view the Change Monitor page. Note: In a live classroom environment, the system clock reflects the current date. Change data staged in the past to support the class exercises may now fall outside of the Report Time Period range and, therefore, may not appear in some of the Change Monitor graphs.
- Click the Recently Visited icon.
- Click the Unpin Navigator icon to collapse the Navigator.



- Windchill has many source of help.



- In the upper-right corner of the Windchill window, click the Quick Links button to expose the Quick Links menu.
- Select Help from the Quick Links menu to open the Help menu.
- Select Windchill Help Center from the Help menu to open the Windchill Help Center window.
- Click the Windchill Help Center Maximize icon.
- In the left panel, click the Book Expand icon for the PTC Windchill Fundamentals topic to expand the topic.
- In the left panel, click the Working with Windchill Objects topic to expand the topic.
- In the left panel, click the Book Expand icon for the Object Overview topic to expand the topic.
- In the left panel, click the Book Expand icon for the Information Pages topic to expand the topic.
- Click the Mini Information Pages topic to open the help information in the right panel.
- Click the Close button to close the Windchill Help Center window.

Specialization program in PLM- Windchill

Day 2 Locating Information

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1



Using Simple Search

- The simple search utility available in the Windchill header enables you to locate information using keyword criteria.
- Simple search capability is controlled by two factors:
 - Scope
 - Object types
 - Keyword Criteria
 - System Configuration
 - Standard Search
 - Index Search



Using Advanced Search

- Advanced search enables you to narrow your search scope by adding specific criteria based on context, object attributes or object relationships.

The screenshot shows the 'Advanced Search' interface. At the top, there are tabs for 'Search' and 'Browse', with 'Search' being the active tab. Below the tabs, there are links for 'Search History and Saved Searches' and 'Advanced Search'. The main area is titled 'Advanced Search' and contains the following sections:

- Type:** A section with a tree view. Under 'All Types', 'My Favorite Types' is selected. Under 'My Favorite Types', several object types are listed with checkboxes:
 - Abstract Specif... (selected)
 - Lot Baseline
 - CAD Document
 - Part
 - Change Notice
 - Problem Report
 - Change Request
 - Requirement
 - Document
- Context:** A section with a tree view. Under 'All Contexts', 'My Favorite Contexts' is listed with an 'Add' link.
- Criteria:** A section where search criteria can be defined using logical operators (AND, OR) and comparison operators (=, <=, >=, !=). Two fields are shown:
 - Name = []
 - Number = []



Using Advanced Search - Nested Queries

- Nested groups provide greater flexibility using the AND and OR operators when specifying criteria attributes.

Criteria

((Name = '*Requirements') AND (Iteration = 'Latest') AND ((State = 'Released') OR (Modified By = 'Smith, Fred')))
[Explain](#)

The screenshot shows a search criteria builder with the following structure:

- An **AND** operator at the top level.
 - A **Name** field set to `= *Requirements`.
 - A **Iteration** field set to `= Select: Latest`.
- An **OR** operator below the first **AND**.
 - A **State** field set to `= Released`.
 - A **Modified By** field set to `= Smith, Fred`.

A purple oval highlights the entire **AND** expression, and a red box with the number **1** highlights the **OR** expression.



Search Results Table

- The Search Results table displays all objects returned for your search

The screenshot shows a search results table titled "Updates Most Recent". The table has the following structure:

	Name	Number	Version
<input type="checkbox"/>	0000000866.prt	0000000686	i A.0 (Design)
<input type="checkbox"/>	0000000867.prt	0000000684	i A.0 (Design)
<input type="checkbox"/>	0000000843.asm	0000000701	i A.0 (Design)
<input type="checkbox"/>	0000000864.prt	0000000699	i A.0 (Design)
<input type="checkbox"/>	0000000861.prt	0000000708	i A.0 (Design)
<input type="checkbox"/>	0000000848.prt	0000000691	i A.0 (Design)

(0 objects selected)

Managing Searches

The screenshot shows the 'Search' tab selected in a software application. At the top, there are tabs for 'Search' (which is active) and 'Browse'. Below the tabs, there are two main sections: 'Search History' and 'Saved Searches'. The 'Search History' section is titled 'Today' and lists three search entries. The first entry is 'Type=Document; State=Released; Context>All Contexts'. The second entry is 'Type=Document; Name=*Requirements; State=Released; Context>All C...'. The third entry is 'Keyword=*Requirements; Type=Document; Context>All Contexts'. Each entry has a pencil icon to its right, indicating it can be edited. To the right of the search history, there is a 'Saved Searches' section with a 'Manage' link. This section is titled 'Created by Me' and shows one entry: 'Released Documents' with edit and delete icons. A vertical 'Navigator' sidebar is visible on the right side of the interface.

The Search History and Saved Searches sections of the Search tab enable you to manage your searches. The Search History list provides access to searches that you have recently conducted.

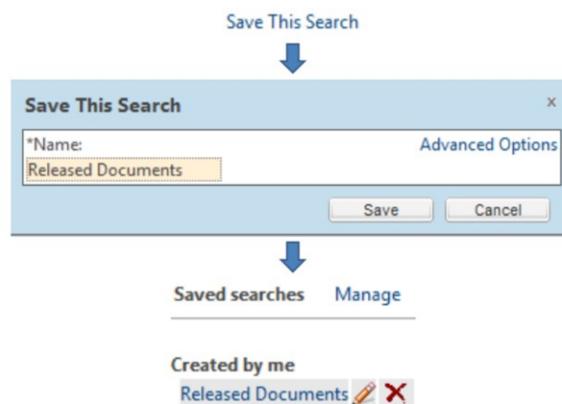
Each search is identified by a summary of the criteria used to conduct the search. The Saved Searches list provides access to searches that you have saved and any searches that your administrator has created for you. Both lists have similar tools for managing the searches:

- To execute a search again, click an entry in the Search History or Saved Search list.
- Use the cursor-over feature to access more details for a search. This will expose a small window that enables you to view all of the search criteria, to edit the search, and to run the search again.
- Use the Edit criteria icon to further refine the search criteria or to use the search as a template.

Use the Delete this search icon to delete any saved searches you have created. The Manage link at the top of the Saved Searches list enables you to further manage your Saved Searches. The Manage link exposes the Manage Saved Searches table, which enables you to delete, show, hide, export, or import searches, in your Saved Searches list

Saving searches

- When using advance search, you can save and name your search criteria so you can execute the search again in the future.
- To save a search, you click the save. This search link once you have established your search criteria.



Following steps are for Save the advanced search.

- Click the Save This Search link to open the Save this Search window.
- Type Released Documents in the Name field.
- Click the Save button to save the search.
- Click the Search History and Saved Searches link to access the Saved Searches list.
- Notice that the Released Documents search is displayed in the Saved Searches list.

Browsing for objects

- Browse for objects within a context's folders.

Browse for objects in the following folders.

- Product Folders
- Library folders.

Modify table views

- Filter the table contents table
- Sort columns

The screenshot shows a software interface for managing objects. On the left, a 'Folders' panel lists four objects: 'GTTC Belgaum', 'Change log', 'General', and 'Policies'. On the right, a 'Folder Contents' panel for 'CAD Documents' shows a table with columns for Number, Name, Version, State, and Last Modified. The table currently displays '(0 objects selected)'. Above the table are buttons for 'Search in table', 'Filter', and 'Sort'. The entire interface is enclosed in a red border.



Using Mini Information Pages

- The mini information page assists with locating information while conducting searches and browsing.
- Provides basic object information:
 - Attributes
 - Where Used
 - Uses Tools
- Navigation Controls



- Lock/ Unlock



- Action Icons



When you cursor over mini-thumbnails in the Search Results or Folder Contents tables, a mini information page opens providing brief information about the object. The left side of the mini information page provides a larger view of the object's default thumbnail and basic attribute information.

Specialization program in PLM - Windchill

Day 3 viewing Information

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Using windchill table



- Windchill tables have a consistent layout and use common functions.
- Viewing Information
- View drop-down menu
- Table display options
- Column configuration options
- Change table height
- Find in table utility

As you navigate and manage information in Windchill, you will find that many of your interactions involve working with tables. Windchill tables use a consistent layout, common set of functions, and often provide the capability to manage and build custom views. You can divide your interaction with Windchill tables into two categories; viewing information and performing actions on information.

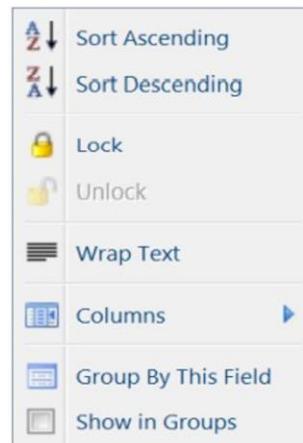


Working with table columns



Windchill table columns can be configured to better suit your work.

- Column Actions:
- Click to sort
- Adjustable column widths
- Drag to reorder
- Column menu
- Sort Ascending/Descending
- Lock/Unlock
- Wrap Text
- Columns
- Group By This Field
- Show in Groups



Column Menu

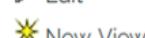
Windchill tables enable you to configure the columns to achieve a layout that better suits your work.



Managing windchill table views



The Customize View List table enables you to manage Windchill table views.



Windchill tables display objects in rows with information about each object listed in columns.



New view: Set Name

- In the Set Name step window, you are required to provide a name for your view.

New View - Windows Internet Explorer

http://ptc-training.ptc.com/Windchill/servlet/WindchillAuthGW/wt.enterprise.URLProcessor/processForm?section=CreateOrEditTableView&u

New View

Set Name Choose Object Types Set Filters Set Column Display Set Sorting

List: Product and Library Folder

*Name: Released Requirements Documents

Description:

Show in current view list

*Indicates required fields

Done Back Next Finish Cancel

Trusted sites | Protected Mode: Off 100%



New View: Choose object types

- In the Choose Object Types step window, you select which object types the view manages

The screenshot shows a Windows Internet Explorer window titled 'New View - Windows Internet Explorer'. The URL in the address bar is <http://ptc-training.ptc.com/Windchill/servlet/WindchillAuthGW/wt.enterprise.URLProcessor/processForm?action=CreateOrEditTableView&u8>. The main content area is titled 'New View' and contains five numbered steps: 1. Set Name, 2. Choose Object Types (which is currently selected), 3. Set Filters, 4. Set Column Display, and 5. Set Sorting. Step 2 is highlighted with a yellow circle containing the number 2. Below the steps is a section titled 'Object Types' with a checkbox next to 'Name'. A list of 61 objects is displayed, all of which have checkboxes checked. The objects listed include: Name, Agenda, Book, Burst Configuration, CAD Document, Change Directive, Change Notice, Change Request, Chapter, Configuration Context, Configuration Item, Context Reference, Deviation, and Document. At the bottom of the list, it says '(61 objects selected)'. Below the list is a note: '*Indicates required fields'. At the bottom right of the window are buttons for Back, Next, Finish, and Cancel. The status bar at the bottom right shows '100%'.

In the Choose Object Types step window, you select which object types the view manages.



New view: Set Filters

- In the Set Filters step window, you specify the rules that govern which items are displayed in table rows.

The screenshot shows the 'New View' window in a Windows Internet Explorer browser. The title bar reads 'New View - Windows Internet Explorer'. The URL in the address bar is <http://ptc-training.ptc.com/Windchill/servlet/WindchillAuthGW/wt.enterprise.URLProcessor/processForm?action=CreateOrEditTableView&u>. The main content area is titled 'New View' and contains five numbered steps: 1. Set Name, 2. Choose Object Types, 3. Set Filters (which is highlighted with a yellow circle), 4. Set Column Display, and 5. Set Sorting. Step 3 is currently active. Below these steps, there is a section for 'Object types' with a dropdown set to 'Document'. Under 'Criteria', there is a dropdown set to 'Cabinet' with an 'Add' button. A 'Filter Criteria' section is expanded, showing two filter entries:

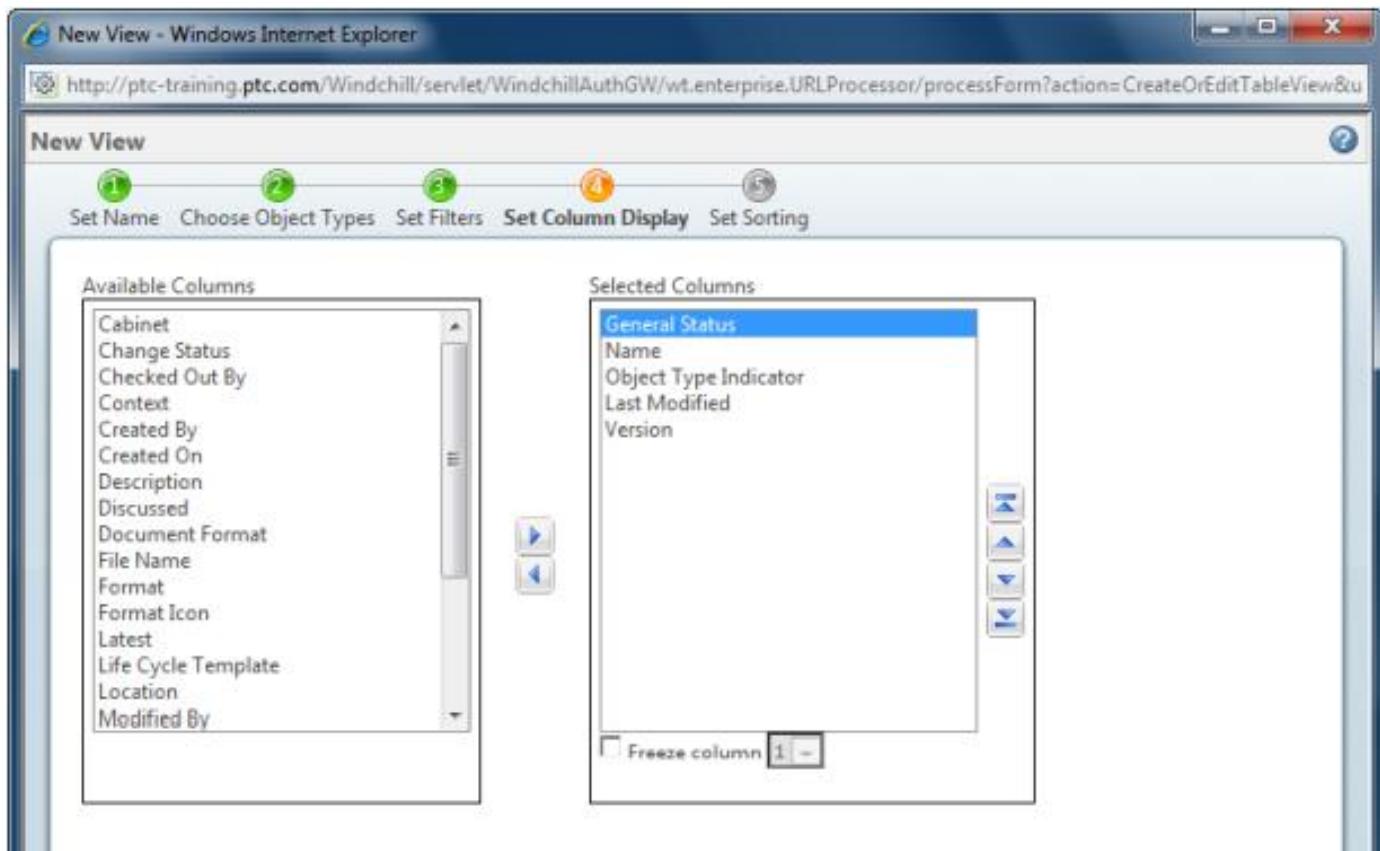
Name	Operator	Value
Name	Equal to	"Requirements"
State	Equal to	Released

There are buttons for 'Match All' (unchecked) and 'Search in table'.

- In the Set Filters step window, you specify the rules that govern which items are displayed in table rows.
- Attributes specific to the selected object type populate the Criteria drop-down list

New View: Set Column Display

- In the Set Column Display step window, you select the columns displayed in the table and the order in which those columns appear.



New View: Set Sorting

- In the Set Sorting step window, you define sorting criteria for the table objects.

The screenshot shows the 'Set Sorting' step window within the 'New View' dialog. The window title is 'Sorting'. It contains a table with three rows for sorting orders:

Sorting Order	Column	Mode
First Sort	Name	Ascending
Second Sort	--Select a Column--	Ascending
Third Sort	--Select a Column--	Ascending

Below the table is a note: "*Indicates required fields". At the bottom are buttons for Back, Next, Finish, and Cancel. The status bar at the bottom shows 'Done' and 'Trusted sites | Protected Mode: Off'.

- In the Set Sorting step window, you define sorting criteria for the table objects.

Understanding Information Pages

- Regardless of the object type, information pages have a consistent layout. To access an information page, click the View information icon .

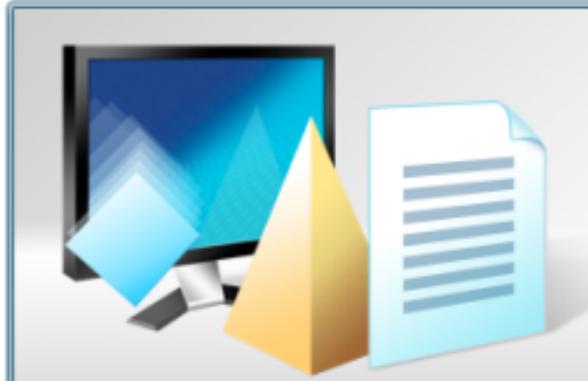
Actions ▾ Document - 0000000086, Chainsaw 220 Owner's Manual, A.2 In Work ?

Actions and Object Name

Details Structure Content Related Objects History +

Tabs

Visualization and Attributes | More Attributes



General

Name: Chainsaw 220 Owner's Manual
Status: Checked in
Primary Content: Chainsaw 220 Owner's Manual.doc  
Modified By: Smith, Fred
Last Modified: 2011-02-28 17:28 EST

System

Description:
Format Name: Microsoft Word State: In Work - Released - Canceled
Context: Chainsaw - 220 Series Location: Chainsaw - 220 Series / Documentation
Life Cycle Template: Basic Team Template:
Created By: Smith, Fred Modified By: Smith, Fred
Created On: 2011-02-28 17:24 EST Last Modified: 2011-02-28 17:28 EST

Specialization program in PLM- Windchill

Day 4 - Windchill MCAD Data management Process Overview

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Cad Data Management Process Overview

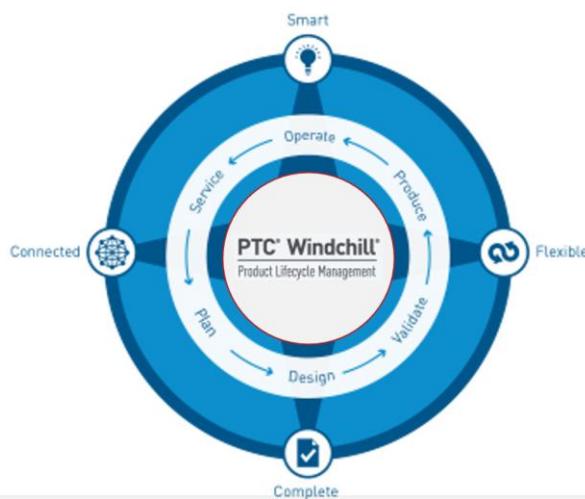
- There are two high level steps in the CAD data management process.
- Manage Design data
- Manage Design

Development

- One of Windchill's functions is as a CAD design data management system intended specifically to address information control and development processes.
- It is capable of managing data for many different CAD systems.
- There are two high-level steps:
- **Manage Design Data** – The goal of this step is to enable the creation and management of CAD data from various CAD authoring tools integrated with the Windchill capabilities.
- **Manage Design Development** – Manage the concurrent development of CAD designs from concept through retirement, ensuring participants have access to the required information

CAD and Windchill Connectivity

- CAD application is integrated with Windchill.

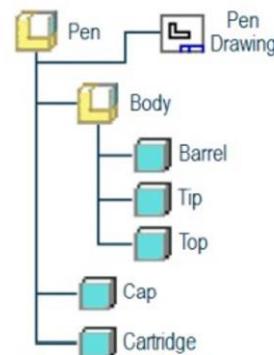


- The product design process generates a tremendous amount of data. To reduce a product's time-to-market, it is beneficial for the engineering team to share design information in a collaborative environment.
- The challenge of a collaborative environment is maintaining control of the data while distributing it to the people who need it. Without the ability to control design data, engineers are often confronted with using different variations of the same data, which can result in overlapping or inconsistent designs.
- One of Windchill's functions is as a data and configuration management system designed specifically to address information control and development processes. It is capable of managing data for many different CAD systems.

- However, this course is specific to Creo Parametric data and primarily focuses on managing Creo Parametric design files, which Windchill accomplishes by using CAD document objects and WTParts (Windchill Technology Parts).
- CAD documents maintain the complex associations and relationships of the stored design files while providing database objects that you can control and administer.
- Each WTPart represents a potential BOM object that is associated to its respective CAD document. The WTParts provide the ability to perform activities such as Configuration Management

Introduction to CAD Documents

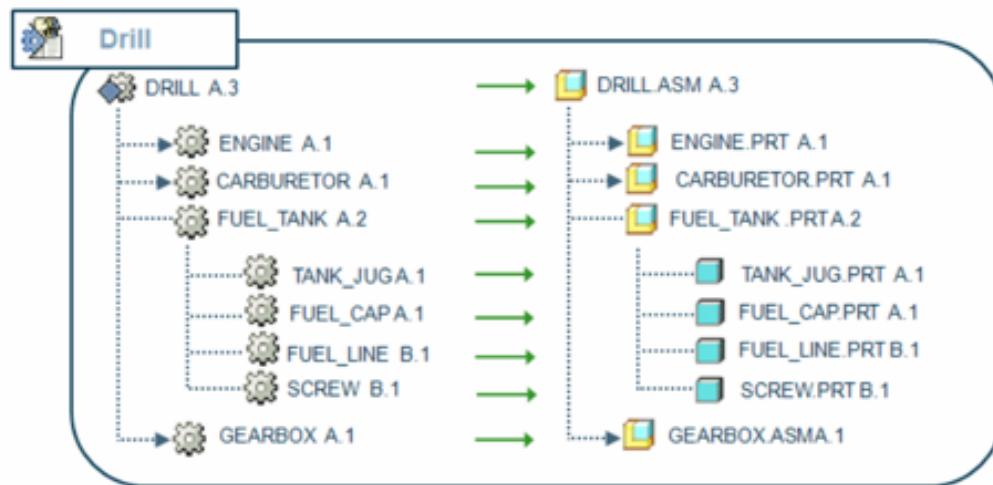
- CAD document objects store and manage information and content.
- CAD Documents



- Windchill uses CAD Document Business Objects to store and manage CAD content and information developed using a CAD authoring tool. CAD document objects store:
- The CAD authoring tool file (primary content).
- Attributes.
- Version history.
- Model structure information.
- Attachments (secondary content).
- Windchill displays an appropriate CAD document icon based on the CAD file type stored as primary content.

Introduction to Windchill Parts

- Windchill parts are metadata objects that may be associated with CAD documents



- In general, a Windchill Part (WTPart) represents a set of metadata (attributes and other related information) that may be described by a CAD document, such as a CAD part or assembly. A WTPart does not have content.
- Instead, it may have a representation of a CAD file, such as a Creo View image. Typically, a WTPart contains more information than what is contained in the CAD document.

Windchill Integration

- Windchill has multiple access points within your CAD application. Windchill access points for CAD:
 - CAD menus
 - Model tree
 - Folder Browser
 - Search
 - Embedded browser
 - Event Management

After you register and activate a Windchill server, a set of tools and utilities are exposed that enable you to upload and manage design information with CAD documents.

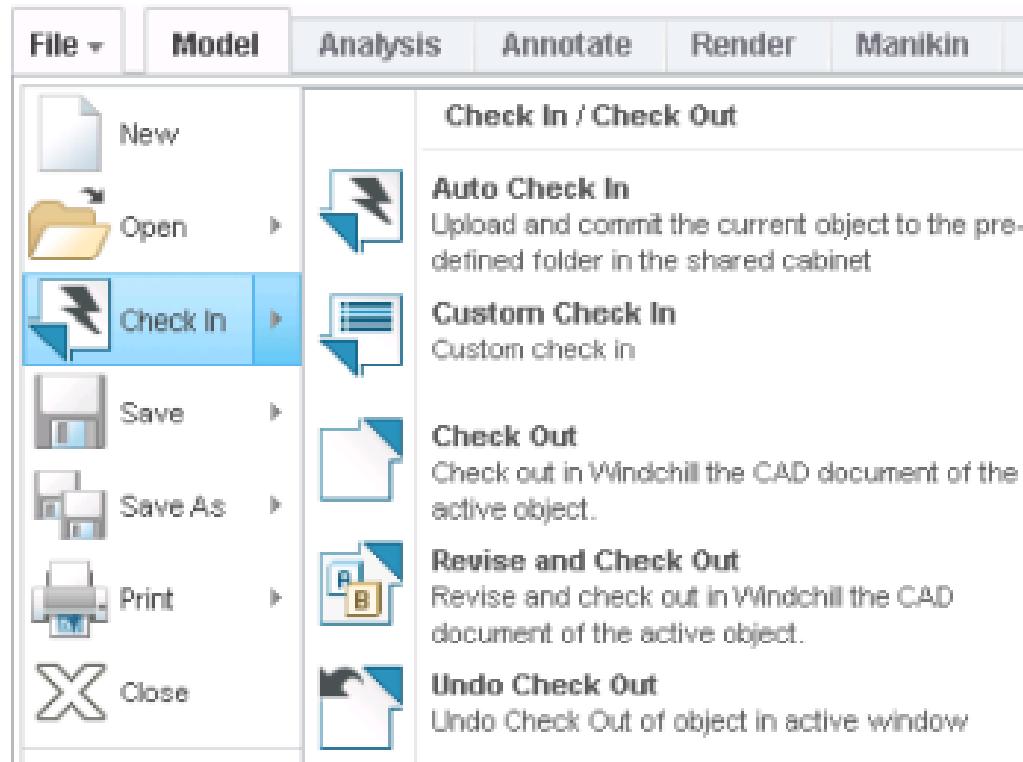
- Windchill access points in your CAD application can be found in:
 - CAD menus
 - Model tree
 - Folder Browser
 - Search

- Embedded browser
- Event Management These tools and utilities are exposed through various changes and additions to the CAD application user interface.

