



GETTING THE MOST OUT OF PTC CREO AND PTC WINDCHILL 11

Steve Shaw

PLM Solution Management Director

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A decorative graphic in the top-left corner consisting of several overlapping, pointed shapes in shades of pink, yellow, and blue.

AGENDA

- What's New with Creo support
- CAD to eBOM productivity gains

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- CAD to eBOM productivity gains

CREO - WINDCHILL INTEROPERABILITY



- Windchill 11.0 F000+
 - Creo 2.0 M190 and above
 - Creo 3.0 M010 and above
- Creo 4.0 support
 - 10.2 M030 CPS20 and higher (Nov 2016)
 - General Creo support
 - 11.0 M020 (Berlin) and higher (Dec 2016)
 - Support for new Creo 4 features

PTC Creo Data Management Compatibility Roadmap
with PTC Windchill (Pro/INTRALINK², PDMLink, and ProjectLink)

	PTC Windchill 8.0 Click here for details	PTC Windchill 9.0 Click here for details	PTC Windchill 9.1 Click here for details	PTC Windchill 10.0 Click here for details	PTC Windchill 10.1 Click here for details	PTC Windchill 10.2 Click here for details	PTC Windchill 11.0 Click here for details
Pro/E Wildfire 4.0	Compatible	Compatible	Compatible	Compatible with Wildfire 4.0 M190	Not Compatible	Not Compatible	Not Compatible
Creo Elements/Pro 5.0 (Pro/E Wildfire 5.0)	Not Compatible	Not Compatible	Compatible with Wildfire 5.0 F000	Compatible with Wildfire 5.0 F000 & 10.0 M030 & 10.0 M010	Compatible with Creo Elements/Pro 5.0 M130	Compatible with Creo Elements/Pro 5.0 M200 & 10.2 F000	Not Compatible
PTC Creo 1.0 ¹	Not Compatible	Not Compatible	Compatible with 9.1 M060	Compatible with 10.0 M010	Not Compatible	Not Compatible	Not Compatible
PTC Creo 2.0 ¹	Not Compatible	Not Compatible	Compatible with PTC Creo 2.0 F000 & 9.1 M070 and PTC Creo 2.0 M050 & 9.1 M060	Compatible with PTC Creo 2.0 F000 & 10.0 M030	Compatible with PTC Creo 2.0 F000	Compatible with PTC Creo 2.0 M070 & 10.2 F000	Compatibility Planned with PTC Creo 2.0 M190 & 11.0 F000
PTC Creo 3.0 ¹	Not Compatible	Not Compatible	Not Compatible	Not Compatible	Compatibility with PTC Creo 3.0 F000 & 10.1 M040	Compatible with PTC Creo 3.0 F000 & 10.2 M020	Compatibility Planned with PTC Creo 3.0 M070 & 11.0 F000
PTC Creo 4.0 ¹	Not Compatible	Not Compatible	Not Compatible	Not Compatible	Not Compatible	Compatibility Planned with PTC Creo 4.0 F000 & 10.2 M030 CPS20	Compatibility Planned with PTC Creo 4.0 F000 & 11.0 M020

¹ For information related to specific PTC Creo App support refer to page 2 in this document.
² Pro/INTRALINK does not support the following PTC Creo Applications: PTC Creo Sketch, Illustrate or Options Modeler.

This information may be out of date. Please check [here](#) for the latest version.
 Forward looking information subject to change without notice © 2015 PTC

- Note: Creo 4.0 Sneak Peek is not supported

PTC CREO DATA MANAGEMENT COMPATIBILITY ROADMAP

WITH PTC WINDCHILL (PRO/INTRALINK², PDMLINK, AND PROJECTLINK)



	PTC Windchill 8.0 Click here for details	PTC Windchill 9.0 Click here for details	PTC Windchill 9.1 Click here for details	PTC Windchill 10.0 Click here for details	PTC Windchill 10.1 Click here for details	PTC Windchill 10.2 Click here for details	PTC Windchill 11.0 Click here for details
Pro/E Wildfire 4.0	Compatible	Compatible	Compatible	Compatible with Wildfire 4.0 M190	Not Compatible	Not Compatible	Not Compatible
Creo Elements/Pro 5.0 (Pro/E Wildfire 5.0)	Not Compatible	Not Compatible	Compatible with Wildfire 5.0 F000	Compatible Creo Elements/Pro 5.0 M080 & 10.0 F000 Creo Elements/Pro 5.0 M090+ & 10.0 M010	Compatible with Creo Elements/Pro 5.0 M130	Compatible with Creo Elements/Pro 5.0 M200 & 10.2 F000	Not Compatible
PTC Creo 1.0 ¹	Not Compatible	Not Compatible	Compatible with 9.1 M060	Compatible with 10.0 M010	Not Compatible	Not Compatible	Not Compatible
PTC Creo 2.0 ¹	Not Compatible	Not Compatible	Compatible PTC Creo 2.0 F000 & 9.1 M070 and PTC Creo 2.0 M050 & 9.1 M060	Compatible PTC Creo 2.0 F000 & 10.0 M030	Compatible with PTC Creo 2.0 F000	Compatible with PTC Creo 2.0 M070 & 10.2 F000	Compatibility Planned with PTC Creo 2.0 M190 & 11.0 F000
PTC Creo 3.0 ¹	Not Compatible	Not Compatible	Not Compatible	Not Compatible	Compatibility with PTC Creo 3.0 F000 & 10.1 M040	Compatible with PTC Creo 3.0 F000 & 10.2 M020	Compatibility Planned with PTC Creo 3.0 M070 & 11.0 F000
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WINDCHILL 11 - FEATURED OBJECTS



- Challenge
 - Large workspace listings can be a drag on performance
 - Gaining users focus on the important information in the users workspace
 - Existing Featured Object rules are hardcoded
- Capabilities
 - New Featured Object Status column displays which items are “featured”
 - Identify featured items with starred icon
 - Sort on column
 - New Manual rule
 - In addition to existing hardcoded rule
 - Users can manually add items of interest to their featured object list
- Benefit
 - Increased awareness of feature
 - End user control of “what is featured”
 - Improve working with large workspaces

Workspace: demo - Pick an Action -

Object List As a list All (31 of 31 objects)

File Edit

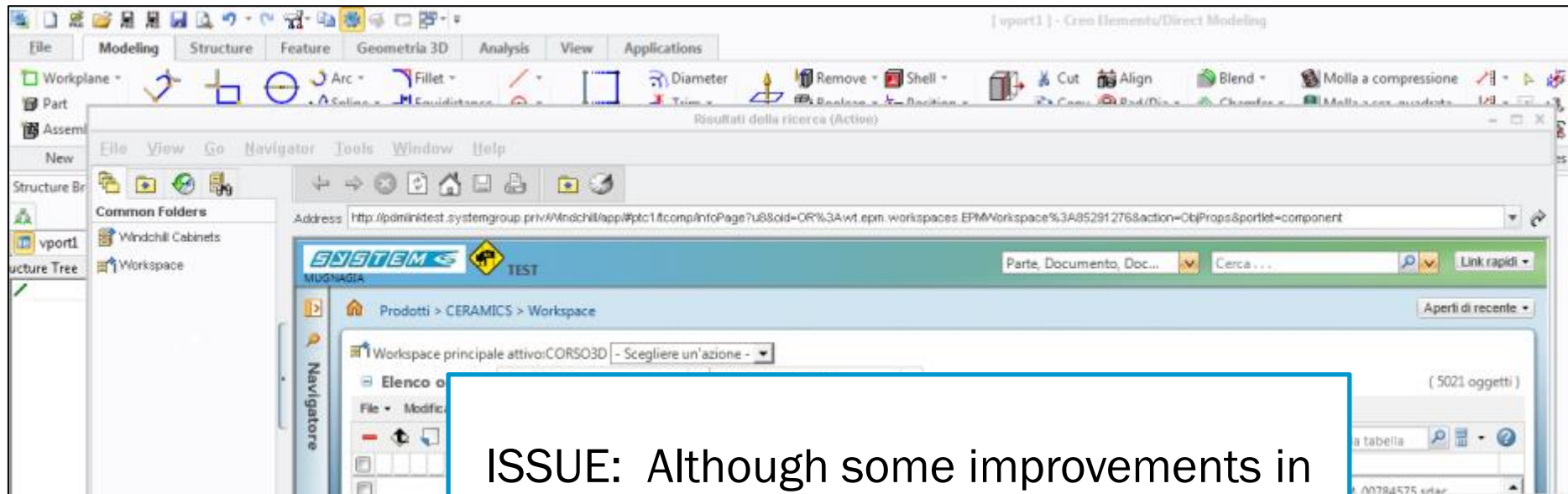
		Number	File Name	Actions	Version	L
☐	★	0000000443	0000000443.DRW	0000000443.drw	A.1	2
☐	★	CAGE_SHAFT	CAGE_SHAFT.PRT	cage_shaft.prt	A.1	2
☐	★	0000000221			A.1 (D...	2
☐	★	INNER_CAGE-XX	INNER_CAGE-XX.PRT	inner_cage-xx.prt	A.1	2
☐	★	0000000203			A.1 (D...	2
☐	★	0000000225			A.1 (D...	2
☐	★	LOWER_PULLEY_ASSE...	LOWER_PULLEY_ASSE...	lower_pulley_assembly...	A.1	2
☐	★	0000000222			A.1 (D...	2
☐	★	LOWER_PULLEY	LOWER_PULLEY.PRT	lower_pulley.prt	A.1	2
☐	★	OUTER_CAGE_INSERT...	OUTER_CAGE_INSERT...	outer_cage_insert.asm	A.1	2
☐	★	0000000209			A.1 (D...	2
☐	★	0000000232			A.1 (D...	2