Windchill Integration CAD menus

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- Windchill actions are found in CAD application ribbons/ menus.
- Windchill access point:
- CAD ribbon / menus

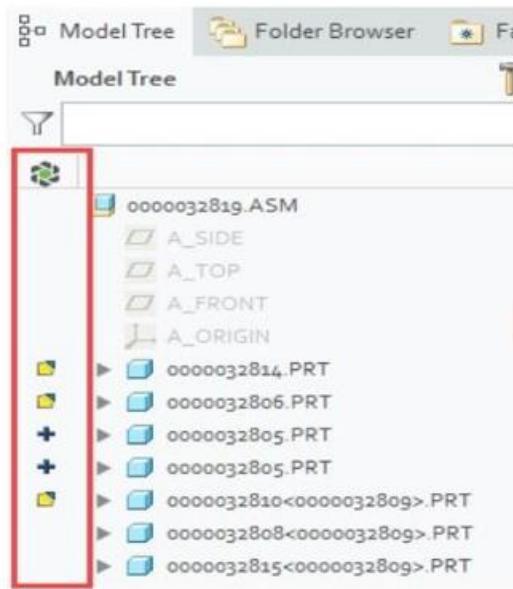


- Additional options are added to your CAD application menus and/or ribbons. These options enable you to perform many of the most common data management functions, such as Check In and Check Out.



Windchill Integration – Model Tree

- Windchill has multiple access points in Creo parametric.
- Windchill access point :
- Model tree

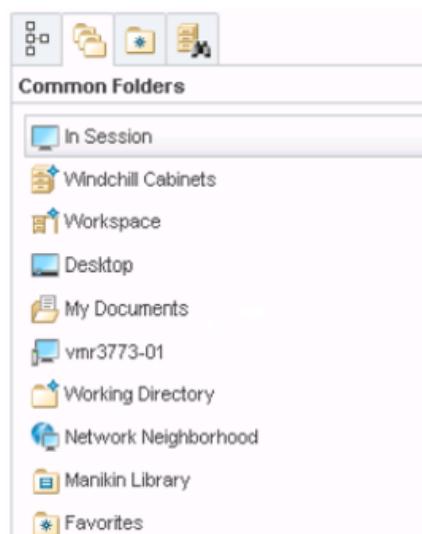


- Menu options are added to object action menus in the model tree. This enables you to perform many common data management functions from within the model tree while working with design files



Windchill Integration – Folder Browser

- Windchill has multiple access points in Creo parametric
- Windchill access point :
- Folder Browser



- Nodes for both the active server and active/inactive workspaces are added to the Creo Parametric Folder Browser. This enables you to quickly browse for design information in both your workspaces and the contexts (common space) to which you have access.



Windchill Integration – Search

- Windchill has multiple access points in Creo Parametric.
- Windchill access point:
- Search

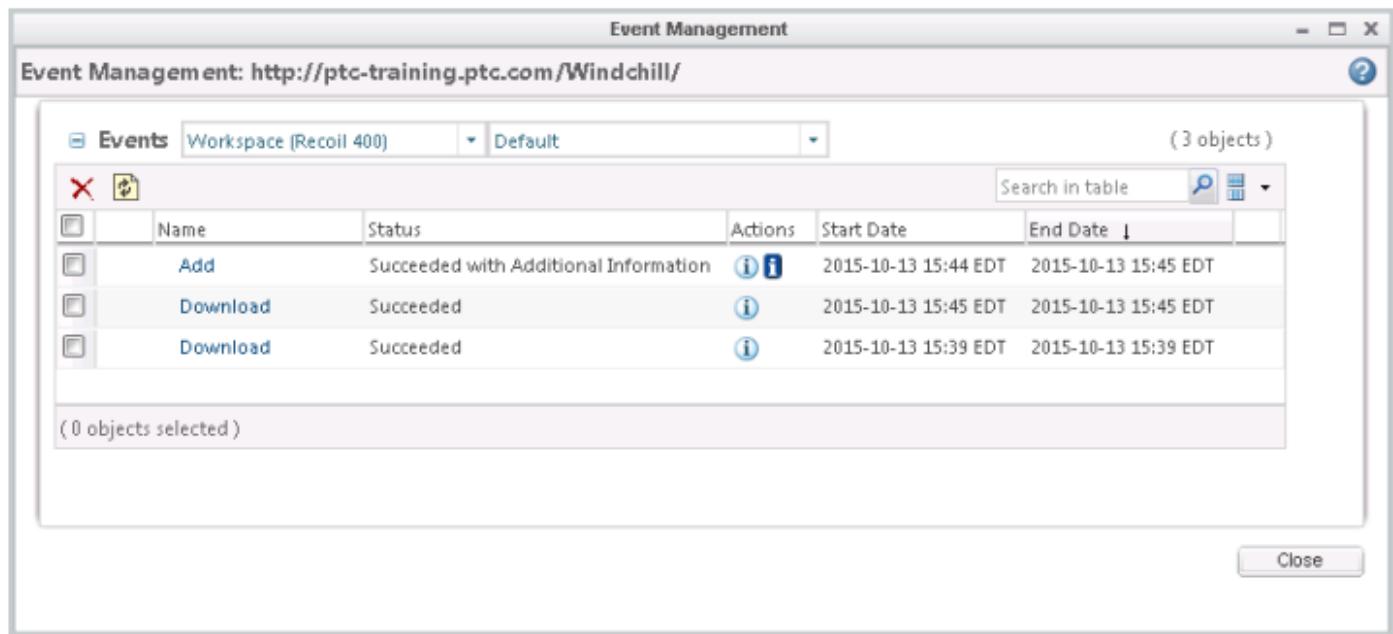


- The Search tab is added to the Creo Parametric Folder Navigator. The Search tab enables you to search for CAD models or Mathcad Worksheets on the primary Windchill server.



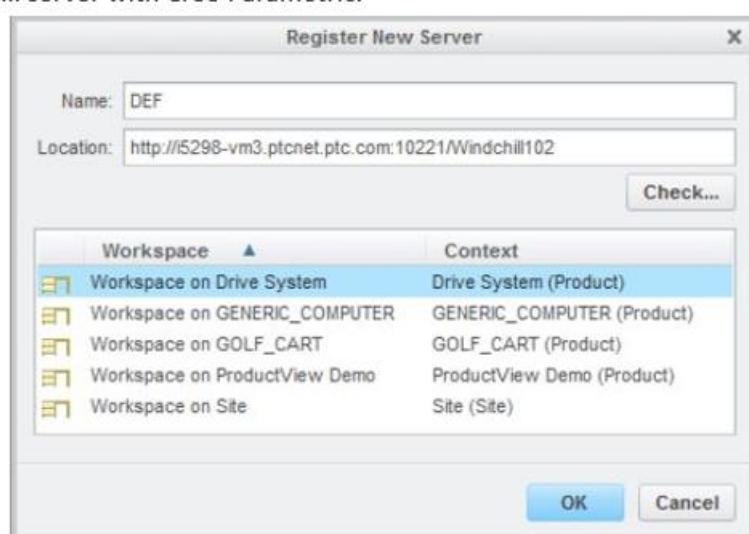
Windchill Integration – Event Management

- Windchill has multiple access points in Creo Parametric.
- Windchill access point:
- Event Management



Registering a Windchill Server

- You must first register a Windchill server with Creo Parametric.
- Server Management
- Register New Server:
 - Name
 - Location
 - Check Connection
 - Set Primary Server



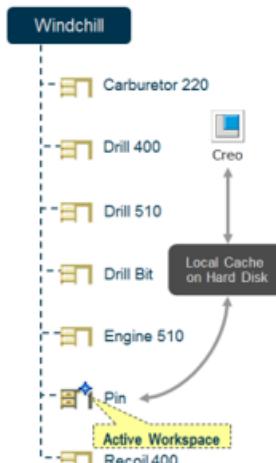
- To generate CAD documents and manage Creo Parametric data, you must upload data to Windchill from Creo Parametric. To do this, you must first register Windchill with Creo Parametric.

Configuring the Creo Parametric Browser

- You have the ability to configure the Creo Parametric browser.
- Embedded Browser location (web_browser_in_separate_window)
 - Within Creo (default)
 - Separate window
- Embedded Browser location (web_browser_in_separate_window)
 - Within Creo (default)-
 - Separate window
- To enable you to access the full functionality of Windchill, Creo Parametric is equipped with a built-in (embedded) Web browser based on Internet Explorer (default) or Chrome.

Introduction to the Workspace

- The workspace is the repository that facilitates communication between Creo Parametric and Windchill
- The workspace does the following:
- Facilitates communication between:
- Creo Parametric
- Windchill server
- Composed of:
- Client-side workspace (local cache)
- Server-side workspace
- Client-side workspace (local cache) – Server-side workspace



Workspace Layout

- All workspaces have a common layout in the embedded browser window

The screenshot shows the 'Object List' table in the workspace. The table has columns for Number, File Name, Actions, Version, Last Modified, and State. The data includes various parts like RECOIL_400, RECOIL_COVER_400, and SCREW_SKT_M4XPT7X15.PRT, all in version A.1 and In Work state.

Number	File Name	Actions	Version	Last Modified	State
RECOIL_400	recoil_400.asm	[Actions]	A.1	2011-03-01 10:16 EST	In Work
RECOIL_COVER_400	recoil_cover_400.prt	[Actions]	A.1	2011-03-01 10:16 EST	In Work
RECOIL_GRIP_400	recoil_grip_400.prt	[Actions]	A.1	2011-03-01 10:16 EST	In Work
RECOIL_PULLEY_400	recoil_pulley_400.prt	[Actions]	A.1	2011-03-01 10:38 EST	In Work
SCREW_SKT_M4XPT7X15.PRT	screw_skt_m4xpt7x15.prt	[Actions]	A.1	2011-03-01 10:16 EST	In Work
SCREW_SKT_METRIC.PRT	screw_skt_metric.prt	[Actions]	A.1	2011-03-01 10:16 EST	In Work
STANDARD_BIT.PRT	standard_bit.prt	[Actions]	A.1	2011-03-01 10:16 EST	In Work

All workspaces have the same fundamental layout. Above the Object List table is the storage context name, workspace name, and Pick an Action drop-down menu.

Changing the Active Workspace

- The Active workspace concept only applies when working within Creo Parametric
- It can only be changed when working from within Creo Parametric in these locations:
 - My Workspaces Table
 - Home Page
 - Product
 - Library
 - Server Management Window
 - Workspace Pick an Action drop-down menu
 - Folder Browser

The screenshot shows the 'My Workspaces' table. It lists various workspaces with their names, contexts, and actions (Edit, Delete, etc.).

Name	Context
Carb 400	Drill - 400 Series
Crankshaft 400	Drill - 400 Series
Dowel Pin	Standard Parts
Drill 400	Drill - 400 Series
Drill 510	Drill - 510 Series
Drill 510 Chuck	Drill - 510 Series
Drill Bit	Standard Parts

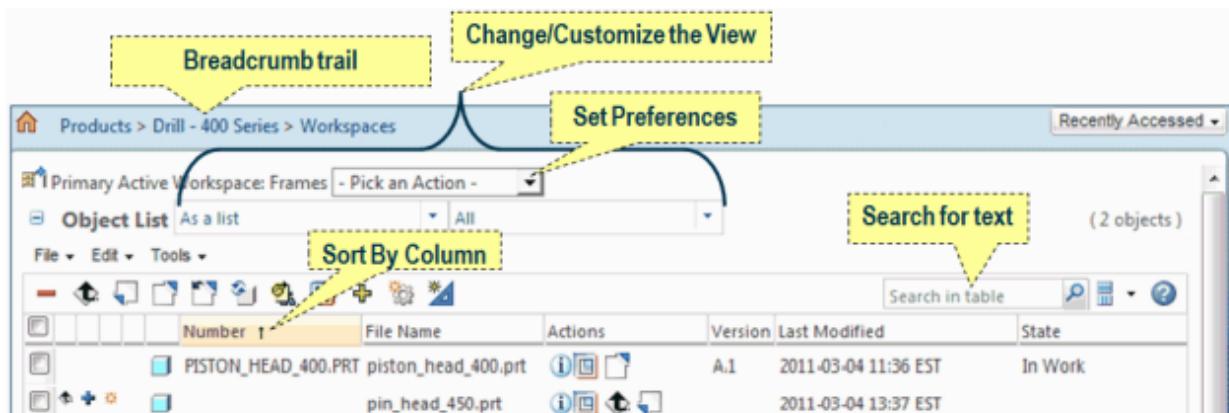
- The active workspace status is only applicable when you are working in an environment in which Creo Parametric is connected to Windchill. Therefore, setting or changing the active workspace can only be conducted from Creo Parametric.

Working with Workspaces

You can configure your workspaces to fit your needs.

- Search in table utility to locate information
- Change or customize workspace table views
- Sort columns
- Set preferences

The screenshot shows the Windchill Workspaces interface. A breadcrumb trail at the top left indicates the path: Products > Drill - 400 Series > Workspaces. To the right of the breadcrumb trail are buttons for 'Change/Customize the View' and 'Set Preferences'. Below the toolbar is an 'Object List' table with columns: Number, File Name, Actions, Version, Last Modified, and State. The table contains two rows of data: 'PISTON_HEAD_400.PRT' and 'pin_head_450.prt'. A 'Search for text' field is located on the right side of the table. A 'Sort By Column' button is highlighted with a yellow callout. The interface is branded with 'DesignTech Edu' and 'Empirical Learning'.



Deleting Workspaces

Workspaces should be deleted once they are no longer required.

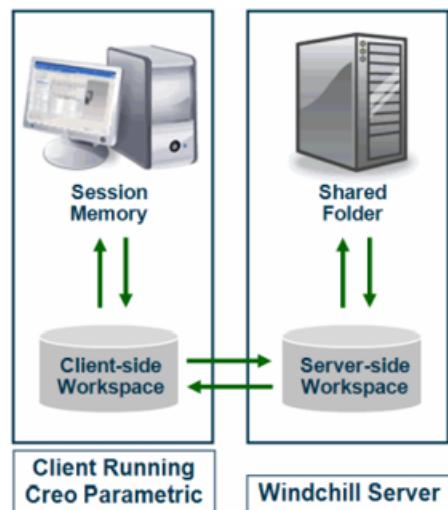
- Workspace Delete
- The active workspace cannot be deleted
- Checked out objects cause an overridable conflict

The screenshot shows a list of workspaces on the left and a 'Message from webpage' dialog box on the right. The dialog box asks, 'Are you sure you want to delete the selected workspaces? All the data which is not checked in to the server will be lost.' It has 'OK' and 'Cancel' buttons. The list of workspaces includes 'Carb 400', 'Crankshaft 400', 'Dowel Pin', and 'Drill 400', all associated with 'Drill - 400 Series'.

- To maintain a manageable list of workspaces, it is recommended to delete a workspace once you have finished using it. The workspace must be inactive (not the current active workspace).

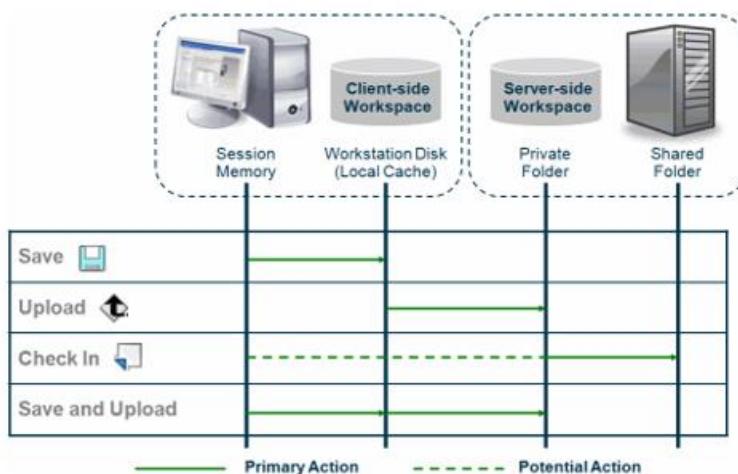
Understanding the Workspace Architecture

- The workspace utilizes both local and server-side storage locations.
- The workspace is composed of:
 - Client-side workspace
 - Server-side workspace
- Benefits:
 - Enables you to selectively download only the files you need.
 - Enables upload and download actions to occur asynchronously.
 - Server-side workspace provides file protection.
- Uploading files to the server-side workspace provides access to your work from other workstations



Understanding the Save, Upload, and Check In Actions

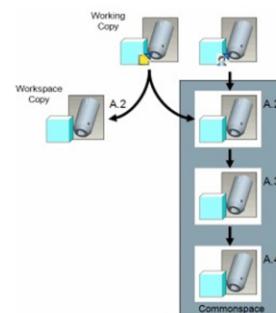
- The four options for saving CAD data differ as to where the design file ultimately reside





Understanding Workspace Objects

- Workspace objects represent copies of the actual objects stored and managed in the Windchill common space.
- Workspace objects:
 - When checked out, workspace objects are working copies.
 - If not removed during check in:
 - Working copies become workspace copies of common space objects.
 - Design files remain in the local workspace cache.
 - Workspace copies and design files are not automatically updated with changes to common space data.
- It is important to understand the differences between using Creo Parametric's embedded Web browser and a standalone Web browser, such as Firefox or Internet Explorer, to access information and perform Windchill functions.
- The Creo Parametric embedded Web browser provides full data management functionality because it can access and reference your Creo Parametric session, your local workspace cache, the server-side workspace, and the common space.



Specialization program in PLM- Windchill

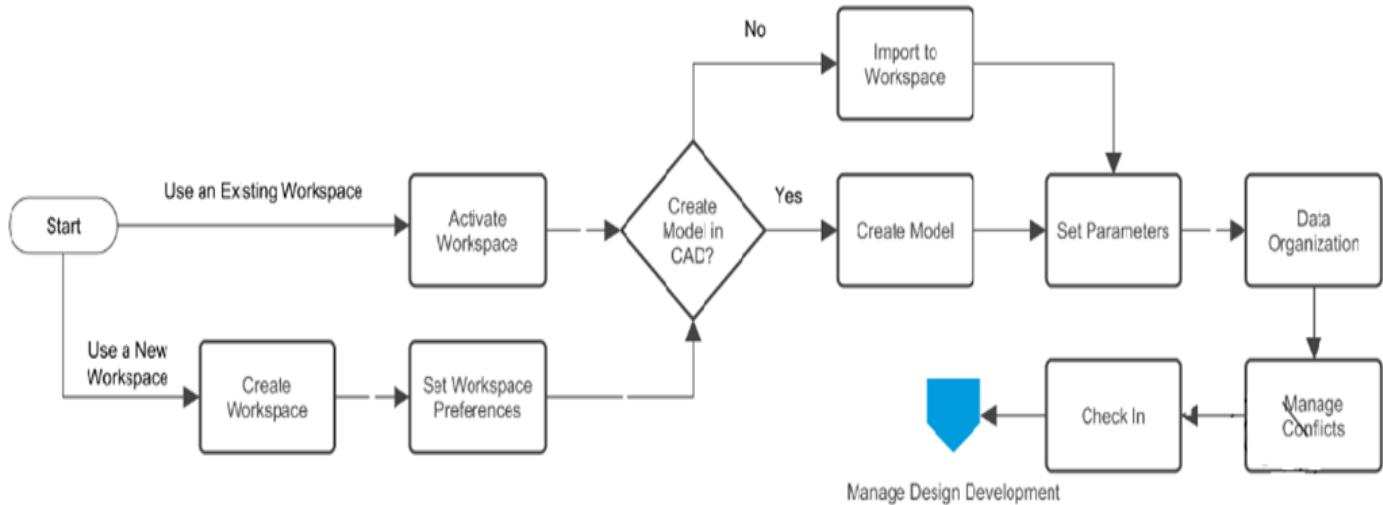
Day 5 Manage Design Data

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- The goal of the Manage Design Data step is to enable the creation and management of CAD data from various CAD authoring tools integrated with the Windchill capabilities.

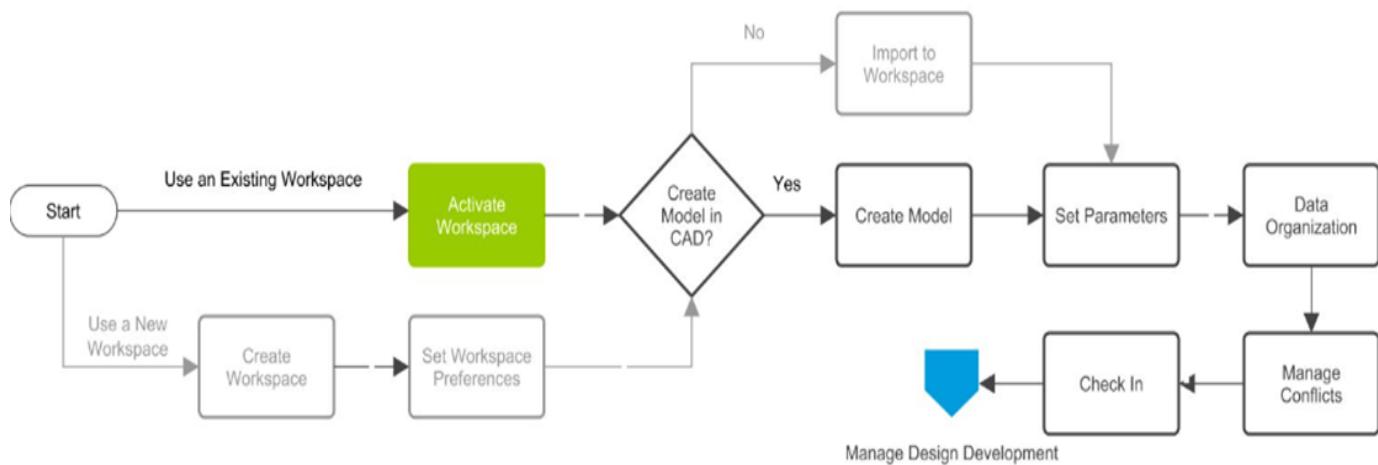


The goal of the Manage Design Data step is to enable the creation and management of CAD data from various CAD authoring tools integrated with the Windchill capabilities. Designer is the primary role for this segment of the process. The process involves the following steps:

- A Designer activates a workspace.
- CAD models are created and predefined attributes have their values specified.
- The specific Windchill location for the new models is specified.
- If any data conflicts arise, they are resolved.
- The check in is completed and the models now reside in Windchill.

Activate Workspace

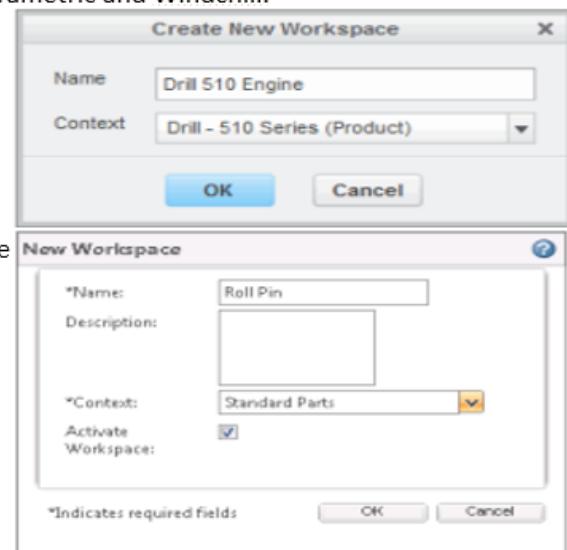
The active workspace status is only applicable when you are working in an environment in which Creo Parametric is connected to Windchill.



The active workspace status is only applicable when you are working in an environment in which Creo Parametric is connected to Windchill.

Creating Workspace

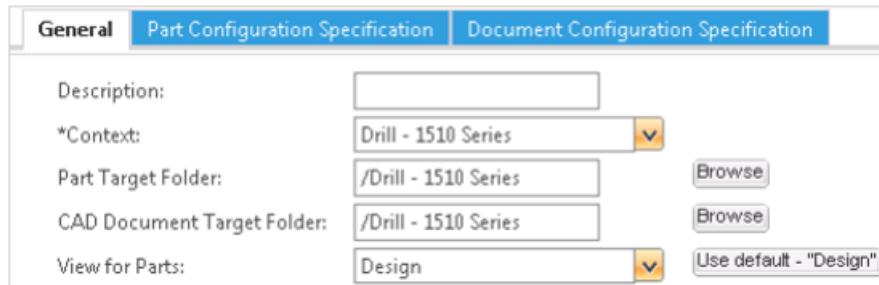
- New workspaces can be created in both Creo Parametric and Windchill.
 - Creo Parametric:
 - Created from Server Management.





Setting the General Workspace Preferences

- The General preferences enable you to specify common space target folders for new objects.
 - General preferences:
 - Description
 - Context
 - Part Target Folder
 - CAD Document Target Folder
 - View for Part



You are able to configure your workspaces by setting workspace preferences. To access the general preferences for a workspace, select Edit Preferences from the workspace Pick an Action drop-down menu. This opens the Edit Workspace Options page, which has three tabs: General, Part Configuration Specification, and Document Configuration Specification.

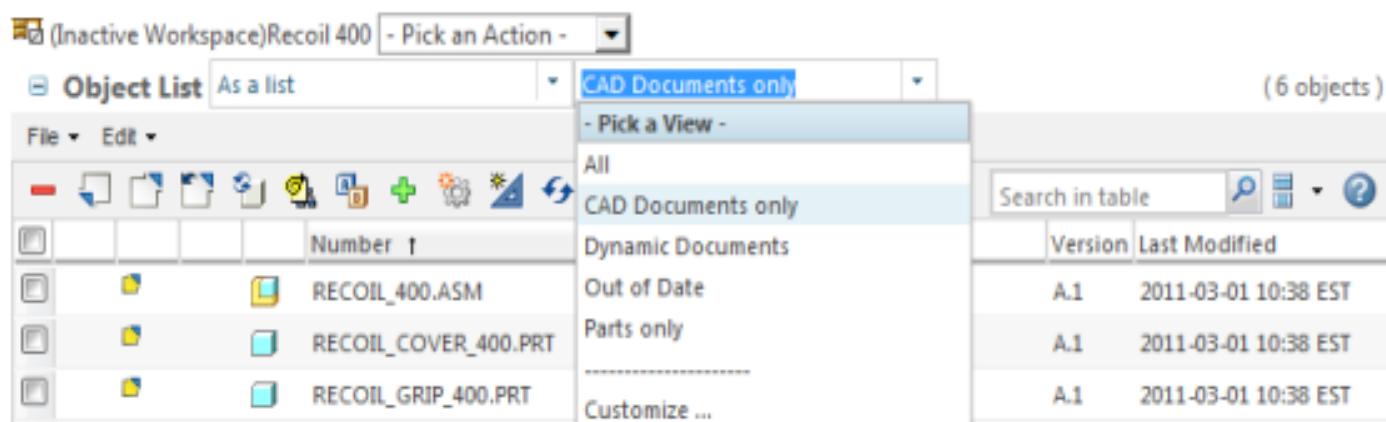
The General tab enables you to set several preferences:

- The Description field enables you to update the description for the workspace.
- The Context field enables you to change the context associated with the workspace.
- The Part Target Folder field enables you to specify the check in location for new WTParts within the folder structure of the associated context.