- New show top-level only filter planned for 11.0 M020!
 - Show top-level assemblies
 - Show parts with no parents in current workspace

WINDCHILL 11 - LARGE WORKSPACE PERFORMANCE



More and more customers are working in WSs with 5000 – 60,000 objects



ISSUE: Although some improvements in 10.2, single object actions in large workspaces are still taking longer than single object actions in small workspaces

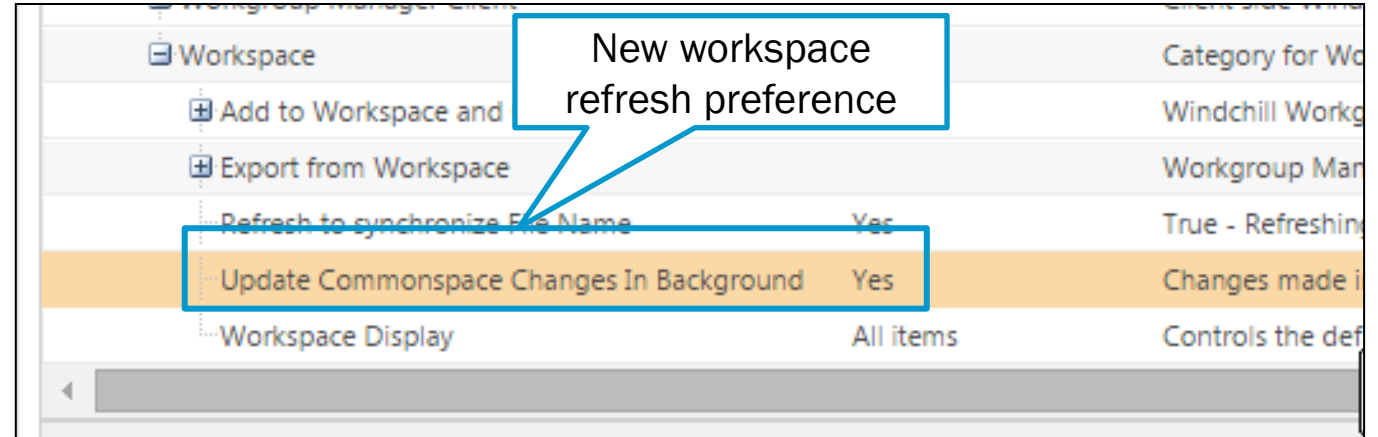
WINDCHILL 11 - LARGE WORKSPACE PERFORMANCE IMPROVEMENTS



30-60%*

Improved performance for single object actions in the workspace

* comparing Windchill 11.0 w/ 10.2 M030 using Creo 3.0 having 20K objects in workspace