Changing the Workspace Table view

- The workspace table view can be modified to fit your needs.
- Modify the table view by clicking the view drop-down menu.



- The View drop-down menu enables you to filter objects displayed in the Object List table by using predefined table views. The right drop-down menu enables you to select from a set of table views already defined by you, other users, or administrators. The left drop-down list enables you to further specify which objects are displayed for the view you selected.

Customizing Workspace Table Views

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- Workspace views can be customized to fit your needs.
 - Manage views: New/Customize View Wizard:
 - Show/Hide Set Name
 - Save As Choose Object Type
 - Set Active Set Filters
 - Table View Set Column Display
 - Edit Set Sorting
 - Delete

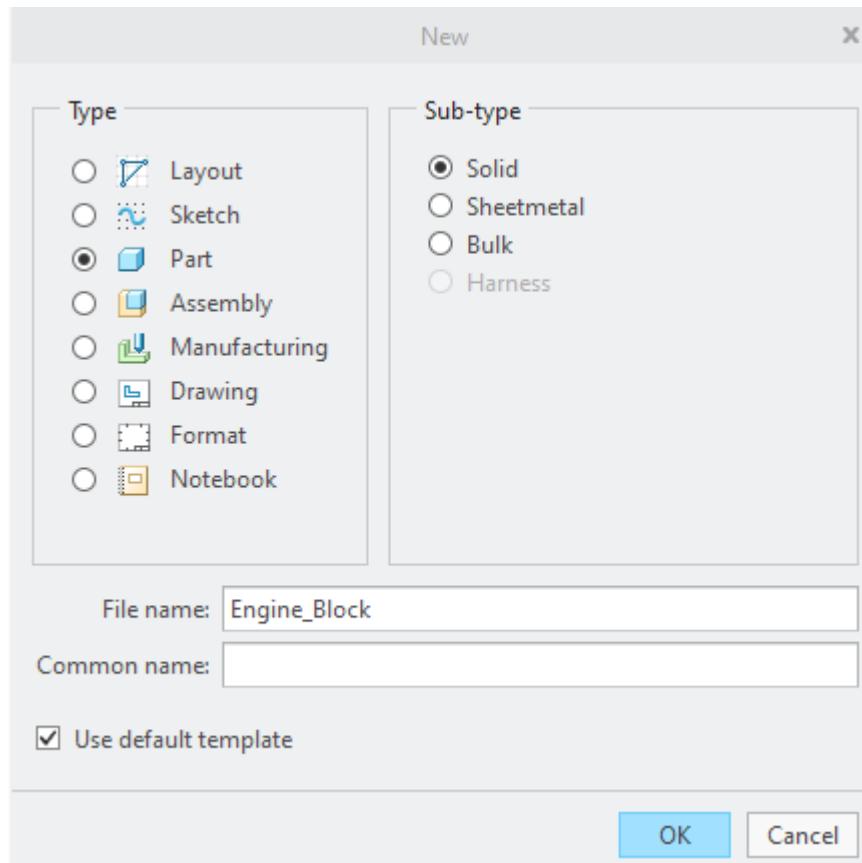
Customize View List: Workspace Listing			
Views:			
	Name	Actions	Show
<input type="checkbox"/>	All		<input checked="" type="checkbox"/>
<input type="checkbox"/>	CAD Documents only		<input checked="" type="checkbox"/>
<input type="checkbox"/>	Dynamic Documents		<input checked="" type="checkbox"/>
<input type="checkbox"/>	Out of Date		<input checked="" type="checkbox"/>
<input type="checkbox"/>	Parts only		<input checked="" type="checkbox"/>

- The workspace Object List table displays objects in rows with information about each object displayed in columns. The number, content, and order of the columns and rows can help or hinder your ability to work with large data sets.
- Windchill provides default workspace table views and your administrator may have defined custom table views for your organization.



Naming a New Creo Parametric Model

- Two naming fields are utilized when creating new objects from within Creo Parametric.
 - Creo Parametric
 - Name (Filename)
 - Common Name (Parameter PTC_COMMON_NAME)



- Values for CAD document naming and numbering and possibly WTPart naming and numbering are derived from Creo Parametric naming elements and Windchill system configuration.
- When you create a new object in Creo Parametric, you are presented with two fields in the New dialog box. When stored locally, Name identifies the file name of the Creo Parametric file; Common Name is stored in the designated parameter PTC_COMMON_NAME.



Naming and Numbering Design Documents

- Creo Parametric filenames can be used as a basis for the document naming schemes.
- In Windchill, the attributes Name and Number provide an identity for the CAD document and WTPart, while File Name identifies the Creo Parametric design file managed as primary content.

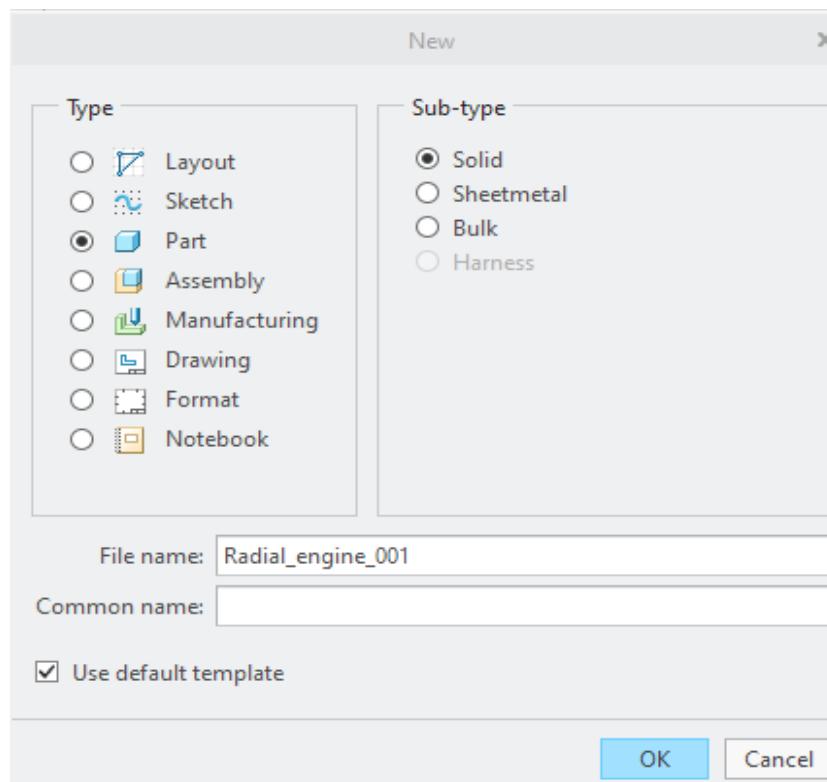
The screenshot shows the Windchill interface with the 'Structure' tab selected. The title bar indicates 'Actions ▾ Part - 0000000749, 0000002818.asm, A.1 (Design)'. The main area displays a table with columns: Identity, Number, Name, and Object T... (partially visible). The table lists several CAD documents and parts, with the row for '0000002818.asm, A.3' highlighted. The 'Number' column contains values like 000000749 and 0000002818.ASM. The 'Name' column contains values like 0000002818.asm and 0000002818.ASM. The 'Object T...' column contains values like Part and CAD Doc... . The 'Identity' column shows icons for each item.

Identity	Number	Name	Object T...
0000000749, 0000002818.asm, A.1 (Design)	000000749	0000002818.asm	Part
0000002818.asm, A.3	0000002818.ASM	0000002818.asm	CAD Doc...
0000002821.drw, A.1			CAD Doc...
0000000748, 0000002819.prt, A.1 (Design)	0000000748	0000002819.prt	Part
0000002819.prt, A.3	0000002819.PRT	0000002819.prt	CAD Doc...
0000002820.drw, A.1			CAD Doc...

- In Windchill, the attributes Name and Number provide an identity for the CAD document and WTPart, while File Name identifies the Creo Parametric design file managed as primary content.

Creating CAD Models

- There are many methods of creating new Creo Parametric CAD models.
- The primary method for creating new CAD models:
 - Use the New icon from the Creo Quick Access Toolbar (or use the Creo Parametric File > New menu selection)

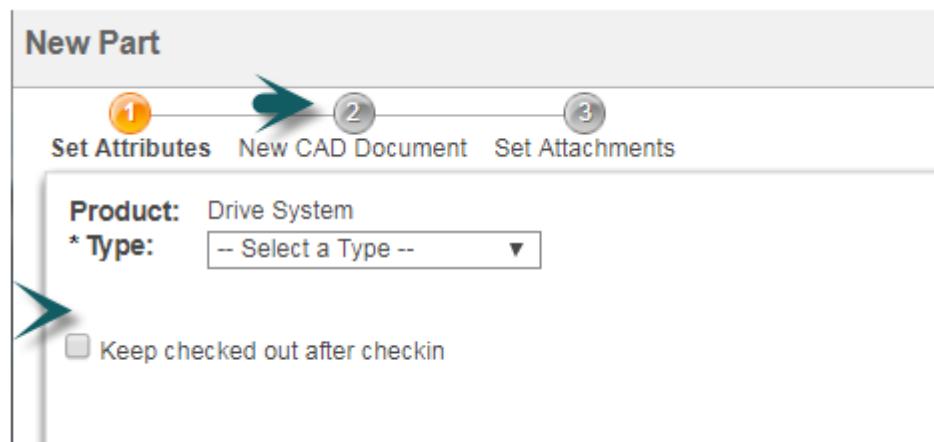


New Creo Parametric Model type



Initiating New Designs Using New CAD Document

New CAD Documents are created directly in Windchill using CAD Document templates.



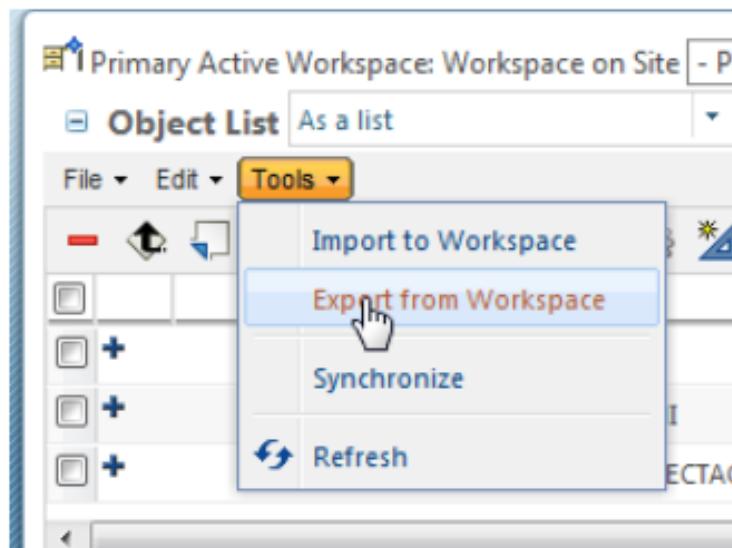
- Typically, you may start new models by selecting New from the CAD application's main toolbar. When saving the model to a workspace, this generates the design file and enables you to define its geometry before the CAD document is created.
- Conversely, the New CAD Document action in the workspace enables you to create and define attributes for a CAD document before you begin modeling the design file. Selecting the New CAD Document actions opens the New CAD Document Wizard.
- In the New CAD Document wizard, you identify general information about the CAD document. You specify items such as the Authoring Application, CAD document type (if more than one exists), Template Name, Name, File Name, storage location, description, and new revision. The file name must include an extension that is valid for the selected CAD document type.
- If your site has defined additional CAD Document attributes, then fields and any default values for those attributes appear in the Attributes section of the wizard. For each attribute, you accept the default value or specify a new value.
- Optionally, you may create an Owner associated WTPart at the same time by selecting the Create and Associate Part check box. When creating a new WTPart using the New Part action, you optionally have the ability to create an Owner associated CAD document.

Importing CAD Data to Workspace

- You can import design data from the file system into the active workspace.
- Importing CAD design data:
 - This function loads CAD data into the active workspace.
- Status columns alert you if an object already exists in the common space or workspace. Import to Workspace Wizard:
 - 1. Select files to import.
 - Add / Remove
 - 2. Specify Options.
 - Add as New.
 - Add as Modified.
 - Reuse existing workspace
- The workspace import functionality enables you to move CAD objects between your local disk and active workspace without explicitly retrieving them into your CAD session. When working with import, you must be aware of a few considerations:

Exporting CAD Data From Workspace

- You can export multiple objects from a workspace in a single operation.
- Export CAD documents:
 - Set Target Directory.
 - Reuse objects that already exist in the target location.



Export CAD Documents

Specialization program in PLM - Windchill

Day 6 Manage CAD Data

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Using Library Parts In Assemblies

- Using library parts improves productivity and reduces cost.
- Library parts:
 - Reduce part proliferation by leveraging existing approved components.
 - Improves product time-to-market.
 - Reduces cost.
 - Are stored and controlled within component libraries.
 - Enables specifying access.
 - Eases the process of locating components for product designs.
- Library parts offer several advantages. Company-approved libraries help enterprises by reducing part proliferation (wherein the same part may be redundantly assigned different part numbers by different engineers working independently).



Locking CAD Documents In The Workspace

Lock CAD documents in the workspace to prevent them from unintentional modification.

- Locking a CAD document in the workspace:
 - Sets the associated content to a read-only status in the local cache.
 - Is used to:
 - Reduce the number of times that the Conflicts dialog box is presented for objects that are not intended to be modified.
 - Provide better visibility into changes made implicitly by the CAD application.
 - Prevent the object from being modified
- Locking a CAD document in the workspace sets a read-only status on the content stored in the local workspace cache. To lock a CAD document, select the object in the workspace and click File > Lock. It is also possible to lock CAD documents when they are being added to the workspace, by selecting the Automatically lock all non-checked out objects added to the workspace option in the Add to Workspace wizard Advanced tab.

Object List			As a list	All
File ▾ Edit ▾ Tools ▾				
			Number ↑	File Name
			CHUCK_510.PRT	chuck_510.prt
			CHUCK_COLLAR_510....	chuck_collar_510.prt
			DRILL_CHUCK_510.A...	drill_chuck_510.asm

- The workspace also provides the capability to modify CAD document attributes. If any of the CAD document attributes are mapped to Creo Parametric parameters, the parameter values are updated without requiring Creo Parametric intervention.
- The Edit Attributes page enables you to check out and individually set attribute values for each object.

Cancelling Modifications

You can undo modifications by canceling a checkout.

- Cancel Check Out:
 - Deletes the working copy.
 - Returns the shared copy to its status prior to checkout.
 - Does not create a new iteration

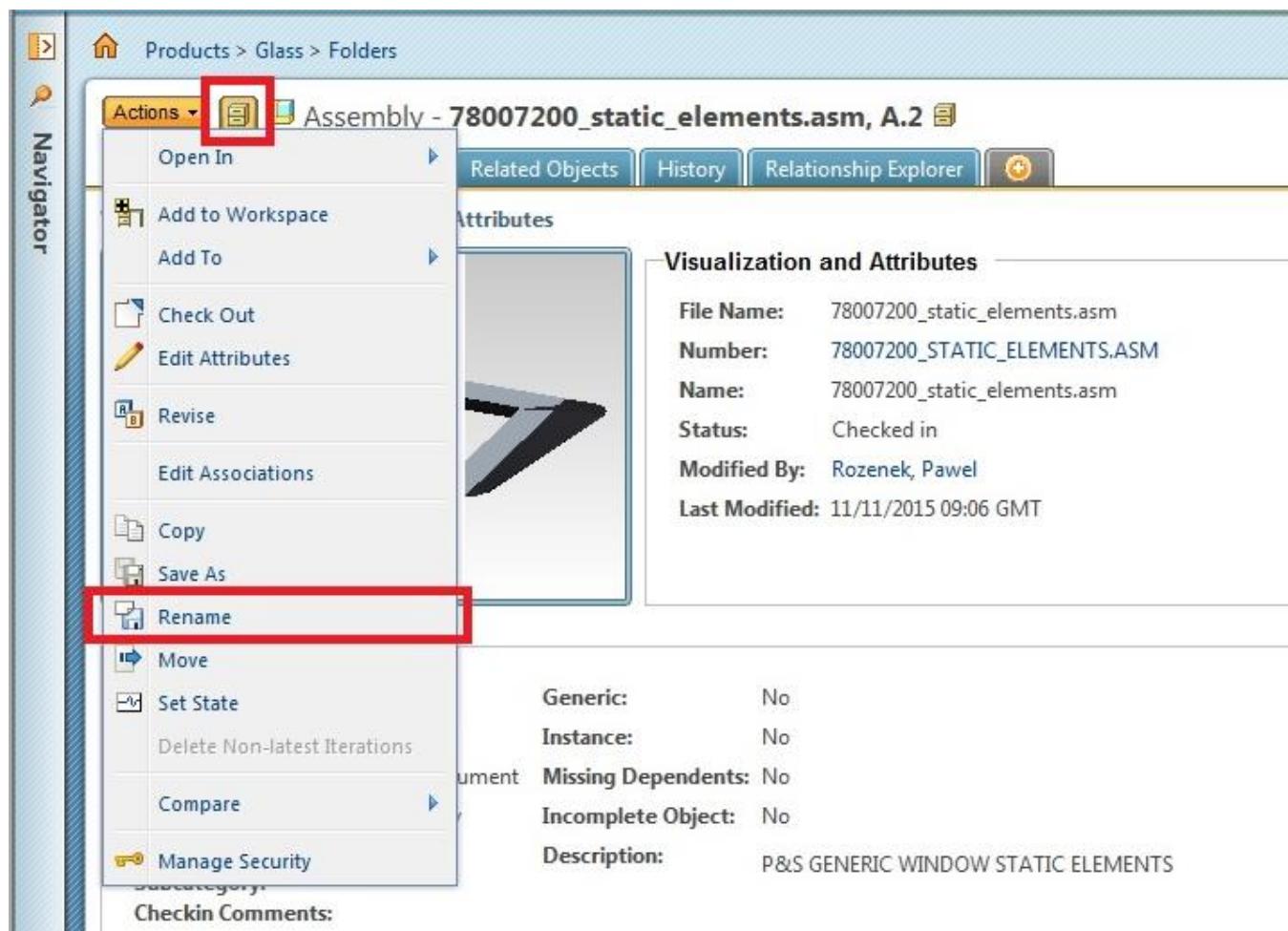
Undo Check Out

Current Settings: Required dependents						
Object List		As a List	Default	Dependents ▾		
		Name ↑	File Name	Version	Actions	Collection Rule
		chuck_510.prt	chuck_510.prt	A.2	(i)	Dependent
		chuck_collar_510.prt	chuck_collar_510.prt	A.1	(i)	Dependent
		drill_chuck_510.asm	drill_chuck_510.asm	A.1	(i)	Initially Selected



The Rename capability enables you to rename new CAD documents in the workspace and existing CAD documents in the commonspace.

- Rename:
- Enables you to change:
 - CAD document Name and Number.
 - CAD File Name.
- Limited by your implementation's naming and numbering policy and uniqueness.
- Use the Set New Name utility to apply a pattern



- The Rename feature in Windchill enables you to reidentify objects and still maintain modeling dependencies. The Rename action enables you to change the Number, Name, and File Name attributes. Rename requires the Change Identity access permission.



Renaming WT Parts

The Rename action enables you to rename new WTParts in the workspace and existing WTParts in the commonspace.

- Rename:
- Enables you to change:
- WTPart Name and Number
- Limited by your implementation's naming and numbering policy and uniqueness
- Use the Set New Name utility to apply a naming pattern
- Rename enables you to change the Number and Name attributes of WTParts. Rename requires the Change Identity access permission.
- The options available for renaming may be limited by your implementation's naming and numbering policies. The Rename page includes object collectors that enable you to include other related objects.



Moving Design Documents

The Move capability enables you to move design documents in common space.

- Move
- Determine which iterations are moved.
- Limitations:
 - Versioning scheme
 - Team template
 - Life cycle
- The Move action enables you to move WTParts, CAD documents, and other objects between contexts and folders. You can choose to move all versions or only the versions displayed in the Move window.
- The system retains all the dependencies during the move. The Set Location window enables you to set the destination context and folder. There are some limitations associated with the Move feature.

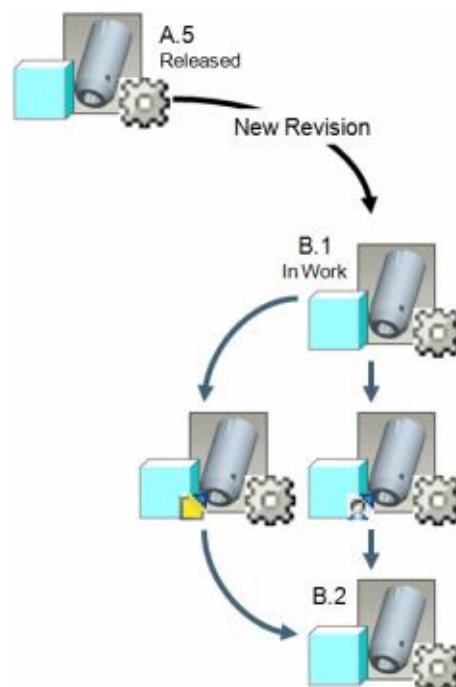


Windchill applies business rules and administrative settings as part of the revision process.

- Revise:
 - Creates a new revision of the CAD document and/or WT Part.
 - Creates a new revision of the CAD document.
 - Revision number or letter is incremented.
 - The iteration value is reset to the beginning of the sequence.
 - The new version is set to a state:
 - Specified by the administrator.
 - That enables modification through checkout and checkin.

Your business rules may preclude you from altering CAD documents past a certain development state. In this situation, you must use a Revise action to generate a new object version before making any changes. As part of the revision process, Windchill applies business rules and administrative settings in generating the new object version.

- These include, but are not limited to:
 - Incrementing the revision number or letter to the next value in the revision sequence.
 - Resetting the iteration number or letter to the beginning of its sequence.
 - Setting the life cycle state. Typically, the administrative rules designate a state with access rules that enable modification using the check in and check out operations
 - There are multiple locations in Windchill where you can manually initiate a Revise action.

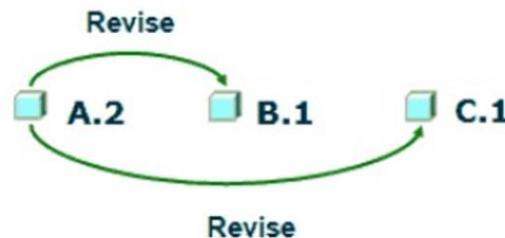




Revising From The Non-latest Version

The Revise action is configurable to enable you to revise from a nonlatest version of an object

- Configured by an Administrative Preference:
- Revise the earlier version by browsing to it using version history



- If this feature is enabled by an administrator, you can revise the latest iteration of a prior revision and use it as a starting point for your incremental work.
- You can browse to the earlier version by clicking the History tab from the object's common space Information page, clicking the View information icon of the version to be revised, and from the Actions drop-down menu of this prior version, selecting the Revise action. Your account must have the revise authorization to perform this or any revise action.

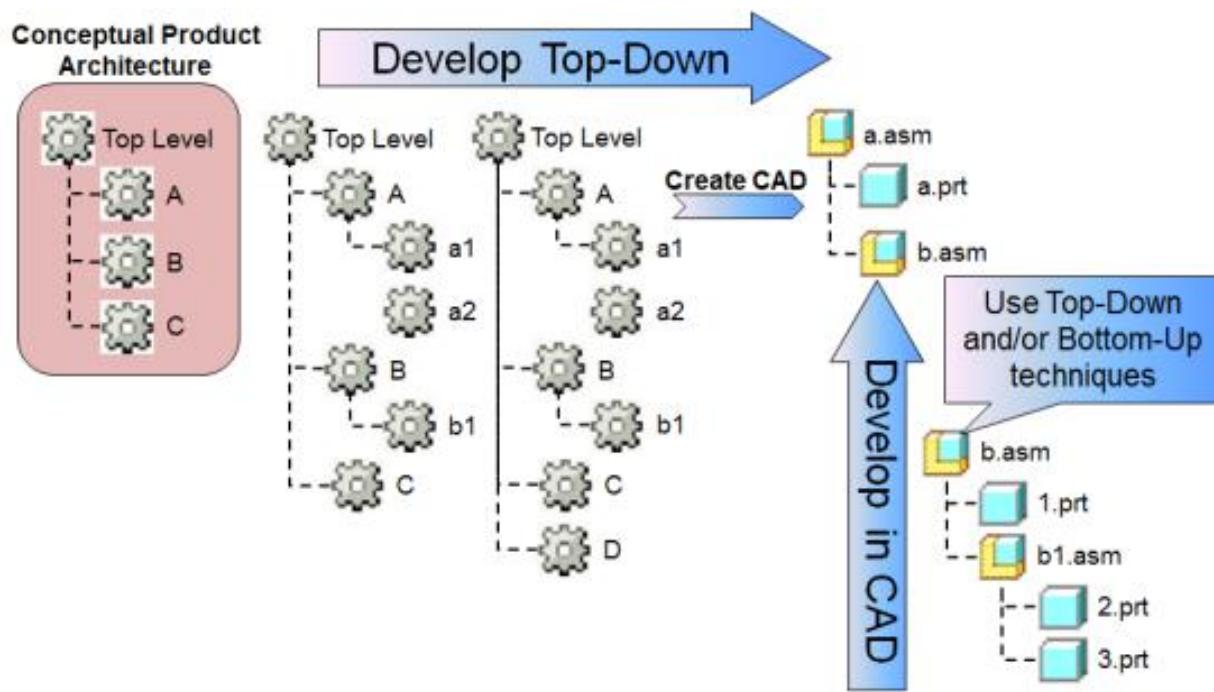


Revise and Check Out Action

- The Revise and Check Out action enables you to check out and revise a CAD document all in one step.
- Available in multiple places:
 - Creo Parametric File menu
 - Model tree
 - Conflicts dialog box
- The Revise and Check Out action is an additional option available in the Creo Parametric File menu, from the model tree, and in the Conflicts dialog box. It enables you to both revise and check out selected objects in a single operation (not available for objects in a project context). Your account must have the proper authorization to perform this action.