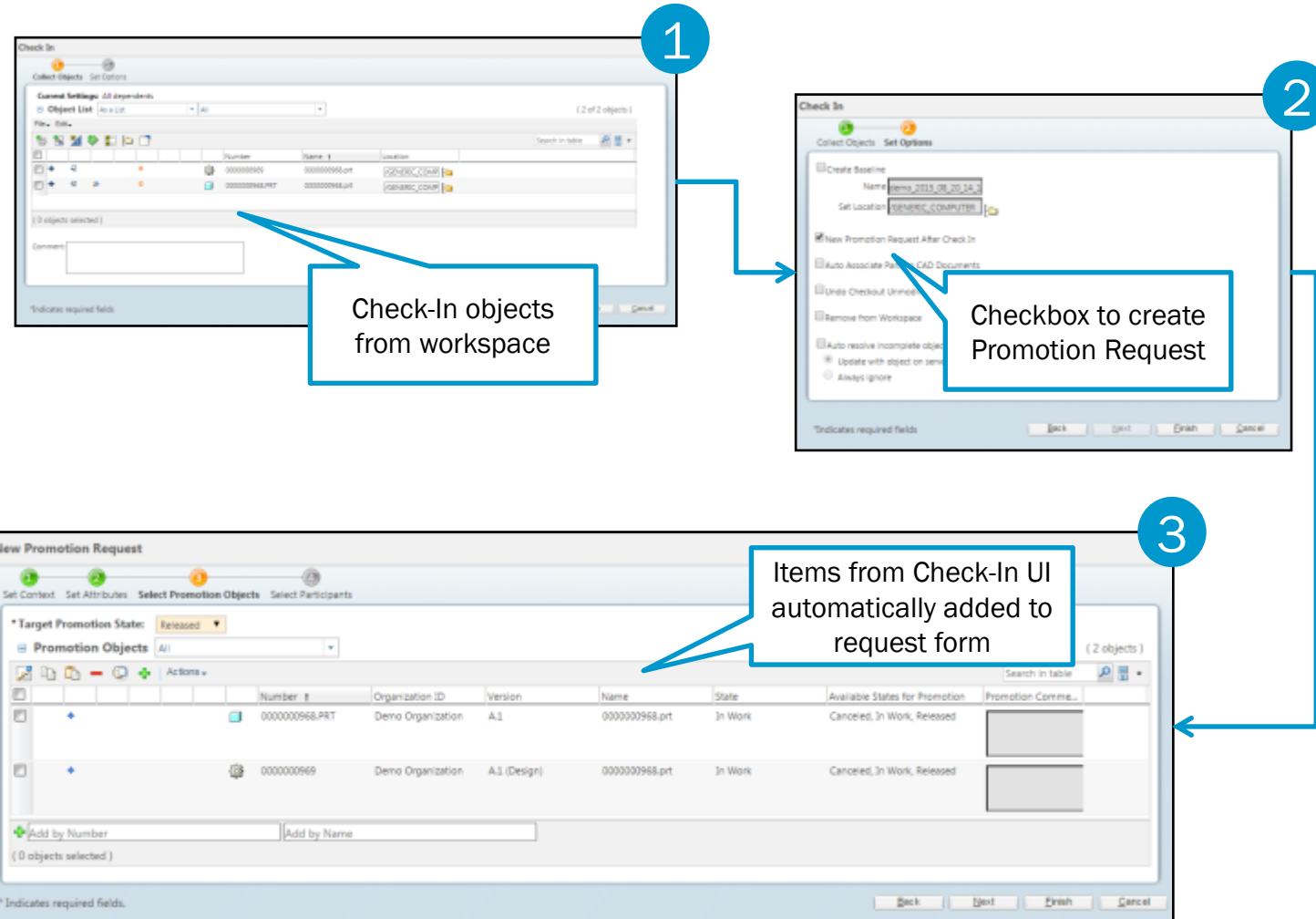
- Keeping workspaces up-to-date has profound influence on overall performance as workspace sizes grow
- New preference refreshes workspace with the server in the background