Specialization program in PLM- Windchill

Day 7 Working with CAD Data



Enterprise Top –Down design

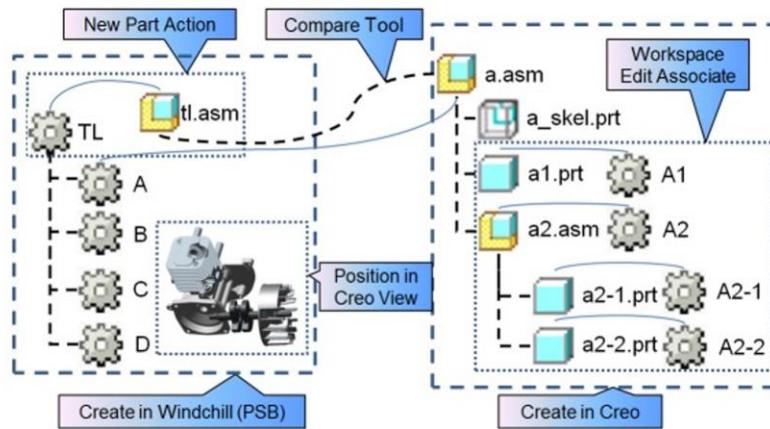
The enterprise top-down design tools enable you to build a CAD structure from a WTPart structure.

- Define a conceptual product architecture and transform it into a detailed product structure.
- Create related CAD content and CAD relationships based on product structure decisions.
- Enable visual comparison of WTPart and CAD structures.
 - Create CAD documents from WTParts.
 - Enable editing of existing product structures.
- Enable the reverse build to create the CAD model structure from the WTPart structure.
 - Build selectively.
 - Pass attributes from WTPart to CAD documents.



Enterprises TOP-DOWN design process

You can use any or all of the Windchill enterprise top-down design tools.



Top-Down design process

You can use any or all of the Windchill enterprise top-down design tools.



Using Compare

Using the Compare tool, you can compare CAD and WTPart structures.

Compare features:

- Side-by-side comparison of structures.
- Highlight and Navigate differences.
- Additional cross-highlighting reporting in lower pane.
- Lightweight editing.
- Bottom-up and top-down design build support.

Windchill has a built-in structure comparison tool. It enables you to make side-by-side comparisons of WTPart and CAD model structures. Using the tool, you are guided by status icons that indicate whether and where there are (child) differences in the two structures.

For example, if comparing a WTPart product structure to a CAD model structure, if there are differences, icons highlight these differences, and you are provided tools with which you can choose whether or not to reconcile these differences. If you choose to reconcile, the Build Status column indicates the intent to build or not build a CAD document into the CAD model structure the next time you initiate a build of the CAD structure from the WTPart structure.

The Compare tool has an efficient layout.

The Compare tool layout is composed of the following:

- A left-side pane that contains the base CAD or WTPart structure.
- A right-side pane that contains the CAD or WTPart structure that is being compared. If the base is a CAD structure then the right-side pane must be a WTPart structure.
- A toolbar that contains icons for navigating to differences in the structures as well as icons for setting the build status and building structure in the right-side pane
- A central column that displays differences and statuses in the structures.
- A lower pane (which can be hidden) that displays additional information about the structures.
- Tools for searching the structures as well as configuring the table display.



Understanding PTC-Creo Unite

PTC Creo Unite enables you to create and maintain design intent across CAD platforms.

Data is managed associatively in a multi-CAD enterprise:

- Easily reuse legacy data.
- Enable a “design anywhere, build anywhere” strategy.
- Reduce the need to create and manage neutral formats.

Format	Import	Open	Update	Save As
CATIA V4	✓	✓	*	*
CATIA V5	✓	✓	*	*
NX	✓	✓	*	*
SolidWorks	✓	✓	*	*
CADD5	✓		✓	✓
Creo Elements/Direct	✓		✓	
Step	✓			✓
Autodesk Inventor	✓			
Rhinoceros	✓			
SolidEdge	✓			✓
AutoCAD	✓			✓

✓ Supported in the Creo Parametric base license

* Requires additional license extensions

PTC Creo unite capability



Understanding PTC-Creo Design Exploration Extension(DEX)

PTC Creo DEX enables you to explore and evaluate multiple design ideas before committing to final decisions.

- Explore design changes safely without risking original designs or committing to any change.
- Simultaneously develop different ideas and evaluate all options before making decisions.
- Eliminate manual data duplication and session clean-up for loading and reloading different versions.

The PTC Creo Design Exploration Extension (DEX) enables users to quickly and effectively evaluate design alternatives and determine the most effective modeling process. Furthermore, if you explore a design path but determine it is not the most effective, you can quickly revert back to an earlier saved “checkpoint” without having to completely recreate the model.

For Creo Parametric 3.0 M030 and later datecodes, the TMU file was introduced to replace the TMZ. Windchill recognizes the TMU as a primary CAD object and establishes the required dependents. You download the TMU with dependents using the As Stored configuration.



Introduction to workspace frames

Workspace frames provide “undo” capability by copying a prior workspace frame and making the copy the current active frame.

Workspace frames benefits:

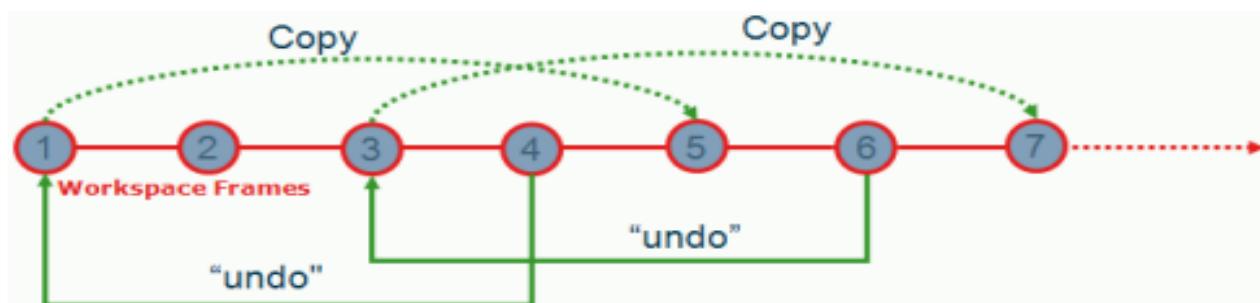
- Provides “undo/redo” capability for workspace actions.
- Data is not lost by undoing actions. Configuring Workspace Frames:
- Controlled by the Workspace Frame Stack Size preference:
 - Set to zero to turn off.
 - Set to an integer to turn on (Restart Creo Parametric)

A Workspace Frame is a snapshot of the existing design configuration in a given workspace at a given point in time.

Actions that are initiated from the workspace and affect the workspace cache state for a given object (modified or new), or that add, remove, or modify an object, are recorded as frames.

Frames functionality is available only in the Creo Parametric browser with access to cache. By default this functionality is turned off, but you can enable it by setting the preference Workgroup Manager Client > Workspace Frame Stack Size to a positive integer that represents the number of recorded frames to retain in the cache.

After the preference is set to enable frames, the frame functionality is available in subsequent Creo Parametric sessions.



Accessing workspace frames

You access workspace frames using the Event Management console.

Access Workspace Frames by:

- Pick an Action drop-down menu (previous frame only)
- Event Management console.

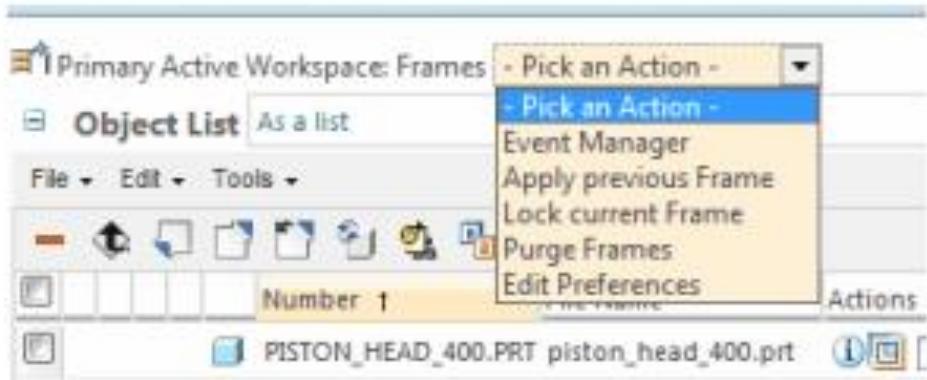
Once recorded, you can access frames of non-current states of the workspace using the Event Management console. In the Event Management console listing, you can select a frame and select the Apply this event action to restore the workspace to the state of the selected frame.

There are important details regarding the use of workspace frames.

Important properties of Workspace Frames:

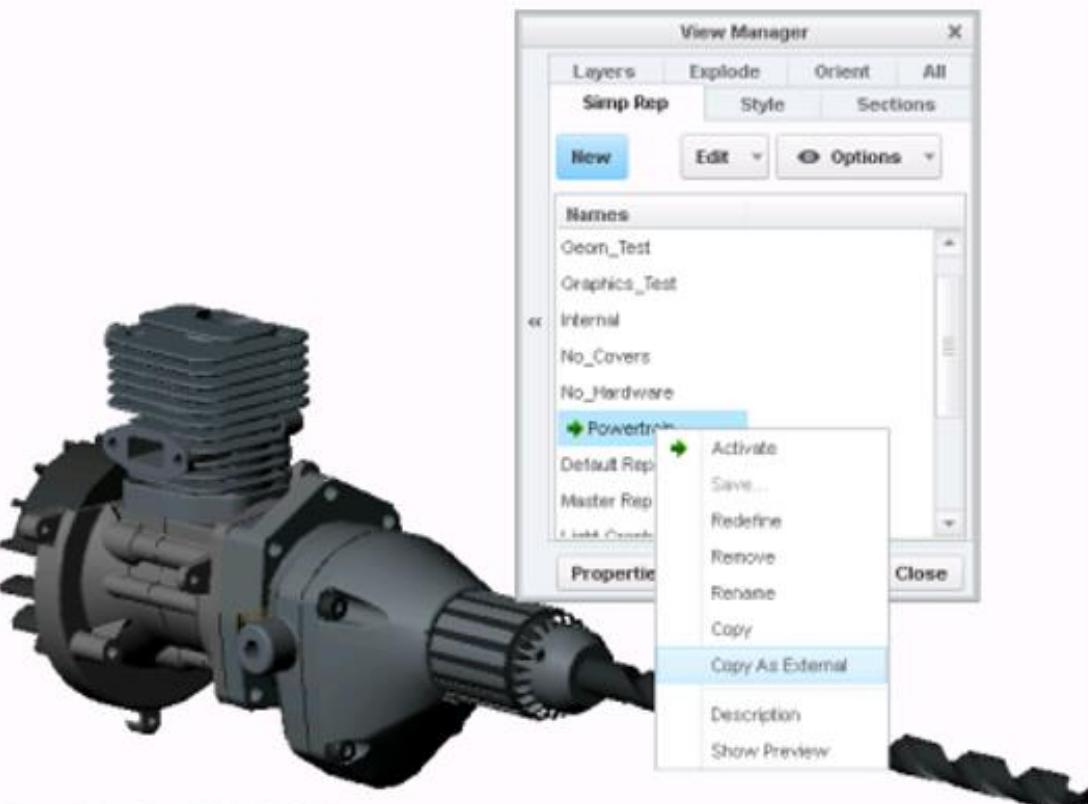
- Each frame contains:
- Action name
- Date/time of recording
- Information
- Cannot purge locked frames.
- Oldest non-locked frame is removed when the number of frames exceeds the stack size.

- Cache cleanup never removes frames.
- Restoring frames may initiate a Creo Parametric action.
- Some workspace server-side actions may create frames.
- No frame support for offline workspaces.
- Frames are preserved across Creo Parametric sessions.
- Frames are only created for successful actions.



Using external simplified representations

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You can search for external simplified representations in Windchill.

External simplified representations:

- Searchable object within Windchill.
- Can be organized into folders.
- Can be retrieved/opened by any method.

Once you have created a simplified representation, you may want to externalize the simplified representation and check it into Windchill. Externalizing the representation will create a searchable object in Windchill.

which you can then use to open the simplified representation directly by any method and only the required components are downloaded to the workspace cache.

To create an external simplified representation, in the View Manager, right-click an existing active simplified representation and select Copy As External. This will create the external representation object in your workspace, which you then check in to Windchill.



Managing secondary content files with windchill

You can manage and attach secondary content to CAD documents.

Secondary file:

- Attached to the CAD document.



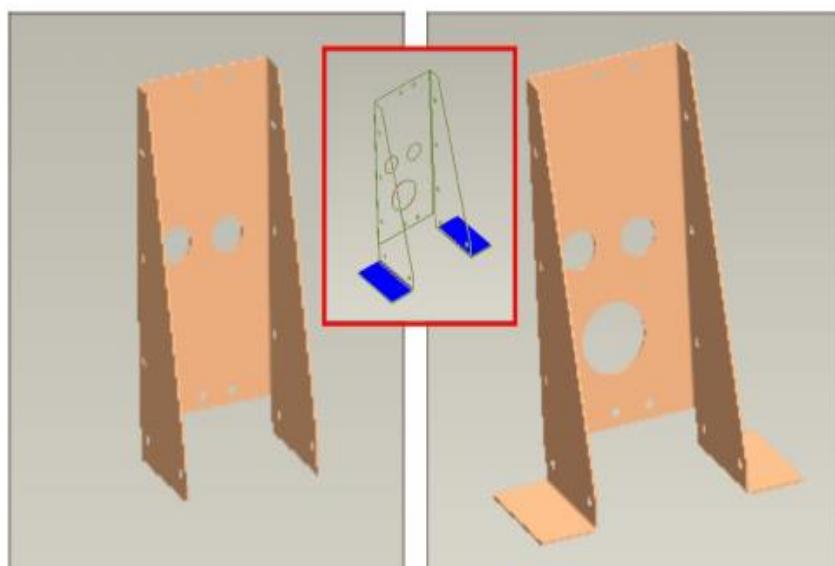
Using Creo parametric to compare PDM model version

You can use Creo Parametric to compare a model's geometry to another version of the model stored in Windchill.

Comparing versions using Creo Parametric:

- Open first version in Creo Parametric.
- Select the Tools > Compare Part functionality.
- Browse to other version within the commonspace.
–> All versions.

You can use Creo Parametric to compare two different versions of the same model for feature or geometry differences. Because Windchill stores all versions of a CAD document, it may be useful to perform comparisons of multiple versions of the same part. An example of this may be a detailed Creo Parametric design review prior to approving a CN for a changed part and revising it from revision C to revision D. It may also be useful if you need to refer back to an old change that was made some time ago, such as for warranty or safety purposes, and to observe exactly what modifications were made between versions. You can perform this comparison by accessing multiple versions of the item from within Windchill.

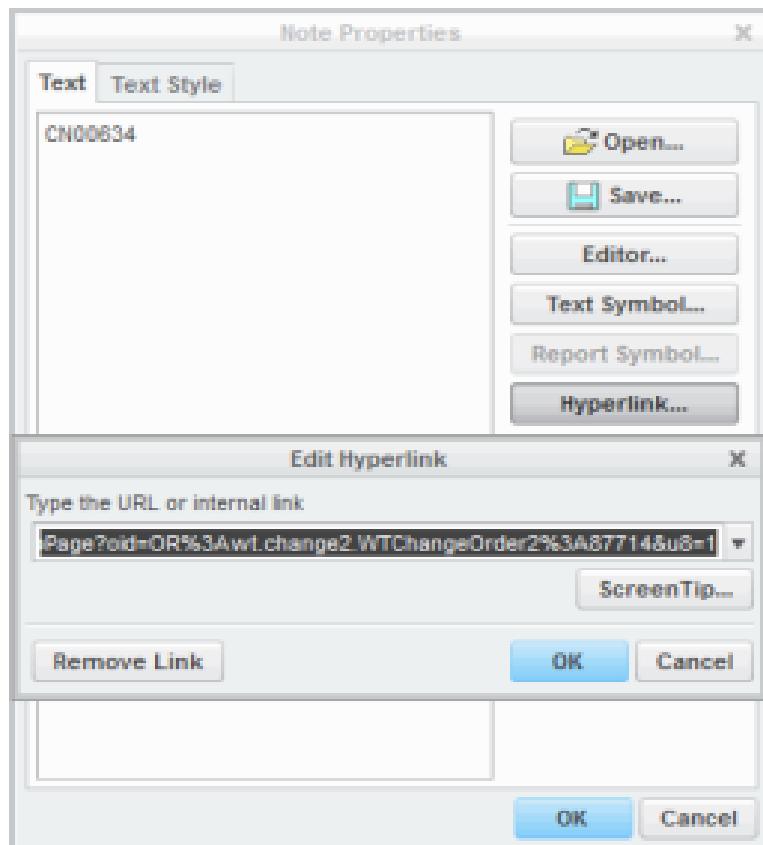




Hyperlinks within a drawing

Drawing notes can contain hyperlinks to additional design information.

- Link a drawing note to the Windchill Change Notice (CN):
 - Create drawing note.
 - Create hyperlink to a Windchill CN



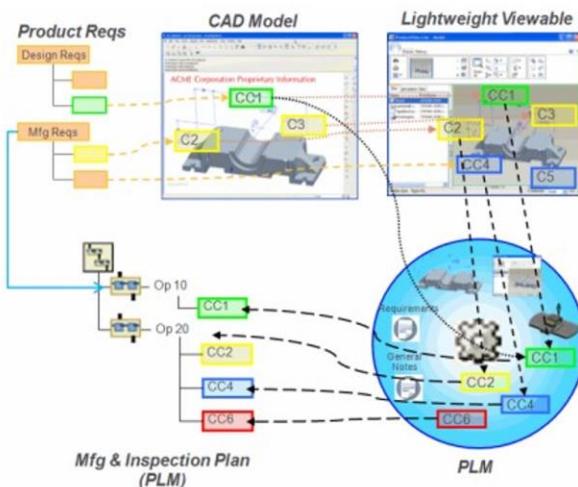
Windchill is Web-based and is completely integrated with Creo Parametric, therefore you can use hyperlinks within Creo Parametric drawing notes to link directly to information internal to Windchill, such as Change Notices (CN), or to any external URL. For example, drawings may contain revision blocks that refer to a CN number. Significant value is captured in these CNs along with the work done in the change process, because often times a CN holds test data, lessons learned, change requestors, and other information. By hyperlinking, this information is all readily accessible.



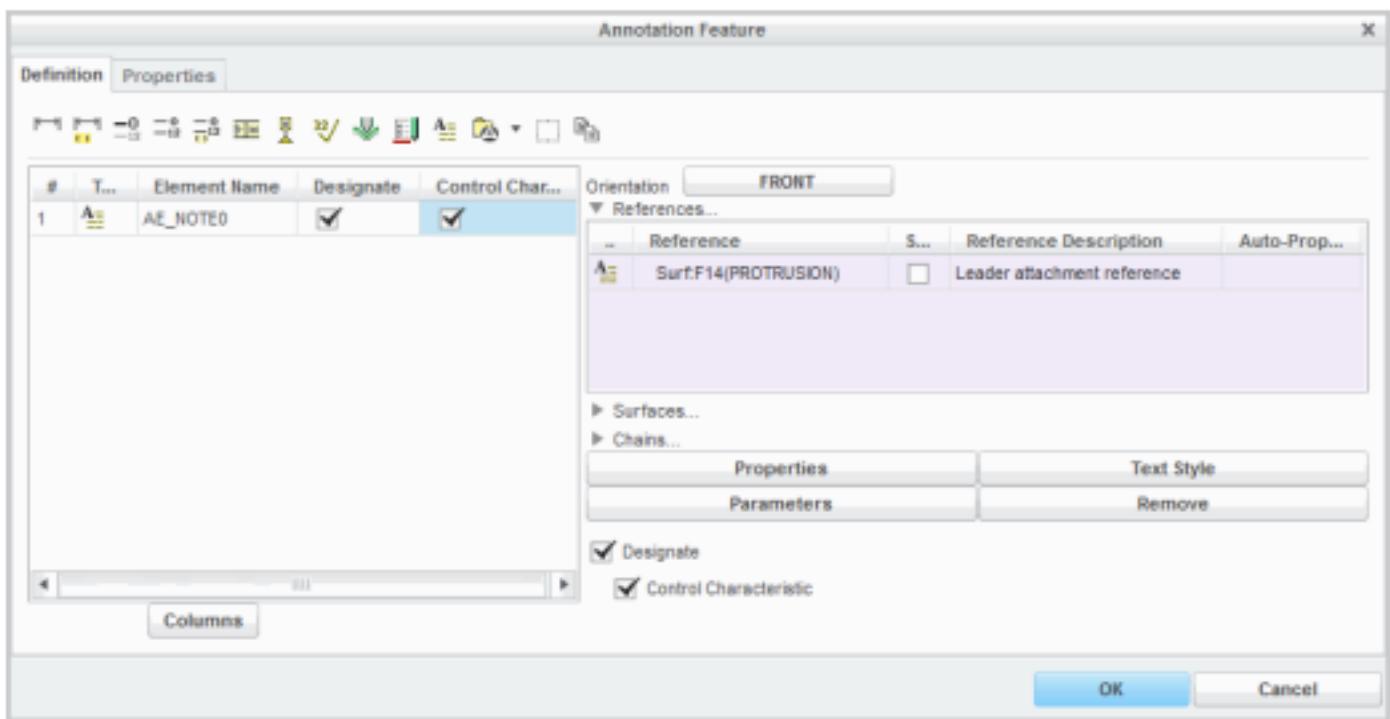
Understanding control characteristics

Annotation elements can be designated as control characteristics and published to Windchill.

- Build quality and inspection plans driven by the definition and management of control characteristics:
 - Publish CAD control characteristics to Windchill.
 - Define control characteristics in Windchill and Creo View Explorer.
 - Associate control characteristics directly to process plan operations (MPMLink).
 - Verify the allocation status of control characteristics.



- Annotation elements contain valuable information such as geometric tolerances, drawing notes, dimensions, surface finishing notes, and welding symbols. This information is particularly useful in the planning of downstream manufacturing processes. Therefore, you can designate annotation elements as control characteristics.
- Control characteristics, like model items, capture and communicate the design intent and critical product information required by manufacturing. Not only can you use these control characteristics to find and reuse model data, but the control characteristics are also useful in the planning stage because you can access information for manufacturing decisions without having to open the models in a CAD system to verify the model data.
- After you check in models with the annotation elements designated as control characteristics to Windchill, applications such as MPMLink can access the designated annotation element information. Therefore, you can designate annotation elements as control characteristics, according to manufacturing and machining requirements.



- Creo Parametric annotation features can be designated as control characteristics.
- In Creo Parametric, you click Annotate > Annotation Feature or right-click an annotation element from the Model Tree and then click Edit Definition. You then use the Annotation Feature wizard to designate annotation elements as control characteristics.
- To designate an annotation element as a control characteristic, you must select an annotation element in the Element Name column and select the Designate and Control Characteristic check boxes from the Annotation Feature dialog box.
- If you click the Columns button on the Annotation Feature dialog box to open the Annotation Columns dialog box, you can then select the Designate and Control Characteristics column names and add them as columns in the Annotation Feature dialog box. The Designate and Control Characteristics columns next to the Element Name column in the Annotation Feature dialog box, list information that can be viewed.
- The designation state of the annotation elements can also be changed. After you designate an annotation element in the Designate dialog box, you select the designated annotation element in the Object Name list and select the adjacent check box in the Control Characteristics column.
- When you check in the Creo Parametric models to Windchill, the CAD Documents and the corresponding WTParts list the designated annotation elements. While you can search features by their IDs, you can search and retrieve annotation elements by their IDs and names, including unique user-defined names. Like model items, control characteristics are associated with the CAD Documents and WTParts checked in to Windchill. You cannot independently edit control characteristics in Windchill.

Specialization program in PLM - Windchill

Day 8 Introduction to Visualization

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Identifying Visualization Concept

You can use visualization to review and interact with data throughout a product's life cycle

- Use visualization during:
 - Concept
 - Design
 - Production
 - Retirement
- Visualization enables you to review and interact with product data as it progresses from the conceptual stage to retirement, without requiring you to know how to use the original authoring application. For example, a new model automobile begins as an idea or a directive from product marketing.
- As that idea progresses, stylists draw how the car should appear, and engineers define the car's specifications, such as the engine size and the wheel size. By using visualization, you can view these initial drawings and specifications. You can also mark up and collaborate on these digital products in real-time with other people who are working on the new car project. You can do this all without knowing how to use the native applications used to create the drawings or specifications.

Identifying Visualized intellectual Property Types

A digital product's intellectual property includes all documented ideas that are owned by your company

- 3-D representations
- Drawings
- Bill of Materials (BOMs)
- Documents
- Schematic



- Visualization enables you to see both the digital product and the digital product's intellectual property.
- Using visualized intellectual property, you can view and share intellectual capital throughout the enterprise, mark up and collaborate on the digital products, and reuse existing product data throughout the product development process.
- The digital product's intellectual property includes all documented ideas that are owned by your company, including drawings, the Bill of Materials (BOMs), documents, and schematics.
- The 3-D representation is the most sophisticated visualization of the digital product. A few examples of the digital product include the CAD parts and assembly models that accurately describe the real-world product.

- These models can define and also be defined by the digital product's associated intellectual property
- Drawings provide you with more information about how the product appears. Drawings instruct engineers how to design the 3-D model's final appearance. Drawings can be derived from the 3-D model to instruct manufacturers how to create the physical components.


Identifying Components In Ptc's Visualization System



Empirical Learning

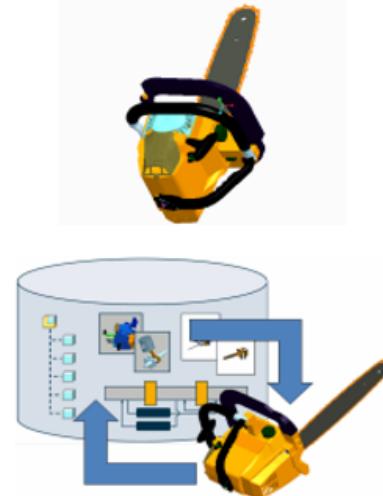
The Windchill Visualization Services (WVS) enables you to view what is stored in the product data management servers by using Creo View.

- Visualization components include:
 - Windchill server
 - PDMLink
 - ProjectLink
 - Pro/INTRALINK
- Windchill Visualization Services (WVS)
- Creo View client
- There are four versions

- Windchill Applications, such as PDMLink, ProjectLink, and Pro/INTRALINK, are product data management servers in PTC's product development system. These applications enable you to manage the digital product to ensure that no one overwrites or deletes your work.
- Additionally, all components of the digital product are stored on these systems in their native format. The Windchill Visualization Services (WVS) enable you to view what is stored in the product data management servers by using Creo View. Before any 3-D CAD models can be viewed in Creo View, however, they must be translated, or published, from their native format to a neutral, lightweight format.
- For example, Creo Parametric viewables are not stored as Creo Parametric assemblies. Rather, they are stored as lightweight, Creo View viewables. The WVS uses Object Adapters to perform this publish process. Publishing is important because if the data has not been published, it cannot be viewed in Creo View.
- Your Windchill administrator decides how viewables are created. This process may be automated, creating a viewable automatically, or it may be on-demand, created when requested by a user.
- The Creo View client enables you to view the translated models and the intellectual property. There are four editions of the Creo View client:



- Before most CAD data can be visualized in Creo View, it must be converted into a neutral file format
- Use the following visualization workflow:
 - Create the native data.
 - Upload the native data to the Windchill system.
 - Translate or publish data to neutral format using WVS worker.
 - View the file in Creo View



- Before you can view most CAD data in Creo View, it must be translated from its native format to a neutral, lightweight format. This process is called publishing and it is important for you to understand this process.
- If you are unable to view a file, then you must determine whether the file has been published. If it has not been published, you must wait while the Windchill Visualization Services (WVS) perform this task. Once published, you can view the files in Creo View.
- The first step in the publishing process is to create the intellectual property in its native application, such as creating a Creo Parametric model or drawing. If there is no original source in the system, a viewable cannot be created.
- After the native file is created, the designer must upload the model to the Windchill system. Now that the file is in Windchill, it can be published. PTC's visualization system within Windchill has three settings for publishing native files into the neutral format:
 - Automatic, which automatically converts files into a neutral format upon check-in to the Windchill system.
 - On demand, which only publishes the files when users specifically request to open them in Creo View.
 - Scheduled, which automatically converts all necessary files into the neutral format according to the schedule created by your Windchill Business Administrator.
- Regardless of when the publishing is performed, the publishing process sends the CAD files from the product data management system to the visualization worker. The worker translates the file into the neutral format and sends it back to Windchill.



Identifying Readable Visualization Object Types

Creo View enables you to open and view different Object types of viewable

- Object types include:
 - 3-D models
 - Drawings
 - Images
 - Document



Creo View reads multiple file types in four object type categories: 3-D models, drawings, images, and documents. Windchill Visualization Services publishes detailed 3-D CAD data into a rich 3-D model, which is Creo View's native format. In most cases, Windchill Visualization Services also publishes detailed 2-D CAD data into a 2-D drawing file.

Creo View also views other object types, including images and documents. As a result, Creo View is an all-purpose viewing tool that can be used to view representations and documents without requiring the authoring application.

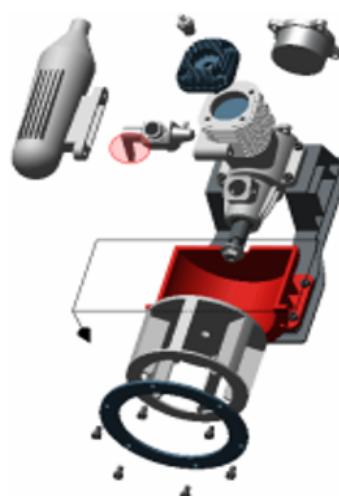
Although you can view all of these object types in Creo View, the application does not manage them identically. For example, you can section a model, but you cannot section a document. Similarly, you can flip pages and highlight in a document, but not in a model.



Understanding Creo Views Basic Capabilities

Creo View enables you to perform many basic functions.

- Basic functions include:
 - Finding and selecting components.
 - Managing multiple views.
 - Orienting models and drawings.
 - Changing model rendering.
 - Exploding model components.
 - Sectioning models.
 - Creating markups.
 - Measuring distances and displaying dimensions.
 - Viewing MBD annotations and combined states





The user interface includes the following areas:

- Viewing area
- Ribbon
- Creo View File menu
- Quick Access toolbar
- Primary Panel
- Upper Data Panel
- Lower Data Panel
- Panel display icons

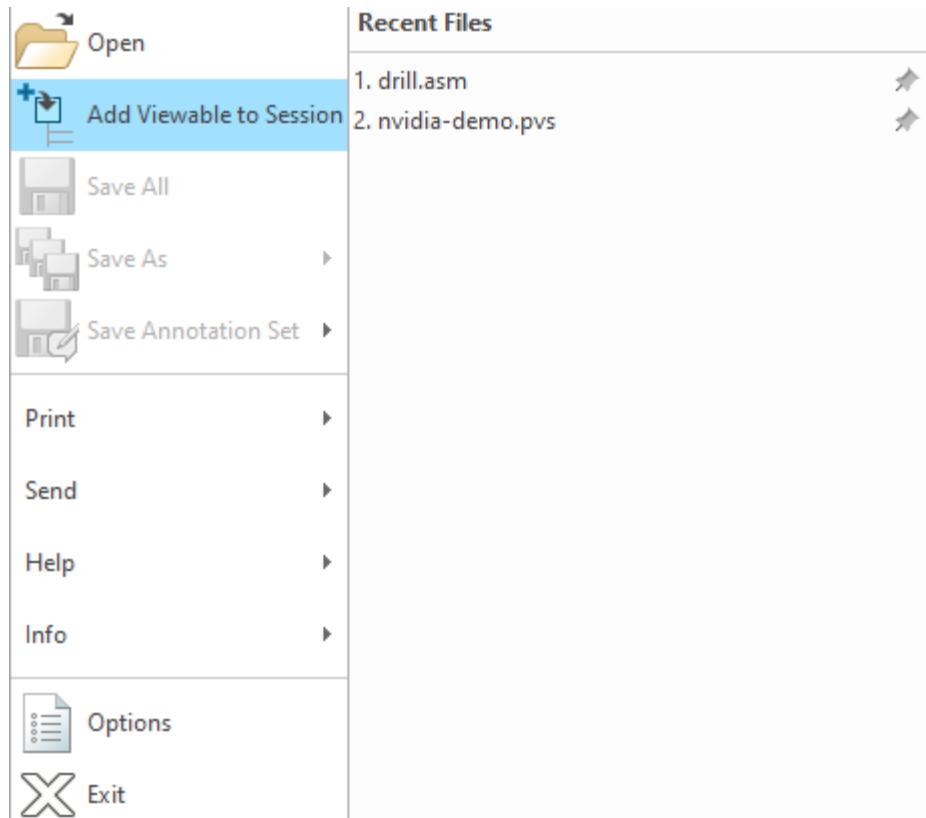
There are many different areas of the Creo View user interface that you use when viewing objects. Areas of the main interface include:

- The viewing area is the graphical area where you can view and manipulate 3-D models, 2-D drawings, images, and documents. This area may contain one or more individual windows that contain viewing objects.

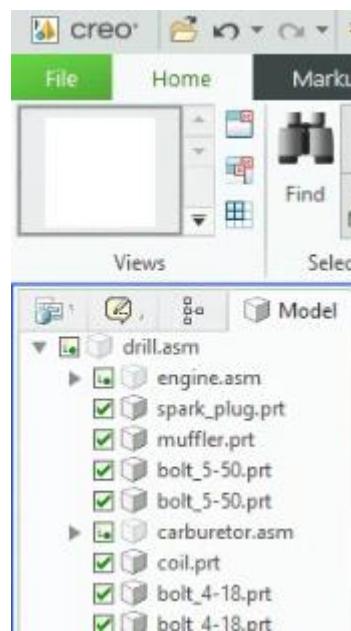


The ribbon provides access to a task-based set of functions. It contains the most commonly used sets of tools to accomplish a specific task. With the exception of the Creo View File menu, there is no menu bar available in Creo View. All of the commands can be found in tabs within logical groupings in the ribbon. Different tabs may display based on what is selected, and from where. The ribbon is similar to the ribbon found in Microsoft Office 2010 products.

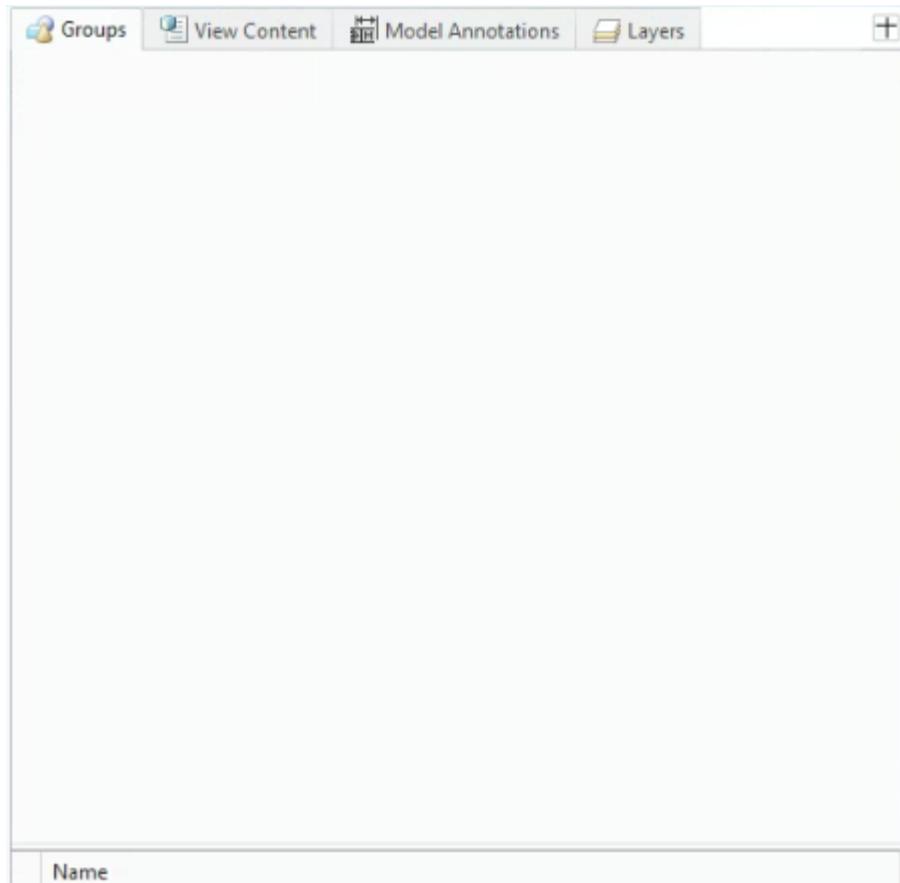
The Creo View File menu enables file operations and provides access to Creo View properties. This menu is accessed by clicking the File tab in the upper-left corner of the Creo View application. Again, it is similar to that of the icon located in Microsoft Office 2010 products.



- The Quick Access toolbar enables you to access a common set of tools that is always available in the interface. You can customize this toolbar to contain almost any command available in Creo View.



- The Upper Data Panel has multiple tabs along the top that enable you to work with groups, view model annotations, and layers. This panel is also called the Results panel.

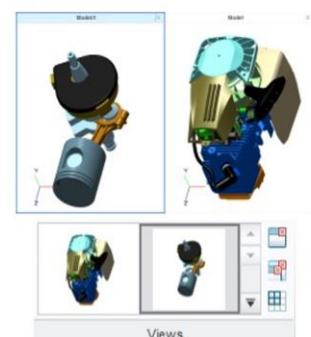


- The Lower Data Panel contains and displays data related to the currently selected assemblies or components. This panel is also called the Metadata panel.
- The panel display icons are a series of icons at the bottom that enable you to toggle the different panels on and off to modify the size of the viewing area.

Viewing Area User Interface

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- The Viewing Area section of the Creo View interface displays the current view or views that are open in View.
- Graphical information is displayed in view windows
- You can open multiple view windows.
- Toggle panels off to increase the viewing area size
- The Views group displays thumbnails of all corresponding open view windows



- The viewing area is the graphical display area in which documents, drawings, images, and models can be opened, viewed, and manipulated. Each separate object, annotation set, image, or document opened creates a distinct view opened within its own window.
- Each view window creates a new thumbnail within the thumbnail gallery area.



Understanding the Ribbon

The Ribbon is a task-based visual menu that enables access to all of the standard commands available in Creo View.

- Tabs

- Home
- Markup
- Tools
- Sectioning

- Groups

- Views
- Selection
- Navigation

- Commands

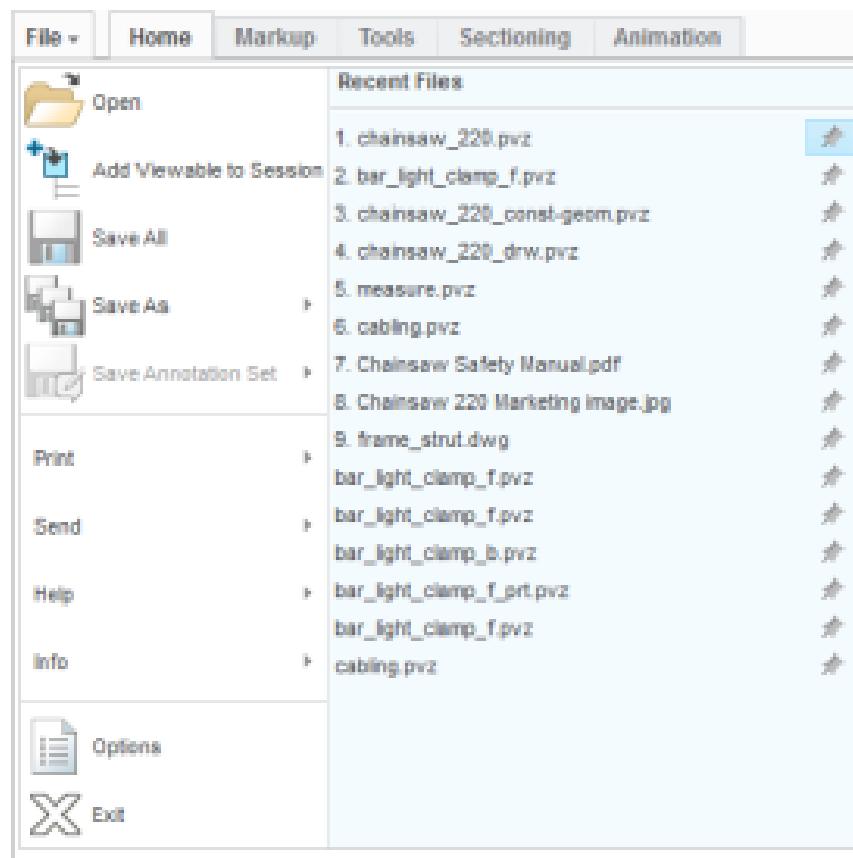
- Zoom All
- Zoom Window
- Zoom Selected
- Spin Center
- Orientation



- Understanding the Ribbon The ribbon is a task-based visual menu that enables access to most of the standard commands available in Creo View based upon the selection of a component, mode, or task. The ribbon is a horizontal strip at the top of the Creo View window and is similar to the ribbon found in Microsoft Office 2010. The command ribbon has three components: tabs, groups, and commands.
- The first component is the tabs located at the top of the ribbon that enable access to a dynamic set of groups. Creo View has many different tabs, and each tab is specifically designed to provide direct access to commonly used sets of commands for a specific function. 3-D models typically start with four primary tabs: Home, Markup, Tools, and Sectioning.
- Additional tabs such as Model Parts and Structure are dynamically generated based upon whether you are selecting objects in the viewing area or the Primary Panel. Other tabs may appear when other features are used.
- The second component of the ribbon is the group. Once a tab or task has been selected, the ribbon displays a set of groups that are typically used for that task. Each group contains a series of related functions and encompasses all of the commands associated with a single feature used in a task. For example, the Home tab includes groups for Views, Selection, Navigation, Display, and Location.



- The Creo View File menu contains general application operations for Creo View
- General application operations include:
 - File operations
 - Annotation operations
 - Print operations
 - E-mail operations
 - Creo View product information



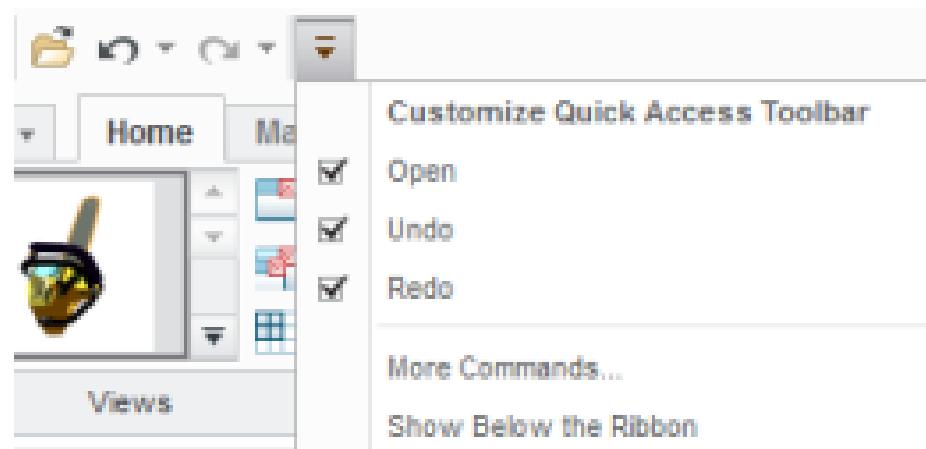
- The Creo View File menu contains file operations for Creo View including Open, Save All, and Save All As. In addition to the standard operations, you can also save annotations and markups, print, send a copy of the current view as an image in an E-mail, and view Creo View product information.
- Options for the Creo View application are accessed from the bottom of the Creo View File menu, and a list of recent files is shown in the second column for quick loading. The Recent Files list also contains push pins to keep specific recent files at the top of the list.



Quick Access Toolbar User Interface

The Quick access toolbar is a customizable toolbar that contains a set of commands that are independent of the tab currently displayed

- Default commands include.
 - Open
 - Undo and Redo
- You can customise the quick access toolbar.
 - Add or Remove commands
 - Move it above or below the command ribbon



- Creo View uses a task-based user interface which provides the most commonly used commands for the task or mode being used. The Quick Access toolbar contains a set of commands that are independent of the tab currently displayed in the ribbon.
- The Quick Access toolbar enables you to customize your environment by enabling your favorite commands to be available when working on any task. The default commands on the Quick Access toolbar include Open, Undo, and Redo.
- You can add a command by right-clicking a button or icon and selecting the Add to Quick Access Toolbar option or by using the drop-down menu that is part of the toolbar.
- The Quick Access toolbar can be relocated above or below the ribbon, or the ribbon can be minimized altogether from the toolbar menu.



Primary Panel User Interface

- The Primary Panel contains the entire list of viewable objects within the file and is the primary way to open different viewable objects, representations, or annotation sets that are contained within the file.
- The Primary Panel has multiple tabs, including the Viewables tab, Annotation Sets tab, Structure tab, and View tabs.
- The Viewables tab contains a list of viewable objects in the opened file. These objects are grouped by the following types: model, drawing, image, or document. The Model type displays any Creo Parametric simplified representations that were created in the viewables. It also displays any MBD combination states that were created in the viewables.
- The Annotation Sets tab displays all annotation sets available in the opened file. The annotation sets

can be listed as either a list view or as a tree view. The Structure tab displays the product structure tree of the opened file in a hierarchy similar to how it was created.

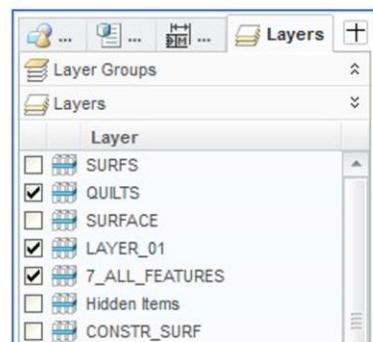
- For example, if you are viewing an assembly, the structure tab displays the order of how the components were added to the assembly. The product structure is analogous to Creo Parametric's model tree.
- The View tabs display information relevant to the object being viewed. One View tab displays per open view window in the viewing area. If three view windows are open in the viewing area, there are three View tabs displayed.

Understanding The Upper Data Panel

- The Upper Data Panel enables you to view various “buckets” and the contents of those buckets.

- The Upper Data Panel contains the following tabs:

- Groups tab
- View Content tab
- Model Annotations tab
- Layers tab buckets.

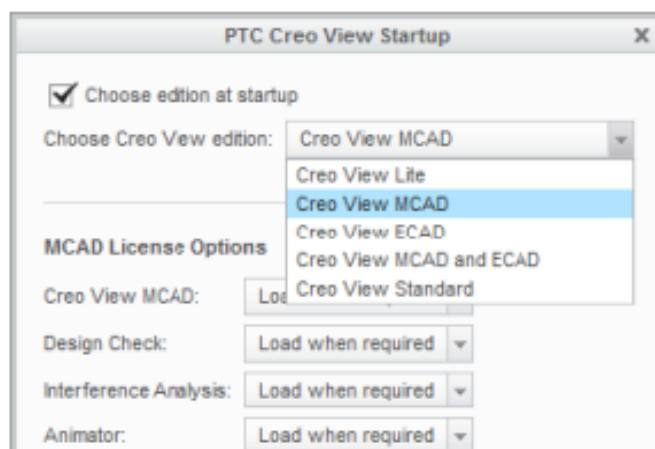


The Upper Data Panel enables you to view various “buckets” and the contents of those buckets. Each bucket is on a different tab, and the tabs include the following: the Groups tab, the View Content tab, the Model Annotations tab, and the Layers tab.

- The Groups tab displays any component groups that have been created in the Creo View session.
- The View Content tab displays a complete list of annotations, measurements, and construction geometry created in the current view. This tab enables you to select any of the elements and have them highlight in the current view, as well as perform numerous operations on them.
- The Model Annotations tab displays a list of all annotations created within the native Creo Parametric model.
- The Layers tab displays any layers that are a part of the model, drawing, or image.



- If you have visualization files on your local hard drive, you can open them into Creo View using Windows Explorer as you would any other file
- You can open any viewable local object type
- Double – clicking a local PVS, PVZ, OL, ED or EDZ file launches Creo view as standalone client
 - Initially you must specify the client type as well as any applicable licensing
- If you have visualization files on your local hard drive, you can open them directly from Windows Explorer as you would any other file.
- Simply double-clicking a PVS, PVZ, OL, ED, or EDZ from Windows Explorer launches Creo View with that file. Of course, you can also launch Creo View using a shortcut on the desktop, Start menu, quick launch toolbar, or taskbar, just as you would with any other application. As a standalone client, you can use Creo View to open any viewable local object type.
- The first time you launch the Creo View client, you must specify the client type to use. If it is available and appropriate at your company, you may decide to use Creo View MCAD or Standard. These require further licensing information that your administrator should have available. Otherwise, you may choose Creo View Lite, which does not require any further licensing.
- You may also wish to choose the desired edition each time the client is launched by selecting the Choose edition at startup check box in the PTC Creo View Startup dialog box.

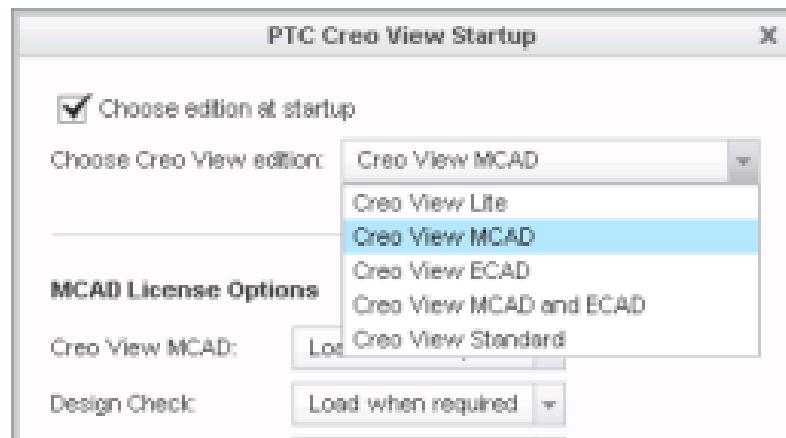
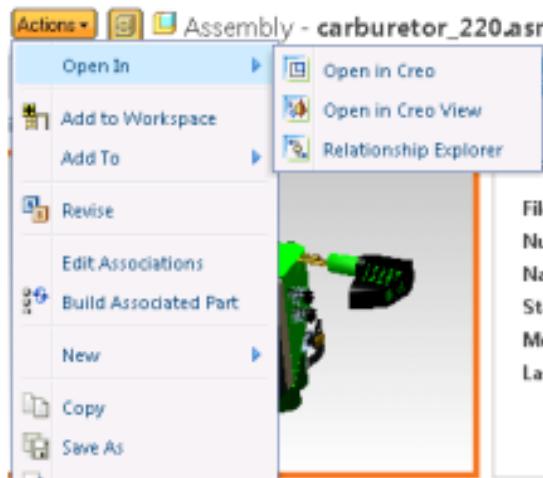


- Visualization files are usually located in Windchill. However, because the capability exists to save Creo View files directly from Creo Parametric, there is always a possibility that you may wish to open Creo View files outside of Windchill. Sometimes customers use local copies of Creo View viewables to create and save custom product structures that they do not want Windchill users to view.