WINDCHILL 11 - PROMOTE ON CHECK-IN



Create Promotion Request during check-in process

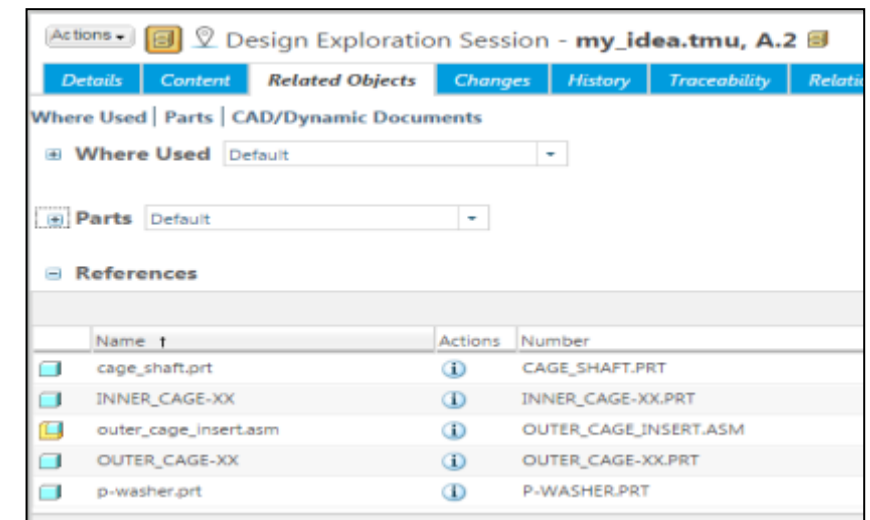
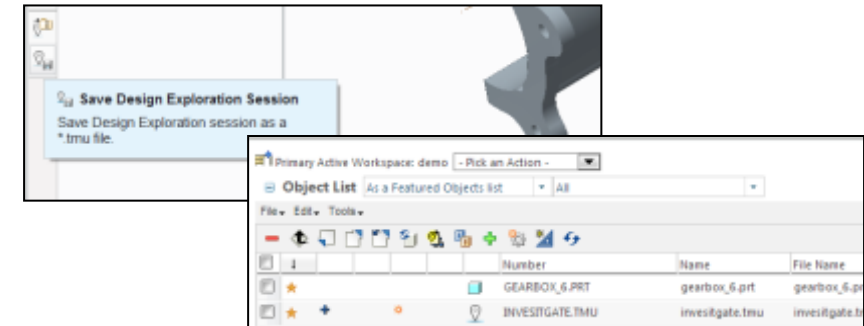
- Automatically launches Create New Promotion Request form
- Items from check-in form automatically added to Promotion Objects list



WINDCHILL 11 - CREO SUPPORT IMPROVEMENTS - DEX



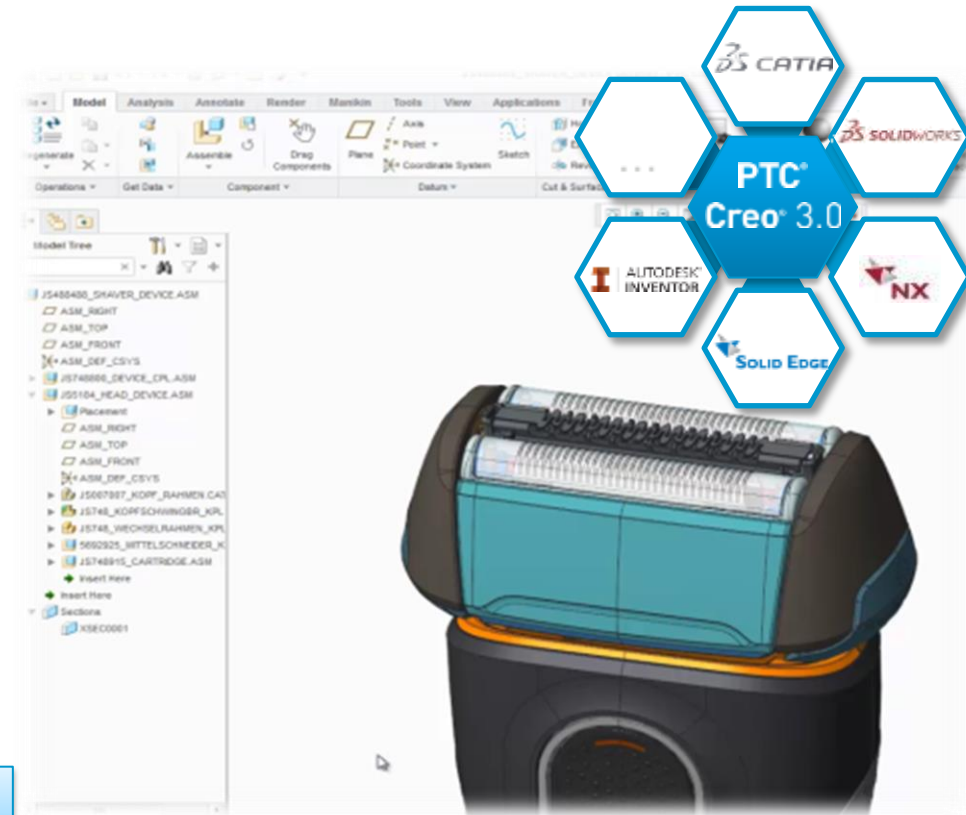
- True support for Creo Design Exploration Extension
- Benefits
 - Eliminates need for individual user to manage different ideas in multiple workspaces or creating unnecessary copies
 - Check-in and share DEX file (tmu) with other users
 - Avoid creation of unwanted/unnecessary versions/iterations of CAD data creating clutter and uncertainty
- Capabilities
 - Native support for TMU files
 - Check-in TMU as regular Primary Object
 - Recognized by Windchill as Design Exploration Session
 - Related models and required versions will be registered in TMU and linked as Reference objects