Starting Creo View From Windchill

- Creo View launches automatically when you specify a valid object to view from Windchill.
- Launch from numerous UI locations, including:
 - Thumbnails on the Information page.
 - Thumbnail navigator.
 - Select the option from the Actions drop-down menu for a specific object.
- Initially you must specify the client type as well as any applicable licensing



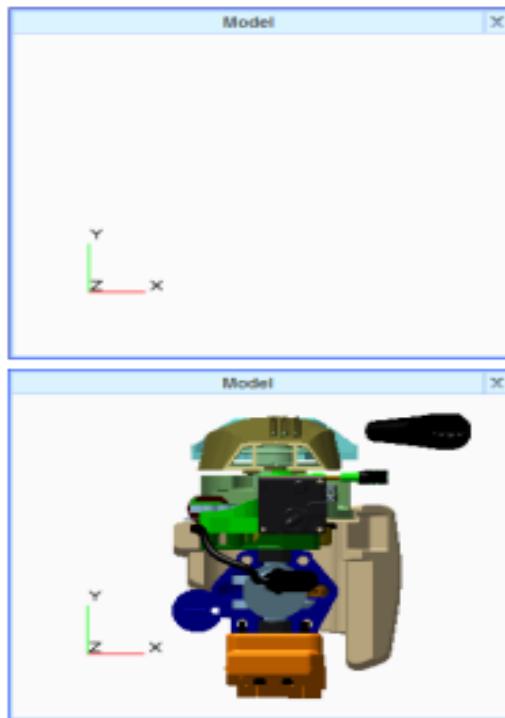
- Once the data has been published by the visualization service, you can view it within Creo View. Creo View launches automatically when you specify a valid object to view from Windchill. There are multiple methods for starting Creo View.
- You can start Creo View by clicking the thumbnail on the object's Information page. The thumbnail is available on the Details tab of the Information page. Also, if your Windchill installation has been configured to do so, you may find thumbnails embedded within the Product Structure table.
- When you cursor over a viewable icon in a table, the thumbnail navigator opens. You can start Creo View from the thumbnail navigator by clicking the Open default Representation in Creo View icon. You can also start Creo View by selecting the option from the Actions drop-down menu. The Actions drop-down menu is available from the Information page as well as within search and folder contents tables.
- The first time you launch the Creo View client, you must specify the client type to use. If it is available and appropriate at your company, you may decide to use Creo View MCAD or Standard. These require further licensing information that your administrator should have available. Otherwise, you may choose Creo View Lite, which does not require any further licensing.

- You may also wish to choose the desired edition each time the client is launched by selecting the Choose edition at startup check box in the Creo View Startup dialog box.

Loading Views

Within the view tab of an assembly, select the empty check box next to the component whose viewable you wish to load

- For 3-D models:
 - There are multiple methods to load assembly viewables.
 - Single models are loaded automatically.
 - You can just as easily unload components.
 - If you load an assembly, all child components are also loaded.
 - You can load any Creo Parametric simplified representations.
- For 2-D drawings, images, and documents:
 - The viewable is automatically loaded.
- Default load behaviors can be modified.



- When you open a 3-D model assembly into Creo View, the system loads an empty view window in the viewing area by default, along with a corresponding View tab in the Primary Panel. The primary reason for this is that some assemblies are very large, being composed of thousands or even millions of components. Instead of automatically loading all these viewables, you can selectively determine which components you want to add.
- The View tab in the Primary Panel displays the assembly product structure tree along with a check box next to each component in the assembly. As the component's viewables are loaded into the view window, the empty check box next to the component is populated with a check mark. If you load an assembly or sub-assembly, all of that assembly's child components are also loaded.
- Within the View tab there are three different methods you can use to load a components' viewables:
 - Select the empty check box next to the component whose viewable you wish to load.
 - Click and drag the component to view from the View tab into the view window.

- Right-click the component you wish to view and select Open in New View.
- To unload a component's viewable, simply select its check box to de-select it. The viewable then unloads from the view window.
- When you open an individual 2-D drawing, image, or document file, the viewable is loaded automatically.

Specialization program in PLM- Windchill

Day 9- Using Creo View Lite to View and Annotate Information

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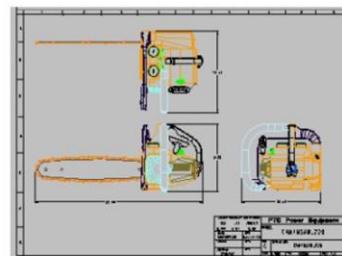
1



- Creo View is capable of opening, viewing, and annotating a large range of files which are not native Creo View models.
- These files are classified into three different groups:
 - Drawings
 - Images
 - Documents



Image



Drawing



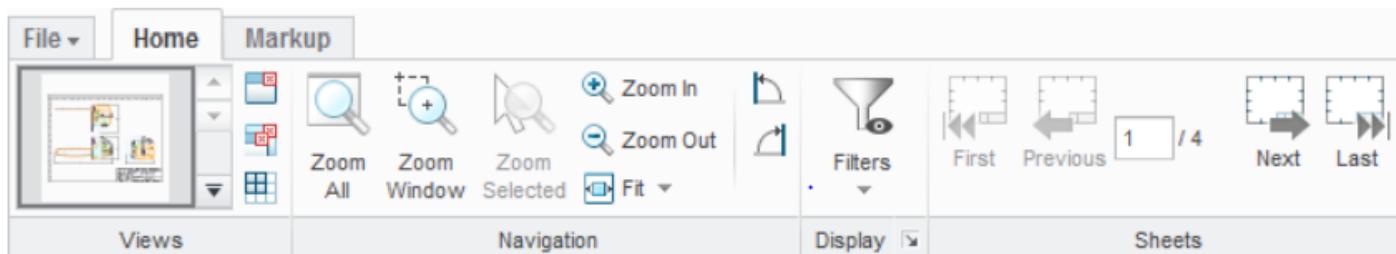
Document

- Drawings are precise technical illustration sets that represent an assembly, part, or system. Images are simply pictures that can be of any type including photographs, paintings, sketches, clip-art, or screenshots. Documents are files primarily composed of text and formatting

Viewing Drawings:

The Creo View functions available within a drawing file are many of the same features as that of a 3-D model, minus the 3-D functions.

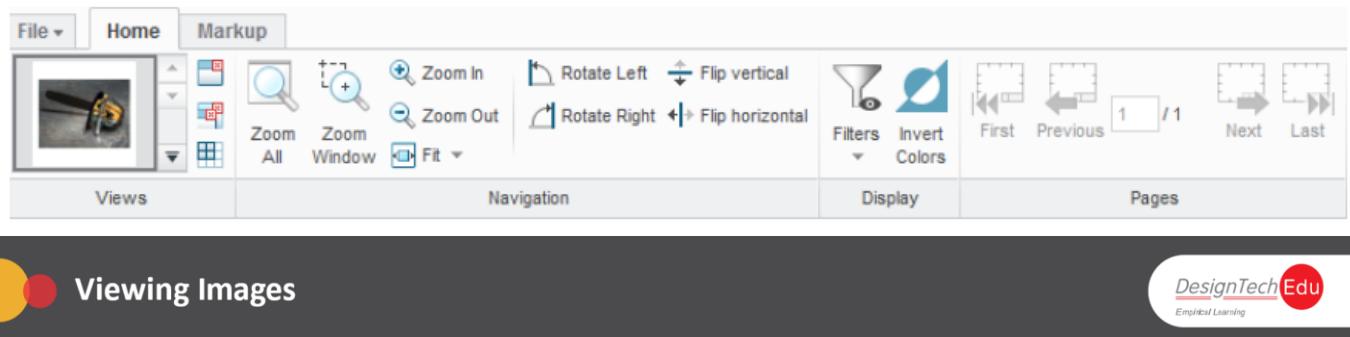
- The Creo View functions available within a drawing file are many of the same features as that of a 3-D model
- The Views group contains view thumbnails and view management functions.
- The Navigation group contains zoom and orientation functions.
- The Display group contains a single function to filter markups.
- The Sheets group contains functions to access the pages of a multi-sheet drawing.



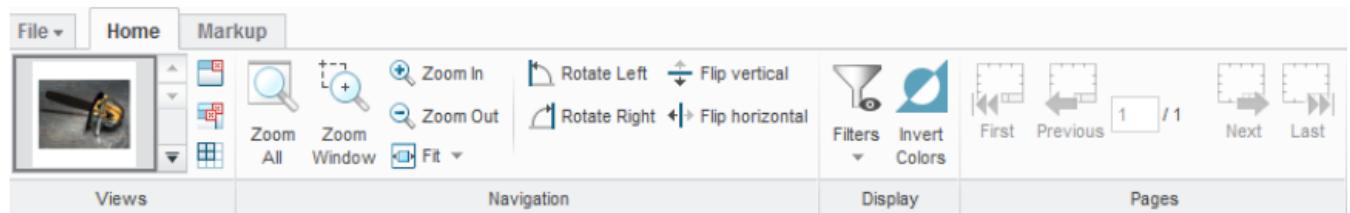
Viewing Drawings

- Aside from 3-D models, drawings are one of the most sophisticated representations of a digital product. As a result, the Creo View functions available within a drawing file contain many of the same features as that of a 3-D model except they do not contain the 3-D commands. For example, a drawing does not spin because spinning is a 3-D function
- Certain drawing types can be opened directly within Creo View without undergoing any publishing. For example, if you upload a DXF file to Windchill, you may open it directly in Creo View.

- The Views group contains view thumbnails and view management functions that enable you to open multiple views of a single drawing or work on multiple files at one time, if applicable.
- The Navigation group contains zoom and orientation functions.
- The Display group contains a single function to filter markups.
- The Sheets group contains functions to access the pages of a multi-sheet drawing.



- Most image types can be opened directly in Creo View natively.
- The Views group contains view thumbnails and view management functions.
- The Navigation group contains zoom and orientation functions.
- The Display group contains a filter function and color inversion function.
- The Pages group contains functions to access the pages of a multi-sheet image.

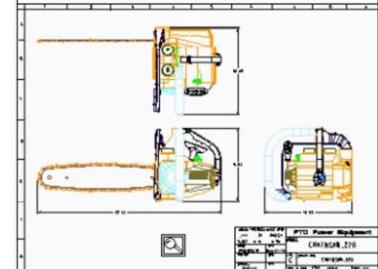


- Images are simply 2-D pictures. As a result, you are limited to a small set of commands for orienting and filtering the image. Note that most image types can be opened directly in Creo View without creating a viewable. For example, if you upload a .JPG image to Windchill, you may open it directly in Creo View and mark it up. Windchill does not need to publish the file to a different format.
- The Views group contains view thumbnails and view management functions in which you can open multiple views.
- The Navigation group contains zoom and orientation functions for the image. The Display group contains a single function to filter markups and an additional command to invert the colors of the image, enabling you to create a negative image. The Pages group contains functions to access the pages of a multi-page image.



Orienting Drawings and Images

- Drawings and images can be manipulated directly in the viewing area
- Keyboard and mouse navigation:
 - Pan
 - Zoom
- Navigation group commands:
 - Zoom All
 - Zoom Window
 - Zoom Selected
 - Fit Width/Fit Height
 - Rotate Left/Right
 - Flip Vertical/Horizontal

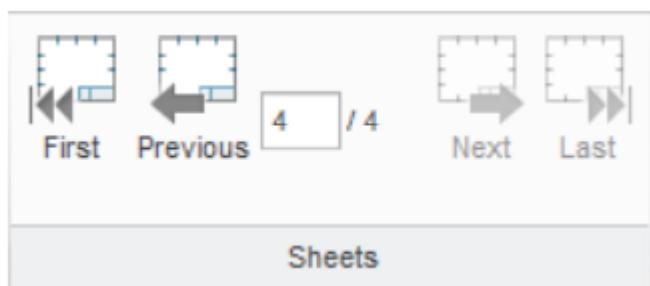
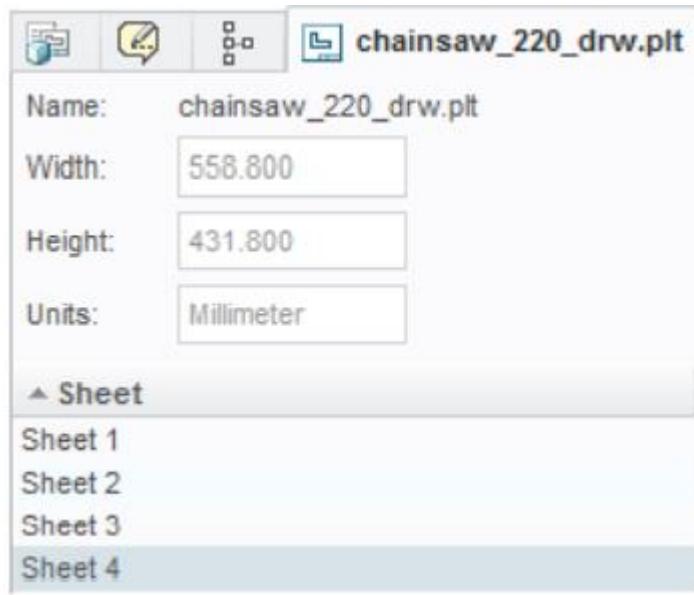


- Drawings and images can be manipulated directly in the viewing area. Similar to navigating 3-D models, you can zoom in, zoom out, and refit the drawing or image, as well as draw a zoom window around the specific area you wish to zoom in on.
- You can select entities in a drawing, but you cannot select entities in images because images do not have entities to select. Consequently, you can also zoom to the selected geometry in a drawing. Under the default Creo navigation model, you can perform the following basic keyboard manipulations on images and drawings within a view window:
- Zoom and Pan. Pressing CTRL while middle-clicking enables Zoom mode on both drawings and images, enabling you to zoom in or out by moving your mouse up or down.
- You can also zoom by using the scroll wheel. Middle-clicking enables panning as indicated by the hand icon. To help remember the correct keyboard and mouse combinations for a given operation, simply look at the status bar at the bottom of the Creo View client. The status bar indicates which mouse clicks correspond with the navigation features
- In addition to the common navigation functionality, you also have the following orientation functions available for drawings and images: Rotate Left, Rotate Right, Fit Width, and Fit Height.
- You can rotate drawings and images left and right, or counter-clockwise and clockwise, respectively. Fit Width and Fit Height functions fit the drawing or image to the view window width and height, respectively, enabling the remainder to expand out of the view window.
- Images also have further orientation functionality. You can flip an image horizontally or vertically. Flipping the image makes it display as if it were being seen in a mirror.



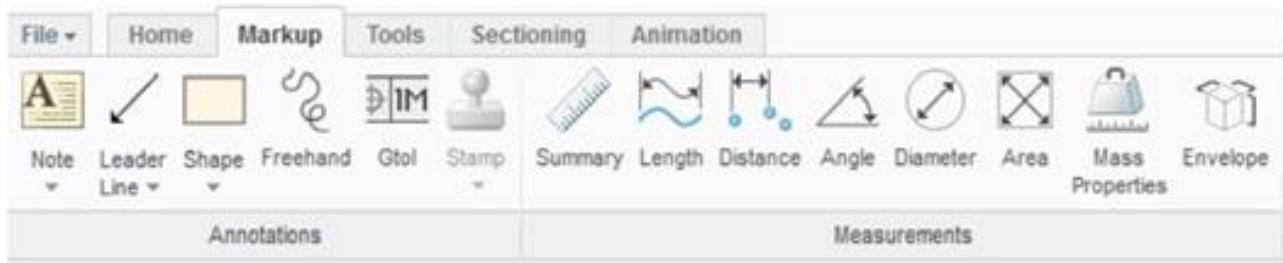
Working with Pages and Sheets

- The Sheets and Pages groups contain operations to access the pages of a multi-sheet or multi-page file.
- we can switch sheets or pages using a group in the ribbon.
- You can switch drawing sheets using the View tab in the Primary Panel.
- You can also type the sheet or page number you wish to view.



- Drawings, images, and documents with multiple pages contain a group in the ribbon that enables you to advance through the available pages or sheets or jump to a specific page or sheet. The First command displays the first page. The Previous and Next commands switch to the previous or next page within a file, respectively.
- The Last command displays the last page. For drawings and images, this group contains a page indicator for the current page out of the total number of pages. Using this current sheet field, you can access a specific page by editing its value to the desired sheet number.
- You can also right-click in the drawing or image and select Go To Sheet, which launches the Go to Sheet dialog box. Within this dialog box, you can also access a specific page. The View tab of the Primary Panel also displays the available sheets or pages of a drawing, image, or document. You can switch to a different sheet or page by selecting the desired number you wish to view.

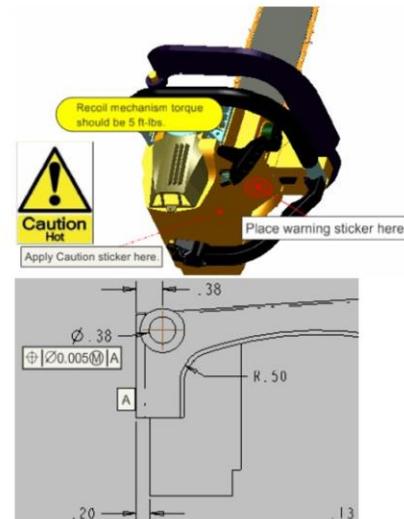
Marking Up Creo ViewViewables



- Marking up includes adding elements to a viewable to help communicate design intent, design changes, or design problems with other Creo View users.
- Markup elements do not modify the original file or viewable representation. They do not enable you to add new components, design additional parts to your model, or add additional pages to a drawing or document. Markups enable you to add text, notes, or shapes to point out issues. To edit model geometry, drawing lines, image pixels, or document text you must return to the original authoring application.

Adding Annotation Objects

- Annotation objects are essentially two-dimensional drawing tools that can call out specific features of the viewable.
- Add the following annotation objects to a viewable:
 - Notes
 - Notes with leaders
 - Leader lines
 - Shapes
 - Freehand lines and curves
 - GTOLs
 - Text markups
 - Stamps
 - Select and edit any added annotation objects.
 - Copy and paste annotations



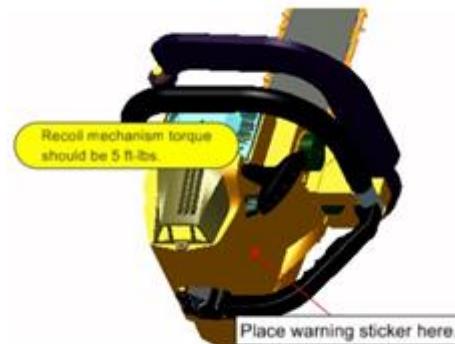
The Annotations group enables you to insert notes, lines, shapes, GTOLs, stamps, and mark up text. These functions are essentially two-dimensional drawing tools that can call out specific features of the viewable. You can select and edit any added annotation objects, as well as copy and paste annotations. In a model, annotation objects are created in the 2-D plane of the current orientation.

The Note function enables you to create a text box with a note within a specified shape.

- The Note with Leader function enables you to create a note within a specified shape with an attached leader line. The leader line enables you to connect the note to a specific location in the viewable or to another markup element.
- The Leader Line function enables you to create a line that calls out a specific feature or markup element.

- The Shape function enables you to create different types of shapes within the model. These shapes can be used to enclose or highlight specific features of the viewable.
- The Freehand function enables you to draw a freehand curve directly into the current view.
- The Gtol function enables you to add formal geometric tolerance notations as a note.
- The Text Highlight function enables you to highlight text in a document.

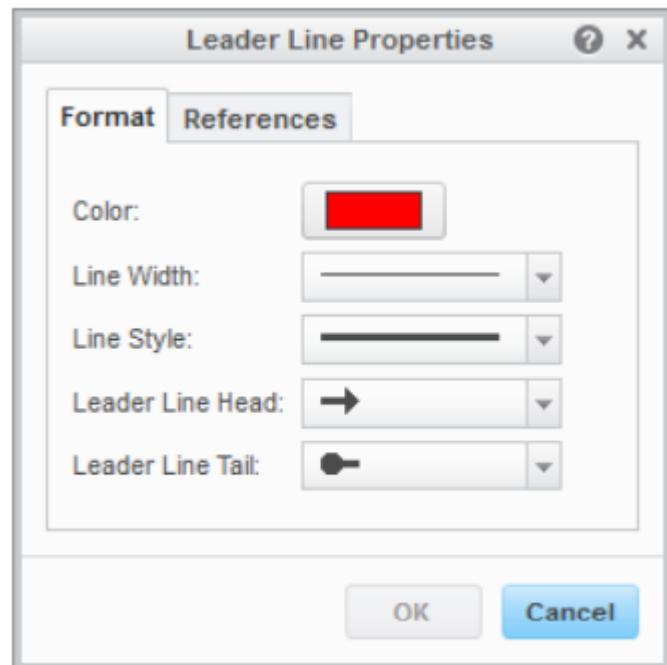
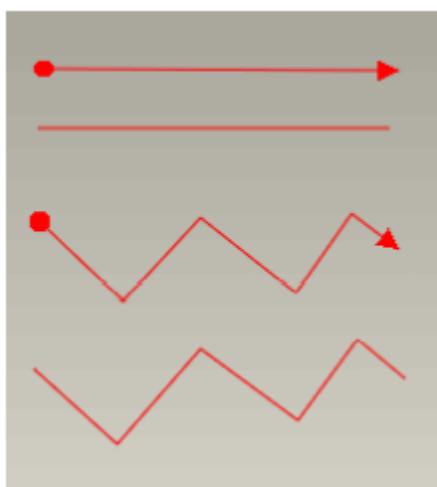
Note Annotations



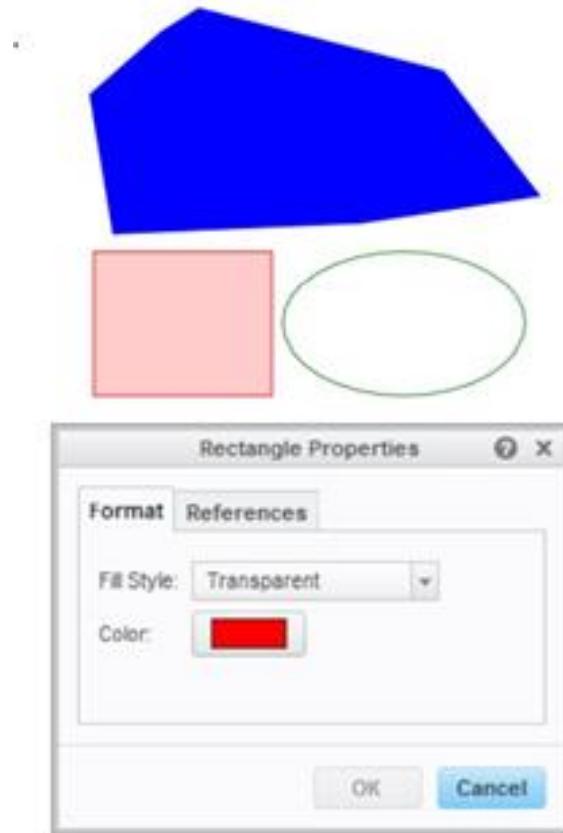
- Note annotations enable you to add text to your current view. Typically, these annotations are used to state issues or reminders or point out specific features which may require review, investigation, or changes. You can also insert attributes into the note text.
- The Note with Leader function enables you to create notes with an attached leader line to call out attention from the note to a related feature in the model view.

Leader Line Annotations

- Leader lines usually point from an annotation to a corresponding spot in the viewable.
- Create two styles of leader lines:
 - With head and tail
 - With multiple branches
- Format existing leader lines:
 - Color
 - Line Width
 - Line Style
- Arrow style:
 - Head
 - Tail
 - Edit the default leader line characteristics.
 - Lock the leader head or tail to a reference model



- Like the standard drawing tool sets in many image and document applications, the Leader Line function enables you to create different styles of leader lines. The two primary styles you can create are simple, straight leader lines with a head and a tail or complex leader lines with multiple branches or turns in the line.
- When creating a simple leader line with a tail and head, click in the view window to place the head location and drag the line to expand it to the tail location. Since all leader lines have a tail and a head, they all imply a direction.
- When creating a multi-branch leader line, select the location for your head, or starting point, and click at each location to jog the line through that point. To place the tail of this type of leader line, double-click to end the line



- Shape annotations can be used to highlight specific areas on a model. The Shape function enables you to create three different shapes:
- Rectangle,
- Ellipse, or
- Polygon.
- Each shape can be moved in the current view by dragging inside the shape's bounding box. You can also resize shapes by clicking and dragging the handles surrounding the bounding box.



Freehand Annotations

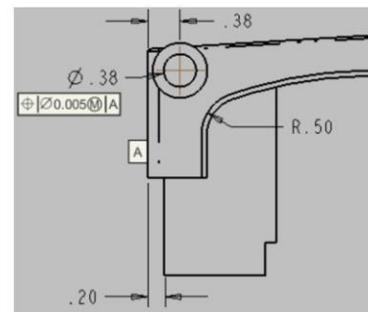
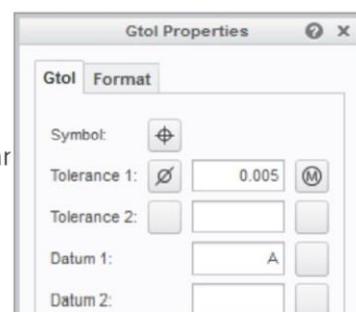


- The Freehand function enables you to draw a freehand curve directly into the current view. This feature creates a line drawn in the 2-D plane of the current orientation.
- Once a curve is drawn, you can double-click it and edit both the curve color and width.
- You can edit the default annotation color and line style in the Creo View Options dialog box under the Annotations > Line Style category within the Defaults option set.



GTOL Annotations

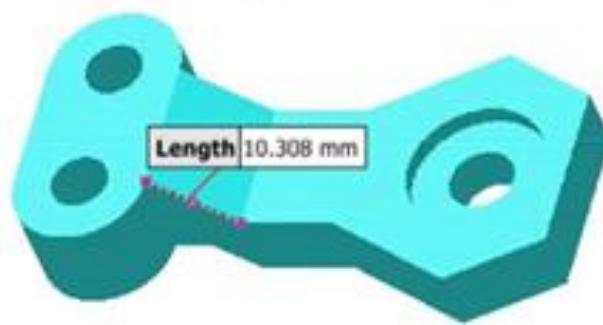
- Create a GTOL annotation to place a proper feature control frame into the viewing area
- Specify the tolerance condition.
- Specify the tolerance value.
- Specify a Diameter tolerance as needed.
- Apply material conditions:
 - MMC
 - LMC
 - RFS
 - Projected
 - Specify Primary, Secondary, or Tertiary datums.



- Geometric Tolerance annotations is a standardized notation set that establishes specific tolerance values of components and materials. Within Creo View, you can create a GTOL annotation to place a proper feature control frame into the viewing area.
- The Gtol function launches a Gtol Properties dialog box, enabling you to select the specific tolerance condition from the Symbol field. Once you have specified the desired tolerance condition to place, the Tolerance and Datum fields become active to fill in specific tolerance attributes.
- The Tolerance fields enable you to enter two overall tolerance values. The button preceding each tolerance field enables you to specify whether the tolerance value is for a diameter. The button immediately after the tolerance field enables you to specify the material condition, whether MMC, LMC, RFS, or Projected



Measuring Lengths

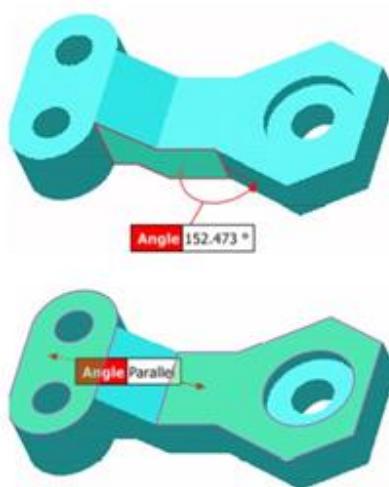


Reference	Measure	Value	Unit
MEASURE.PRT:Edge(Line)	Length	10.308	mm
MEASURE.PRT:Edge(Line)	Length	9.800	mm
MEASURE.PRT:Edge(Line)	Length	15.146	mm
All References	Total Length	35.254	mm

- The Length function enables you to measure the length of a curve or edge in a drawing or model. Pressing CTRL and selecting multiple curves or edges also calculates a cumulative length measurement. You can expand the Measure dialog box and view the individual edge lengths on a cumulative length measurement



Measuring Angles



- The Angle function enables you to measure the angle between two references within a drawing or model. The Measure Angle dialog box displays, enabling you to specify references. The system measures the angle between these two selected references.

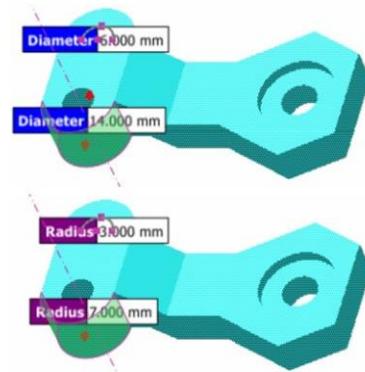
You can select Markups, Faces, Edges, or Datum's for the reference collectors. The Measure Angle dialog box has a check box that enables you to calculate the supplementary angle between the two references, measuring the external angle between the references.

If the two specified references do not connect or intersect, the system extends tangent planes or lines until they intersect to calculate the angle. If the selected references are parallel, the system indicates this in the viewing area.

Measuring Diameter or Radius

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- The Diameter function enables you to measure diameter or radii of curves or surfaces.
- You can specify references whose diameters are measured.
- You can toggle the measurements between a diameter and radius.



The Diameter function enables you to measure the diameter or radius of curves or surfaces in a model or drawing.

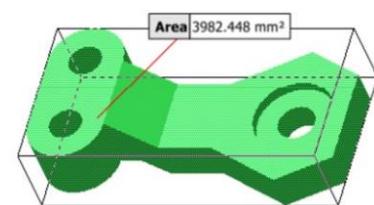
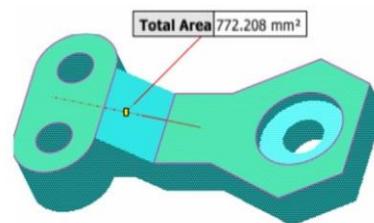
The Measure Diameter dialog box displays, enabling you to specify the diameter of a reference or references.

You can also select the Measure Radius check box to toggle the diameter measurements of all selected references to a radius measurement.

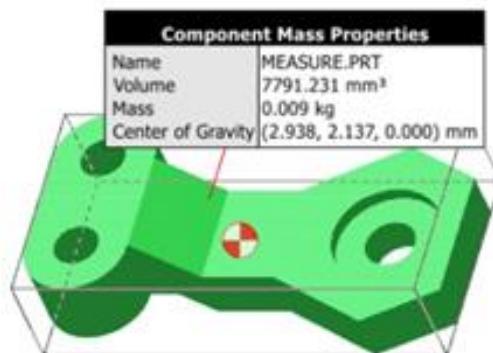


Measuring Area

- The Area function enables you to measure the area of a surface in a model.
- You can specify surface references that have measurable areas.
- Selecting multiple surfaces provides a cumulative measurement.
- Selecting the model measures its entire surface area



Measuring Mass Properties

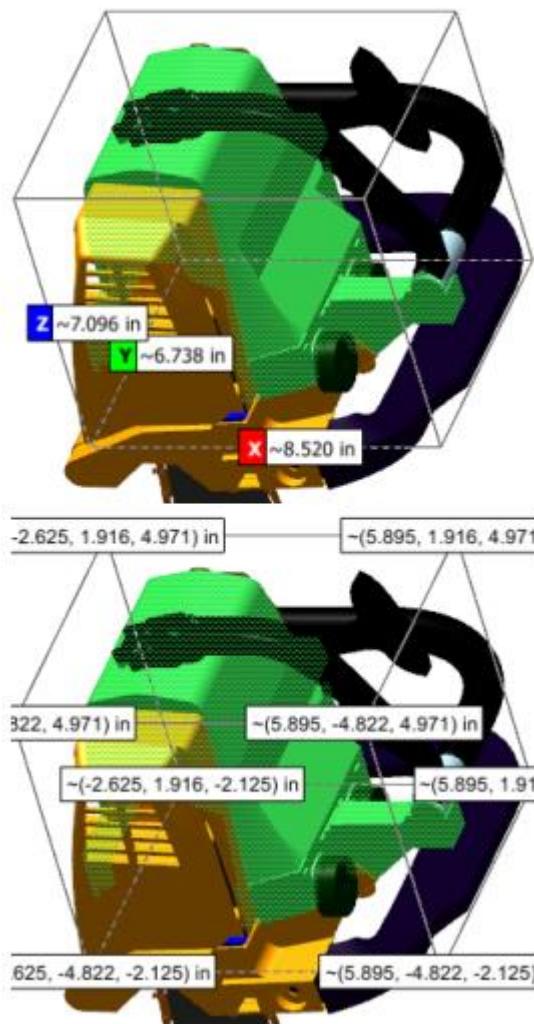


- The Mass Properties function enables you to measure the mass properties of a model
- Mass properties include:
 - Volume
 - Mass
 - Center of gravity
- Selecting multiple models provides a cumulative measurement.
- COG location displays on the model.
- The Mass Properties function enables you to measure the mass properties of a model. The Measure Mass Properties dialog box displays, enabling you to specify the model for which the mass properties are measured. Mass properties information includes the volume, mass, and center of gravity.
- You can also select multiple models to measure the cumulative mass properties. The viewing area displays a target at the physical center of gravity location on the model or models.



Reference	Measure	Value	Unit
MEASURE.PRT::Face(Plane)	Area	106.023	mm ²
	Perimeter	44.292	mm
	Normal	(-0.462, 0.000, 0.887)	
MEASURE.PRT::Face(Plane)	Area	126.017	mm ²
	Perimeter	50.005	mm
	Normal	(1.000, 0.000, 0.000)	
All References	Absolute distance	10.973	mm
	Angle	62.473	°
	Total Area	232.040	mm ²
	Total Perimeter	94.297	mm

- The Summary function calculates the appropriate measurement summary elements for any reference selected in a model or drawing.
- The Summary measurement information displayed is context-specific to the type of reference selected.
- Length for line and edge references.
- Volume for part and assembly references.
- Area for surface references.
- Multiple references yields the sum total of applicable measurements.
- Individual reference summaries are displayed in the Results section



- The Envelope function enables you to view the approximate X, Y, and Z dimensions of the selected component's bounding box. This measurement type is only available on 3-D models because you can only select components as references.
- You can press CTRL to select multiple components. The selected components display in the Measure Envelope dialog box as references. If multiple components are selected, the system displays the dimensions of the smallest bounding box that fits around all selected components.
- As an alternative to displaying the X, Y, and Z bounding box dimensions, you can also toggle the display to that of the corner coordinates of all corners of the bounding box.



Saving Annotation Sets to a PDM System

- Annotation sets are automatically stored within Windchill.
- Annotation sets are stored as markup objects within the PDM system.
- Do not confuse a markup object with Creo View's markup capability.
- Markup objects within the PDM system are iteration-specific.
- Copy Forward enables you to keep annotations. –
- Advantages: You can keep comments between iterations.
- Disadvantages: Product structure may change between iterations, rendering markup objects obsolete.

Representations/Annotations

Name	Thumbnail	Description	Details	Locked By	Owner	Date
batch		Scheduler	EPM only EPM: As Stored Derived From: Assembly ...		Jones, Mike	2011-02-28 16:40 EST
wt.maturity.PromotionNotice:70329		Promotion R...	EPM only EPM: Baseline - Derived From: Assembly ...		Smith, Fred	2011-02-28 17:14 EST
default			EPM only EPM: As Stored Derived From: Assembly ...		Smith, Fred	2011-02-28 16:34 EST
Shaft-Rod gap			Open in Creo View Save Annotation Lock Annotation Enable Copy Forward Copy Delete	Disabled	Smith, Fred	2011-03-17 10:31 EDT
Fred's Comments				Disabled	Smith, Fred	2011-03-17 10:33 EDT

- It is also important to understand that markup objects within Windchill are iteration-specific. This means that whenever a user checks out a file, makes changes, and checks in the file, all annotation sets can only be viewed if you retrieve that specific iteration.
- There is a method in PDM systems called Copy Forward that enables you to keep all annotation sets. This option copies the annotation set regardless of the iteration. However, you should note that the annotation set may become obsolete as new information is added.



- There are two methods you can use to open annotations.
- Open the viewable file and then open the annotation set from the Annotation Sets tab within Creo View.
- Directly open the annotation set from Windchill..



- When you select the Creo View representation file from within Windchill, the appropriate viewable file and attachments are downloaded to your local system, and Creo View is launched to open that file.
- Annotation sets stored in a Creo View file can then be retrieved from the Annotation Sets tab in the Primary Panel. By default, annotation sets are displayed as thumbnails, and you can open a new view window containing the annotation set by double-clicking it. A new view tab also opens.

Specialization program in PLM- Windchill

Day 10 Creating and Managing Documents

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1



Document Capabilities

Documents are used to store and manage file content.

- Attributes:
- Document name
- Document creator
- Content:
 - File or URL (Primary)
 - Additional attachments
- Relationships:
 - Automatic
 - Explicit

Windchill documents are used to store and manage files, URLs, or externally stored content.

- A document has attributes, including the document name and document creator.
- A document can have primary content, such as an electronic file or URL. You can also add additional files as attachments.
- A document can have relationships with other Windchill objects. Relationships can be created automatically as part of another activity. For example, when a document is checked out, a link is created between the working copy and the public copy. Relationships can also be created explicitly. For example, a document may be contained within a baseline configuration.



Creating Documents and Links

You can create documents and links from several different locations in Windchill in many different ways.

Documents created four ways:

- New document
- New from template
- New multiple
- Bulk load

Create actions accessed by:

- Folder contents table icon
- Folder actions
- Automatically

- There are four methods for creating documents: Create a single document – This option provides the most flexibility in creation, as you have control over all attributes, attachments, and files. However, it is also the most labor-intensive option.

- Create documents from a template – This option is available when creating single documents. The template may provide default attributes and/or content files. For example, a specifications document template may automatically contain a Microsoft Word document with header and footer information as primary content.
- Create multiple documents – This option enables you to select several files and create a separate document for each. This can save significant time as opposed to creating several single documents, but some flexibility is lost. For example, you can only add primary content and cannot add attachments.
- Upload files in bulk – This option enables you to upload multiple documents within a zip or jar file. These documents are then extracted by Windchill into the folder location. Additionally, if you capture subfolders within the zip file, you can set Windchill to create these subfolders in the folder location
- You can create documents by clicking the New Document icon at the top of the Folder Contents table. You can also create documents, multiple documents, and links from the Actions drop-down menu of a specific folder.
- When you create an object from the Actions drop-down menu, the folder is preselected for you in the Create Wizard and you do not have to set it manually. Lastly, some documents are created automatically. Arbortext dynamic documents automatically generate a standard document when published. Bulk loading processes also create documents automatically.

New Document: Set Attributes – Type

The document type may affect the attributes, administration, and access control for the document

Document type may affect:

- Document Attributes
- Page Layouts
- Access Control
- Related Windchill Processes
- Object Initialization
- Advanced Search

The first step of the New Document Wizard requires you to specify the type of document you are creating. The type you select may determine unique attributes and administration for the document object. Two common out-of-the-box document types are Document and Reference Document. Selecting a different document type may affect:

- Attributes – For example, a plan document may have a due date attribute while a minutes document may not.
- Layouts – The pages used to view, edit, and create the document may differ. For example, the object information page may display the creator of a plan document more prominently than the creator of a reference document.

- Access Control – Layouts can affect access control. For example, a manager may be able to edit a plan document, but an engineer may be required for other types of documents that are directly product-related.
- Windchill Processes – Different document types may start different workflow processes and tasks, resulting in different items in the team's work list. For example, a presentation document may not start a workflow at all, but a plan document may start an approval process for the plan.
- Object Initialization – Selecting a different type may automatically populate or constrain attributes. For example, a minutes document may only permit the meeting date to be in the past while a plan document may require it to be in the future. Or, all plan documents could be pre-configured to be created in the same folder.
- Search functionality – In Windchill advanced search, you can search for a specific type of document.

A template may prepopulate attributes and content files.

- After you set the document type, the rest of the Set Attributes page appears in the wizard. The options available on this screen vary depending upon the document type and the document template.
- You may optionally select a document template. A document template may pre-populate primary content files, attachments, and attribute values.
- For example, a presentation document may automatically insert a Microsoft PowerPoint presentation with the appropriate design template for your organization. Similarly, a memo template may include a blank Microsoft Word memo. If you do not select a template, you are able to add your own primary content.
- Typically, the content file included with a template is not complete. At some point, a user must modify the document to complete the content file. However, this user does not need to be the same user that created the document.



When creating a document without a template, the New Document Wizard enables you to set primary content.

New Document

The screenshot shows the 'Set Attributes' step of the New Document wizard. The 'Product' field is set to 'Drill - 400 Series'. The 'Type' field is set to 'Document'. The 'Template' field has a dropdown menu with options: Local File, No Content, Local File (selected), URL Link, and External Storage. The 'Primary Content Source' field is set to 'Local File'. The 'File Name' field is empty. The 'File Description' field is also empty. A 'Browse...' button is available for selecting files.

- The New Document Wizard enables you to specify the source of the primary content. A number of options are available from the Primary Content Source drop-down menu.
- The No Content selection enables you to create a document object without any primary content. Typically, a document object without content is used as a placeholder to which content is added at a later time
- The Local File selection enables you to select and upload a file from your local system. You can populate the Local File Path field by clicking the Browse button to browse to the file on your computer.
- The URL Link selection enables you to define a URL as the document's content. This creates a document that is similar to an advanced link. It links to a URL but it can also have attachments, iterations, and attributes that link objects do not have.
- The External Storage selection enables you to designate a path to a file stored in another server. This is useful for managing large files, which can be maintained on a local server without requiring the file to be uploaded and stored in the PDMLink Oracle database.



New Document: Set Attributes (Attributes)

The New Document Wizard enables you to set document attributes.

New Document

1 2

Set Attributes Set Attachments

Product: Drill - 400 Series

*** Type:** Document

Template: -- Select a Template --

*** Primary Content Source:** Local File

*** File Name:** Drill Safety Manual.doc

File Description:

Attributes

Number: (Generated)

*** Name:** Drill Safety Manual

Description: Drill Safety Guide

*** Location:** Autoselect Folder (/Drill - 400 Series) Select Folder /Drill - 400 Series/Documentation

*** Life Cycle Template:** (Generated)

Team Template: (Generated)

- The New Document Wizard enables you to set document attributes,
- The Name attribute may be pre-populated based on rules. If you select to store a local file, the Name field is automatically populated with the file's name. You can then change that name and type a description for the document. The asterisk identifies required fields.
- If your implementation does not employ auto numbering for documents, you must also type a document number.



New Document: Set Attachments

The second page of the New Document Wizard enables you to add attachments.

The second page of the New Document Wizard enables you to specify the source of additional files that you want to store as attachments. Attachments can be any combination of three types:

- A local file
- A URL link
- An external storage file

	*Label or File Name	*URL/External Location	Attachment Description	
<input type="checkbox"/>	drill cover image.jpg	<input type="button" value="Browse..."/>	<input type="button" value="Up"/>	<input type="button" value="Down"/>

(0 objects selected)

- The second page of the New Document Wizard enables you to add attachments
- The second page of the New Document Wizard enables you to specify the source of additional files that you want to store as attachments. Attachments can be any combination of three types:
- A local file can be added as an attachment by clicking the New local file attachment icon and browsing to the file.
- A URL link can be added as an attachment by clicking the New URL link attachment icon.
- An external storage file can be added by clicking the New external storage attachment icon.



Modifying Documents - Check Out

- Public copy
- Shared copy available to other Windchill users.
- Cannot be edited but may be viewed.
- Linked to the user who checked it out.
- May only be checked out by one user at a time.
- Checked Out Copy

- Working copy
- Working copy available only to person who checked out public copy.
- Can be edited.
- Other users do not see edits until document is checked in.
- Working Copy

Modifying Documents - Check Out

- To modify a document, you must check it out. This signals to others that you are modifying the document. This process creates two copies, a working copy that you can modify and a checked out copy that others can view.
- The public copy is:
 - Viewable, but not editable by other Windchill users.
 - Linked to the user who checked it out.
 - Public copy has the icon addition next to it in Windchill tables. The working copy is:
 - Accessible only to the user who checked it out.
 - Working copy may be edited.
 - Edits may not be seen by other Windchill users until it is checked in.
 - Working copy has the icon addition next to it in Windchill tables.

As long as you have an object checked out, others may view the checked out copy but they cannot modify it. This prevents others from making changes to the object before you have completed your work.

• Modifying Documents - Check In

- Checking in a document creates a new iteration
- Public copy
- Original public copy retained as old iteration.
- New public copy created as new iteration

Modifying Documents – Edit

Edit

1 2

Set Attributes **Set Attachments**

- Public copy
- Unchanged.
- Public copy is no longer the same as working copy.

- Working copy
- Changes to working copy only.
- Interim edits to working copy are not maintained

Edit



Set Attributes **Set Attachments**

Product: Drill - 400 Series

Type: Document

Number: 0000000116

*** Primary Content Source:**

Local File

Upload file

Do not upload file (The server copy of Drill 400 Owner's Manual.doc will be kept)

*** File Name:**

Drill 400 Owner's Manual.pdf

Browse...

- When you edit a document, you change the working copy. You may change its files or attributes. However, these changes do not affect the public copy. As a result, after an edit, the working copy and public copy are no longer identical.
- In addition, when you edit a document, the old working copy is overwritten and cannot be retrieved.



Modifying Documents - Check In

- Public copy
- Original public copy retained as old iteration.
- New public copy created as new iteration
- Working copy
- Becomes new public iteration.
- Object is no longer checked out, so no working copy remains

Checking In Document - 0000000116, Drill 400 Owner's Manual, A.1

* Primary Content Source: Upload file Do not upload file (The server copy of Drill 400 Owner's Manual.pdf will be kept)

* File Name:

File Description:

Comments: Converted to PDF for review Keep checked out after checkin

Once you have made your changes, you can check the object back in. When you check in an object, the following actions occur:

- The public copy remains the old iteration.
- You have the option to change attributes or files upon check in. This is a shortcut, functionally identical to editing the document and then checking in the working copy without changing its files or attributes.
- The working copy becomes public and it is the most recent iteration.
- The most recent iteration is available for other users to check out.
- The old iteration is locked. You may view it but cannot check it out or update it



Modifying Documents - Undo Checkout

- Public copy
- Original public copy is unchanged.
- Public copy is unlocked and may be checked out by another user
- Working copy
- Deleted.
- Not saved in Windchill.
- Log may be kept for audit purposes.

The screenshot shows a software interface for managing a document. At the top, there's a header bar with tabs for 'Actions', 'Document - 0000000143, Drill 400 Owner's Manual, A.1', and other options like 'Changes' and 'History'. On the left, a sidebar titled 'Actions' lists various document management tasks: 'Download primary file', 'Open primary URL', 'Open in ProductView', 'Check In', 'Check Out', 'Check Out and Download', 'Check Out and Edit', 'Undo Checkout' (which is highlighted with a dashed border), 'Edit', 'View', and 'Replace Content'. To the right of the sidebar is a large preview window showing a document icon with horizontal lines. Below the preview, under the heading 'Attributes', is a table with the following data:

General	
Name:	Drill 400 Owner's
Status:	Working copy, cl
Primary Content:	Drill 400 Owner's
Modified By:	Hill, Erica
Last Modified:	2011-02-17 14:25

Undo checkout releases the lock on the public copy and deletes the working copy. This is useful in a few circumstances:

- You do not want to save the changes you made to the document.
- Another user asks you to unlock the file, but your work is not yet complete. You can release the file and perform your work later.



- Public copy
- Original iteration is unchanged
- New iteration has an updated content file.
- Working copy
- Used internally by Windchill, but never seen by users

Replace Content

* Primary Content Source: Local File Upload file
 Do not upload file (The server copy of Drill 400 Owner's Manual.pdf will be kept)

* File Name: Drill 400 Owner's Manual.doc

File Description:

Comments:

Keep checked out after checkin

Replace Content is a convenient action that performs several subactions:

- It checks out the document.
- It edits the working copy, changing the primary content file. The working copy is not normally seen by the user, as it is checked back in quickly.
- It checks the document back in, creating a new iteration and removing the working copy.



You can associate standard documents with Windchill parts.

A document can be associated with parts in two ways.

- The document can describe a part or,
- It can be referenced by a part.

The screenshot shows the Windchill interface for managing part documents. The top navigation bar includes 'Actions', a search icon, and the title 'Part - DRILL_720, drill_720, A.3 (Design)'. Below the title are tabs for 'Details', 'Structure', 'Related Objects' (which is highlighted in orange), 'Changes', 'History', 'Where Used', and a '+' button. Under 'Related Objects', there are two expandable sections: 'Described By Documents' and 'References Documents'. Both sections have a table with columns: Number #, Version, Name, Context, and Status. In the 'Described By Documents' section, there is one entry: Number # 0000000141, Version A.1, Name New Products Launch, Context Drill - 600 Series, and Status In View. The 'References Documents' section also has one entry: Number # 0000000142, Version A.1, Name Drill Owner's Manual, Context Drill - 600 Series, and Status Relates To.

Number #	Version	Name	Context	Status
0000000141	A.1	New Products Launch	Drill - 600 Series	In View

Number #	Version	Name	Context	Status
0000000142	A.1	Drill Owner's Manual	Drill - 600 Series	Relates To

Specialization program in PLM- Windchill

Day 11 Change Management Process Overview

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Change management process overview

In general terms, the process has five major steps,

- Identify the need
- Investigate the need
- Change planning
- Change implementation
- Review and Audit change

Windchill provides an integral and automated change process with all the necessary tools to accommodate change throughout the product lifecycle. In general terms, the process has five major steps:

1. Identify need – The goal of this step is to enable product development contributors and external parties to report problems and enhancements to products. The information is captured using Windchill problem reports.

2. Investigate the need – When one or more problem reports have been approved, the change team can proceed with an in-depth change investigation. The change investigation is captured in a change request and the goal is for product experts to accurately assess the scope of the change, feasibility, downstream product impact, solution proposals, cost, and business justification.

3. Change planning – The goal of this step is to plan the detailed implementation of the change including schedule, work tasks, and resources. The change planning is captured in a change notice.

4. Change implementation – The goal of this step is to implement the change. Upon approval of the implementation plan, all the work tasks per the implementation plan are automatically delivered to the designated users.

5. Review and Audit Change – The goal of this step is to review, audit, and release the change.



Change objects

Problem reports, change requests, change notices, and variances are integral to product configuration.

Change Item Capabilities:

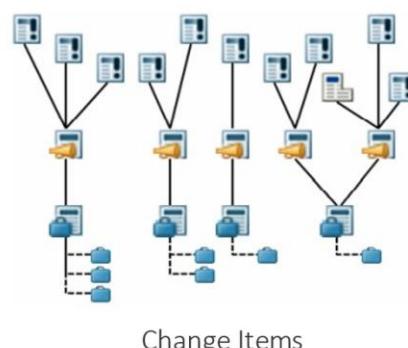
- Problem Reports
- Change Requests
- Change Notices

Assign:

- Change Tasks

Variances:

- Deviation
- Waiver



Items such as problem reports, change requests, change notices, and variances become integral to the life and structure of the product model. These items capture the information that is routed through the change process.

- Problem report – Can be completed by anyone suggesting a change or reporting a problem with a product/part. Typically, problem reports are used to document problems submitted in testing, in the field, or by customers
- Change request – Can be used to gather common problem reports and submit a formal request for a change. A change request can be initiated without any problem reports.
- Change notice – Can be used to assign tasks that complete the implementation plan and implement the changes.
- Variance – An authorization to depart from the designed configuration for a specific number of units or for a specified time period. Variances consist of deviations and waivers. A deviation is a planned departure from the as-designed configuration before the part is built. A waiver is a written acceptance of nonconformance

All of these change items can be associated to one another and to the Windchill parts and documents they affect. The process for completing this involves utilizing several roles over some period of time

Roles in the change process

There are several distinct roles within the change process.

Change Process Roles:

- Change Review Board
- Change Implementation Board
- Change Admin I
- Change Admin II
- Change Admin III



- The following are the out-of-the-box roles for the change process, but your administrators can easily configure the change process roles to match your organization's requirements:
- Change Review Board – Reviews and either approves, denies, or requests further investigation of a change request.
- Change Implementation Board – Reviews and either approves or denies the implementation plan included in the change notice.
- Change Admin I – The first change administrator screens the problem reports, reviews the change request, collects impact information, and communicates the decision to reject the change or implement it. Change Admin I also creates the change requests from unresolved problem reports.
- Change Admin II – The second change administrator is responsible for creating the implementation plan captured in the change notice.
- Change Admin III – The third change administrator acts as an auditor of all of the material related to a change, ensuring that all resulting documentation is clear, concise, and valid.

Specialization program in PLM - Windchill

Day 12 Change Implementation

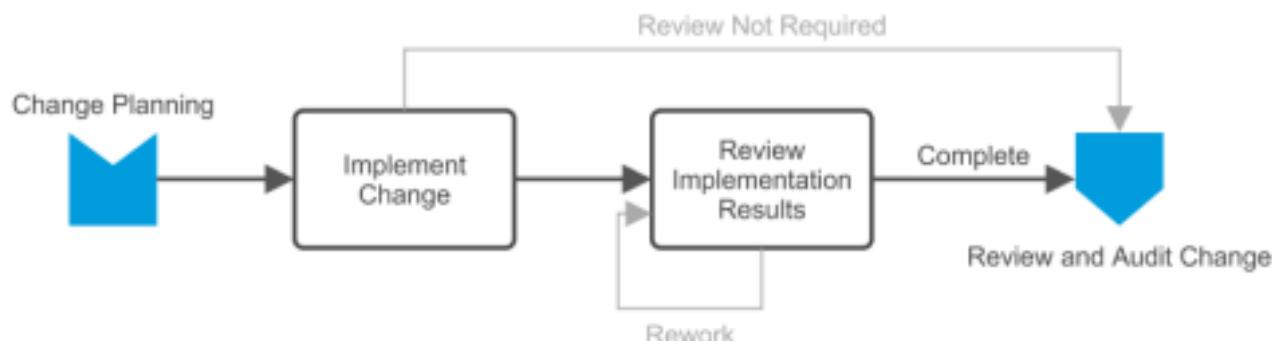
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Change Implementation Process Overview

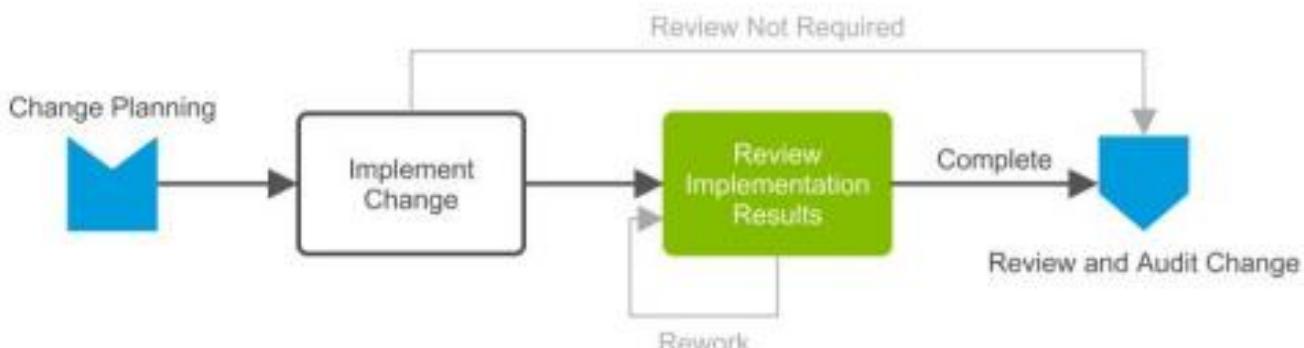
Change plan is delivered to those responsible for implementing the change.



- Whether the plan was routed through the Change Implementation Board in the full track or simply delivered to those responsible for implementing it during the change planning, the next step is to implement the plan.
- The process pictured in this diagram is performed by every individual who has been assigned to implement or review a line item task in the change notice's implementation plan. The assignee edits the product data, making the necessary modifications per the instructions from the change notice.



Implementing Change



Data creators edit product data as directed by the change notice tasks.

- Regardless of which route the change takes, when approval is given, the work to implement the change begins and the product structure, product model, and/or documentation is modified.
- Data creators edit product data as directed by the change notice tasks.
- Data users must then check and approve the work of the creators before the task is complete.
- All change implementation tasks contained within a change notice must be complete before the change notice can be considered resolved.



Reviewing Implementation Results

During the review of the implementation task, the reviewer:

- Ensures that the work done aligns with the intent of the originating change request.
- Reviews the work done from a content perspective.
- Documents any comments before routing to final auditing.
- Re-evaluates updates resulting from the routing of the change task back for rework

The reviewer ensures that the work done aligns with the intent of the originating change request



Review and Audit Change- Process Overview



- When all tasks associated with the change notice have been completed, the results are audited.
- When all the tasks associated with the change notice have been completed, Change Admin III audits the result to be sure it is clear, concise, and valid.
- Depending upon what the Change Admin III discovers, they either approve or reject the change notice.
- If the Change Admin III approves it, the object is released and the corresponding problem reports and change requests are considered resolved. With the object released, the change process is complete.



Actions ▾ Task - Change Notice Workflow_00021-Audit Change Notice

Details Related Set Up Participants

Attributes

General

Subject: Change Notice - 00021,Front Screw Housing Clearance

Instructions: The change notice has been modified.
1. Review the change notice displayed.
2. Enter comments in the Comments text field below.
3. Click Complete or Rework.
4. Click Complete Task to advance the change notice.

Special Instructions:

Process: Change Notice Workflow_00021-Front Screw Housing Clearance

Process Initiator: Taylor, Bob

Process Initiated On: 2013-08-28 14:19 EDT

Assignee: Brown, David

Role: Change Admin III

Priority: Highest

Deadline:

Status: Potential

Inputs

Comments:

Routing Options: Complete Rework

Change Implementation

```
graph LR; Start(( )) --> Audit[Audit Change Notice]; Audit -- Complete --> End([End]); Audit -- Rework --> Audit
```

- The Audit Change Notice is the final interactive step in the change process.
- The Audit Change Notice is the final interactive step in the change process. In this step, the CA III examines all of the objects and bill of material changes implemented by the change notice tasks and verifies that each part has a proper effectivity.

For changes that result in updates to systems outside of Windchill, ensure that this integration was executed accurately and completely.

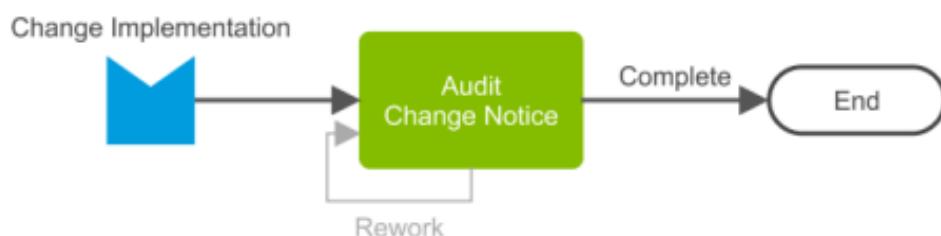
- Determine what systems were updated as a result of the change.
- Interrogate each system to ensure that content that was sent per the integration was accurately inserted into the system.
- There may be administrative logs in Windchill that logs the results of the integration, these should be reviewed.
- Repair any issues that resulted in a faulty integration.

Reviewing Business Rule Conflicts

Business rules are object validation tests.

The following list describes a few business rules:

- Checkout Rule
- Attribute Rule
- Release Target Rule
- BOM Release Rule



Actions ▾ Task - Change Notice Workflow_00782, A-Audit Change Notice

Details Related Set Up Participants

Attributes

General

Subject: Change Notice - 00782, A,CN 301

View Conflicts

Instructions: The change notice has been modified.

View Conflicts

Release Target Rule

2012-03-21 22:53:56.426, WPH2, ChangeOrganization, A.1
Error : No release target defined for the current state.

The following list describes a few business rules:

- Checkout Rule – Fails if objects are checked out.

- Attribute Rule – Fails if object attributes do not meet specified criteria.
- Release Target Rule – Succeeds if objects have change transition from current state to target state.
- BOM Release Rule – Verifies all first-level dependants in CAD structure are in an appropriate state to release.

Executing Ad Hoc Modifications

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Action items can be used any time in the change process to execute ad hoc work that needs to be tracked

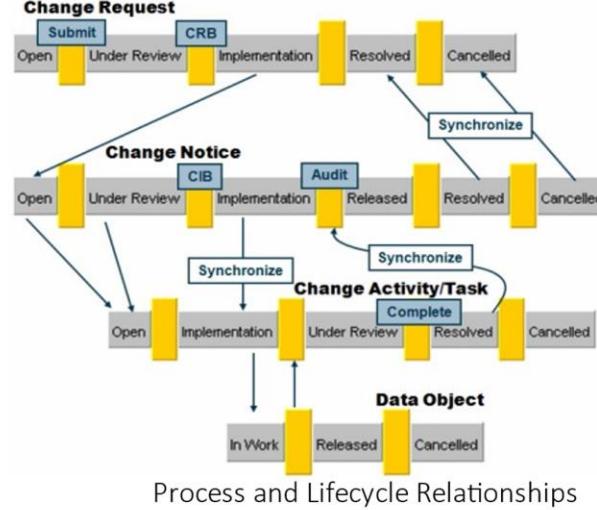
The screenshot shows the PTC Windchill interface for a Change Notice titled "ECN_Automotive - 00025". The "Action Items" tab is selected. A modal window titled "New Action Item" is open, showing the "Identify Action Item" and "Set Attributes" tabs. Under "Attributes", there is a table with columns "Name" and "Value". One row shows "Action Item Name" with the value "The length of Name of is not in specified range 1 and 20.". Another row shows "Owner" with the value "Nitesh Design Engineer, Nitesh (Nitesh: CM)". A third row shows "Description" with the value "Modify the Name of these objects 1)0000000056, Custom car seat for ProClip series,". The "Audit Change Notice" task is highlighted in the background.



- Action items can be used any time in the change process to execute ad hoc work that needs to be tracked.
- During the audit process of the change notice, if the Change Admin III discovers some discrepancies, which are required to be addressed, worked on, and tracked separately, an action item can be created and attached to the change notice.
- Action items can be used any time in the change process to execute ad hoc work that needs to be tracked. It is supported on all change objects, such as problem report, variance, change request, change notice and change notice task.

Completing the Change Process

All of the process elements work together to control and record information and decisions for the change process.



- A change request is submitted and reviewed. If the change request is full track, then an additional review is conducted by a Change Review Board (CRB). When the change request reaches the Implementation state, a change notice is created along with change notice tasks, detailing the required changes

Specialization program in PLM- Windchill

Day 13 Managing Your Work

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The screenshot shows the Windchill interface with three main sections:

- Tasks | Updates | Checked-Out Work**
- My Tasks**: Shows workflow tasks.

Name	Subject
Review Promotion Request	Promotion Request - 00121,...
Track Package	Package - 000002, A.1, Pack...
Approve Promotion Request	Promotion Request - 00003,...

(0 objects selected)
- Updates**: Shows recently modified objects.

Name	Number
measure	MEASURE
measure.prt	MEASURE.PRT
drill_810	DRILL_810
spark_plug_650.prt	SPARK_PLUG_650.PRT
gearbox_chuck_650.asm	GEARBOX_CHUCK_...

(0 objects selected)
- Checked-Out Work**: Shows checked-out items.

Name	Number
gearbox_chuck_650.asm	GEARBOX_CHUCK_650.ASM
chuck_650.prt	CHUCK_650.PRT
gearbox_front_650.prt	GEARBOX_FRONT_650.PRT

- Windchill provides tools that help you manage your work.
- For example, you can manage your tasks, access recently modified objects, and view checked-out items.
- Notebook enables you to store links to Web pages both within and outside of Windchill. You are able to subscribe to many types of objects within Windchill and receive an e-mail notification.
- When updates have been made. Moreover, Windchill enables you to collect useful information in the form of reports.



- The Tasks page provides tools to help you manage tasks for the context.
- The Tasks table enables you to view and sort the tasks for the context. To view the full list of your tasks from all contexts, you can also view the My Tasks table on the home page.



Products > Drill - 900 Series

Tasks Open

	Name	Subject	State	Owner
	Review Promotion Request	Promotion Request - 00084,chuck_900	Under Review	Smith, Fred
	Approve Promotion Request	Promotion Request - 00084,chuck_900	Under Review	Evans, John
	Track Package	Package - 000002, A	In Work	Jones, Mike
	Add Package Content	Package - 000002, A	In Work	Evans, John
	Review Promotion Request	Promotion Request - 00083,crank_900.asm	Under Review	Smith, Fred
	Approve Promotion Request	Promotion Request - 00083,crank_900.asm	Under Review	Evans, John

My Notebook

Actions

Name
My General Links
PTC Power Equ...
My Hot Links
DrillImage
drill_810

Select Subscribers

- The Notebook tool in Windchill enables you to store links to Web pages.
- Windchill Notebook:
 - Stores links to Web pages.
 - Add to Hot Links
 - Manually create links
- Uploads and stores files.
- Organizes your Notebook with folders



- Windchill is a communication and collaboration tool. Almost every feature enables you to share information and communicate that information to others.
- The subscription and notification tools automatically notify users about events in Windchill. Other tools help you reach other team members through e-mail, and can even provide links to Windchill pages to facilitate discussion.
- The discussion forum feature is another tool that helps you communicate about a product or an object. Similar to bulletin boards, discussion forums enable you to post topics, enabling everyone on the team to reply



Managing Subscriptions

- Windchill notifies object subscribers when an event occurs involving that object
- Subscriptions:
 - Content Objects
 - End Items
 - Parts
 - CAD Documents
 - Documents
 - Discussion Objects
 - Discussion Topics
 - Discussion Postings

Subscriptions My Subscriptions ▾				
X	■	Name	Events	Subject
□	□	trigger_900.prt	(i) Check Out/Check In	Drill - 900 Series
□	□	Air filter cover 900 Requirements	(i) Check Out/Check In Document Modified	Drill - 900 Series
(0 objects selected)				

- Windchill notifies you of changes to an object, document, or part if you subscribe to events that occur to that object.
- You are able to subscribe to many types of objects within Windchill.
- You can also subscribe to postings and topics within discussions.
- Though you create your subscriptions within the object you are working with, the Subscriptions page on your home page enables you to manage all of your subscriptions.

Creating Subscriptions

The screenshot shows the 'Subscribe' dialog box with the 'Identify Events' tab selected. The subscription name is 'trigger_900.prt' and the 'Subscribe to all versions' checkbox is checked. Under 'Types', there is a list of checkboxes for CAD Document events. Under 'Events', there is a list of checkboxes for various document lifecycle events. A dropdown menu for 'Life Cycle State' is set to 'All'. At the bottom, it says '* Indicates required fields.' and has 'Next' and 'Cancel' buttons.

To create a subscription, you must complete a three-step process.

- Specify the subscription name – You can accept the default name, which is also the object's name you are about to monitor. Next, select the events you want to send as notifications, such as a check in or a workflow state change. When the selected events occur, you are notified by an e-mail. Most events are self-explanatory, and for details of the events you can access Windchill Help.
- Select the subscribers – Subscribers are the people who receive the notifications. By default, your name is listed as the only subscriber. However, you can add other individuals or team members to

the list. You can also indicate from whom the subscription will come and the delivery details.

- Define the e-mail notification you receive – This includes defining the subject and body of the e-mail. If you leave the subject field blank, the subject is automatically completed by Windchill. You can type a Message that is the content of the e-mail.
- This informs the notification recipients why they are receiving the e-mail, and if there are any actions they are expected to take. You can then set the expiration date of the subscription, but you can also leave the expiration date field blank if you do not want it to expire

Holding Discussions

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- Discussion forums are message boards that enable you to collaborate with team members.
- Discussion Forums
 - Discussion forums are message boards for collaboration.
 - The context forum enables you to discuss topics that affect the entire Product or Library.
 - An object forum enables you to discuss a specific object.
- Discussion Features
- Topics have:
 - Participants
 - Comments
- Comments have:
 - Attachments
 - Links
- Subscriptions

Discussions Tree View			
		Topics/Comments	action subscribe Created By
		Contract	wcadmin
		ACME Carburetors acquisition of DE Carburetors	Smith, Fred
		I was just informed that DE Carburetors was acquired by ACME Carburetors.	
		RE:ACME Carburetors acquisition of DE Carburetors	Evans, John
		Thanks for the information.	
		Design	wcadmin
		Prototype	wcadmin
		Specification	wcadmin
		Testing	wcadmin

- Discussion forums are message boards that enable you to collaborate with team members. There are two different types of discussion forums:
- A context forum is not specific to an object, but to a context, such as a product or project. The context forum is available from the Discussions page of the context.
- An object forum is associated with a specific Windchill object, such as a CAD document. You can access this discussion several ways:
- Create a custom tab for the details page, by adding the Collaboration > Discussions table.

- Access an existing discussion using the Discussions icon for that object on the Folders page.
- If you are a participant in the discussion, you can access it from the Discussions table of your home page. You may need to add the Discussions table to your home page by clicking the Customize button. A discussion forum has a specific structure.
- The E-mail and E-mail Team actions enable you to compose an e-mail to all or some of the context team. To e-mail only certain members of the team, select your recipients in the Members table and use the E-mail action. In both cases, a window appears with the selected members or all members listed in the Recipients table.
- From the window, you can compose a message to the recipients and click OK to send the message. Clicking the Use my own Mail Client link opens a new e-mail in your installed mail client, such as Microsoft Outlook, and populates the subject of the message.

E-mailing Team Members

E-mail Team

- A method for communicating information to other team members is to e-mail a Windchill page. To e-mail a page, click Quick Links, and then click the Email Page link that appears in the drop-down menu at the top of any page.
- Clicking the link opens a window that enables you to select the groups or users to receive the currently active page. You can also add an e-mail subject and body text. Once the notification is complete, an e-mail with a link to the Windchill page is sent to the recipients you selected.

Specialization program in PLM - Windchill

Day 14 - Participating in Processes



Participating in Processes

- Windchill uses a business process system to manage object development, control object maturity, and assign task:
- Business Process System
 - Manages object development.
 - Promotes objects.
 - Assigns tasking
- Using My Tasks Table
 - Complete tasks.
 - Reassign tasks to others.
 - Accept or decline tasks.

The screenshot shows a software interface titled "Tasks | Updates | Checked-Out Work". A dropdown menu is open, showing "My Tasks" and "Open". Below the menu is a toolbar with icons for file operations like New, Open, Save, and Print, along with a search bar and an "Actions" dropdown. The main area is a table titled "My Tasks" with columns for Name and Subject. There are three rows of data, each representing a "Review Promotion Request" task. The first two rows have a small thumbnail icon next to the name, while the third row has a larger thumbnail icon. Each row also includes an "Info" icon (a blue circle with an 'i') and a "Details" icon (a blue square with a white gear).

	Name		Subject
	Review Promotion Request	(info)	Promotion Request - 00141, fuel_tank_900.asm
	Review Promotion Request	(info)	Promotion Request - 00084, chuck_900
	Review Promotion Request	(info)	Promotion Request - 00083, crank_900.asm

- Windchill uses a business process system to manage object development, control object maturity, and assign tasking. When the system assigns you a task, the task is delivered to the My Tasks table. From the My Tasks table, you can complete a task, reassign it to another user, and accept or decline a task.



Completing Tasks

- If you are identified as a participant in a promotion process, you receive a task in your My Tasks table.
- Steps required to complete the task:
 - View and open the task.
 - Follow instructions.
 - Click Complete Task .

This is a 'Set Up Participants' task. Use the Set Up Participants tab to assign participants to different roles used in this process.

Actions Task - Gearbox Assembly 900 Requirements1307132393906-Review Promotion

Details Related Set Up Participants

Attributes

Save Complete Task

General

Subject: Promotion Request - 00121, Gearbox Assembly 900 Requirements

Instructions: You have been selected as a reviewer for this promotion request.
1. Review the object displayed.
2. Review the promotion request.
3. Enter comments in the Comments text field below.
4. Click Complete Task to advance the promotion request.

Process: Gearbox Assembly 900 Requirements1307132393906-Gearbox Assembly 900 Requirements

Process Initiator: Jones, Mike

Process Initiated On: 2011-06-03 16:19 EDT

Assignee: Smith, Fred

Role: Reviewer

Priority: Highest

Deadline:

Status: Potential

Inputs

Comments:

- If you are identified as a participant in a promotion process, you receive a task in your My Tasks table.
- The basic steps for completing any task involve viewing the task, following the instructions, and then clicking the Complete Task button.
- Once the task is complete, it is removed from your assignments list and the process automatically routes the object to the next user or users in the process. Completing some tasks changes the state of the subject object or objects.



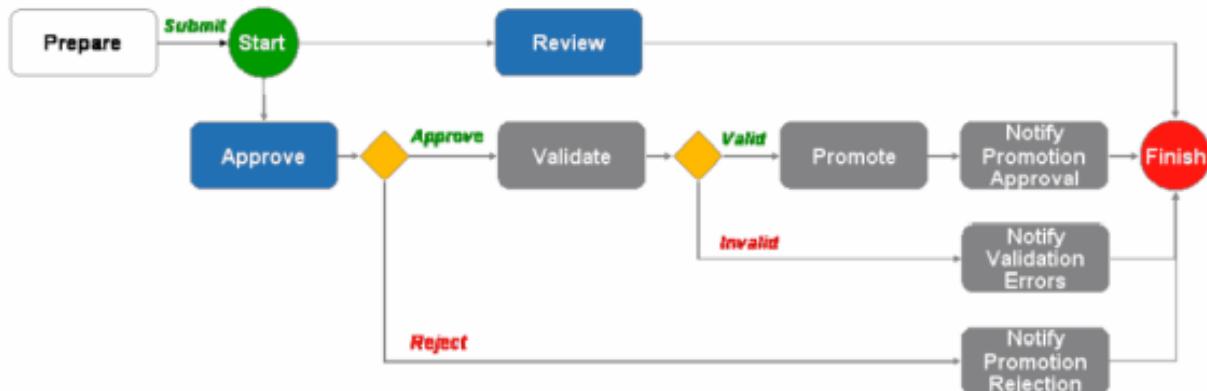
Inputs

Comments:

Routing Options: Approve Reject

Save **Complete Task**

- To complete a promotion task, you must first access the task by clicking its name in your My Tasks table. On the task page, you read the instructions to gain a full understanding of what is required to complete the task.
- Next, review the promotion objects in the Promotion Objects table. This may include viewing the details and content of the promotion objects.
- Also, review the promotion process information located under the Process tab. Finally, you complete the task by typing in comments, selecting a disposition such as Approve or Reject, and clicking the Complete Task button.



Actions ▾  Task - Gearbox Assembly 900 Requirements1307132393906-Approve

Details **Related** **Set Up Participants** 

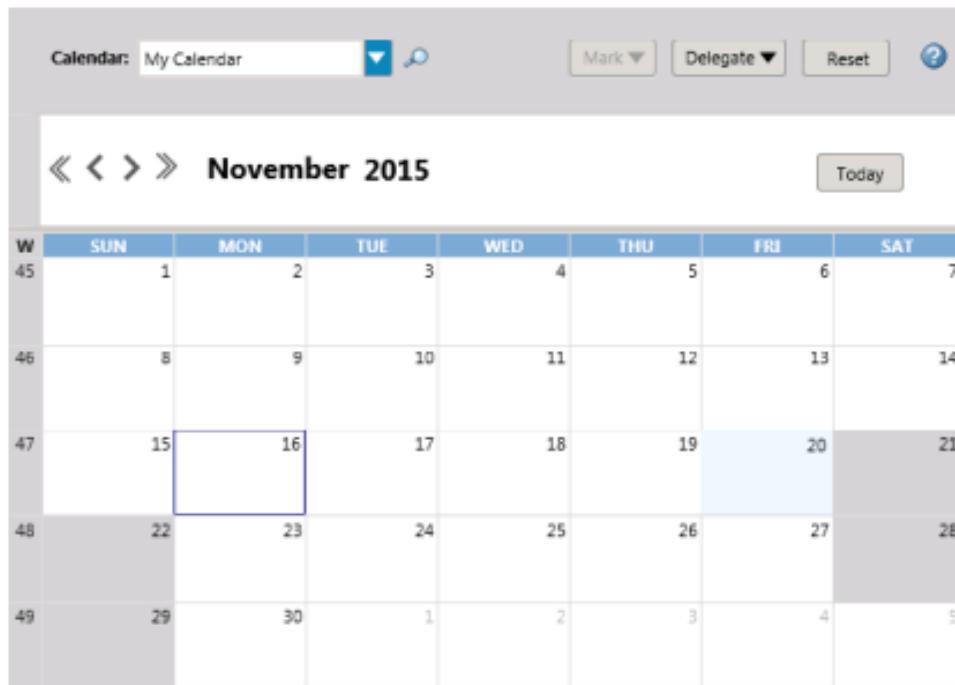
Attributes

General

Subject:	 Promotion Request - 00121, Gearbox Assembly 900 Requirements
Instructions:	<p>You have been selected as an approver for this promotion request.</p> <ol style="list-style-type: none"> 1. Review the object displayed. 2. Review the promotion request. 3. Enter comments in the Comments text field below. 4. Click Approve or Reject. 5. Enter comments in the Comments text field below. 6. Click Complete Task to advance the promotion request.
Process:	 Gearbox Assembly 900 Requirements1307132393906-Gearbox Assembly 900 Requirements
Process Initiator:	Jones, Mike

A red arrow points from the text "Windchill enables you to reassign workflow tasks to other users by using the task's Actions menu or the icon." to the "View Process Image" icon in the task details header.

- From the task details page, you access the workflow diagram by clicking the View Lightweight Process Image icon. The icon is located to the right of the Process label.
- This diagram provides details such as life cycle name and the related workflow activities.
- Windchill enables you to reassign workflow tasks to other users by using the task's Actions menu or the icon.
- In the Reassign Tasks window, you select a person to whom you want the task reassigned and a reason for the reassignment.



- You can use the Windchill calendar to view your own calendar, other users' calendars, or the System calendar. Each calendar displays a work schedule indicating workdays and non-work days. This schedule is used for assigning tasks that have deadlines. Workdays are displayed in white and non-work days are shaded. Calendar functionality depends on which calendar you are viewing.
- When viewing your calendar, you can highlight a day or consecutive days and click Reset to cancel any of your changes or delegations. You can mark days as non-working, or automatically delegate tasks that are received on certain days.
- To mark a day as non-working, select the day or days and then select Mark as Nonworking from the Mark drop-down menu. Tasks can have deadlines that are relative.
- If you mark a day non-working, the task does not count days that are marked as non-working against your allotted time.
- To delegate tasks, select the day or days that you wish to delegate tasks, and then select the Delegate selected days... or Delegate indefinitely... from the Delegate drop-down menu. You can then search for and select the user that receives tasks delivered to you during those day.