WINDCHILL 11 - CREO SUPPORT IMPROVEMENTS - UNITE



- Usability and Process Improvements for Creo Unite with Windchill 11
- Streamline 'check in' of Creo assemblies referencing non-Creo data which user has no PDM access to (e.g. Released)
 - .creo wrapper will be added as attachment based on preference (Site/Organization)
 - Allow system changes on CAD Documents = <No/Yes>
- Creation of External Simplified Representations (ESR) for Creo Unite Assemblies referencing NX, SolidWorks or CATIA data is now supported
 - Windchill, Creo and Creo view have consistent and stable IDs for working with heterogeneous CAD data
- Coming with Creo 4.0 support in 11.0 M020
 - Support for check-in of non-Creo formats directly from Creo 4.0
 - Maintaining originating authoring application

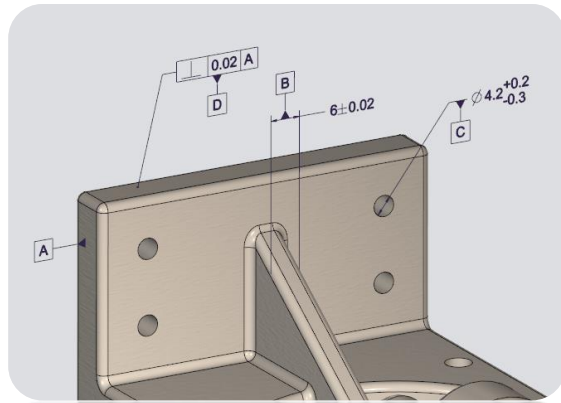


Note: Refer to the [CS188975 - PTC Creo Unite Technology interaction with Windchill PDMLink 10.2: Information, Support Detail, Implications and Caveats](#) for latest details

WINDCHILL 11 - CREO 4.0 SUPPORT

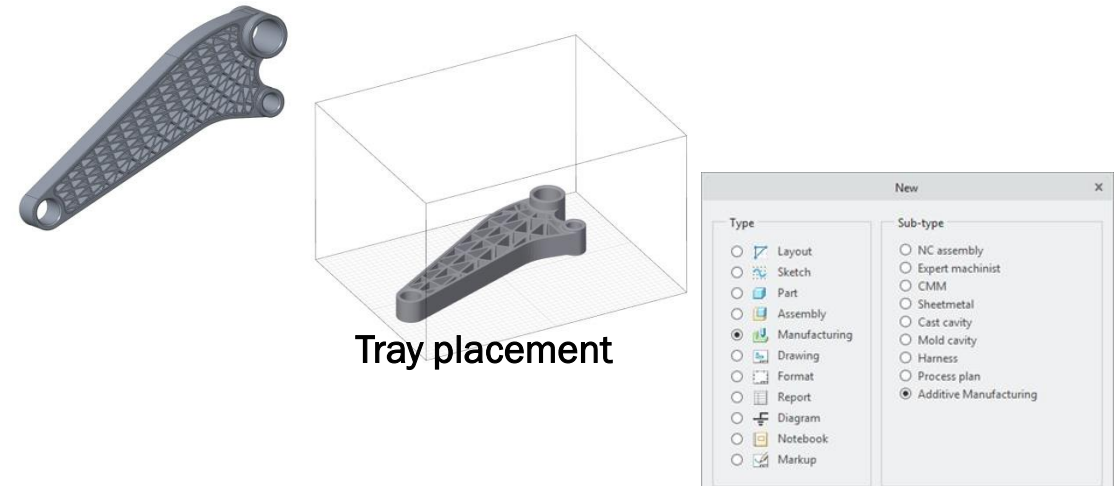


- GTOL Improvements Support
 - Creo 4.0 updates to support latest GD&T standards



- Updated Model Item support
 - Additional parameter support for capturing updated standards introduced by Creo 4.0
- Facilitates downstream usage by MPMLink for process allocation

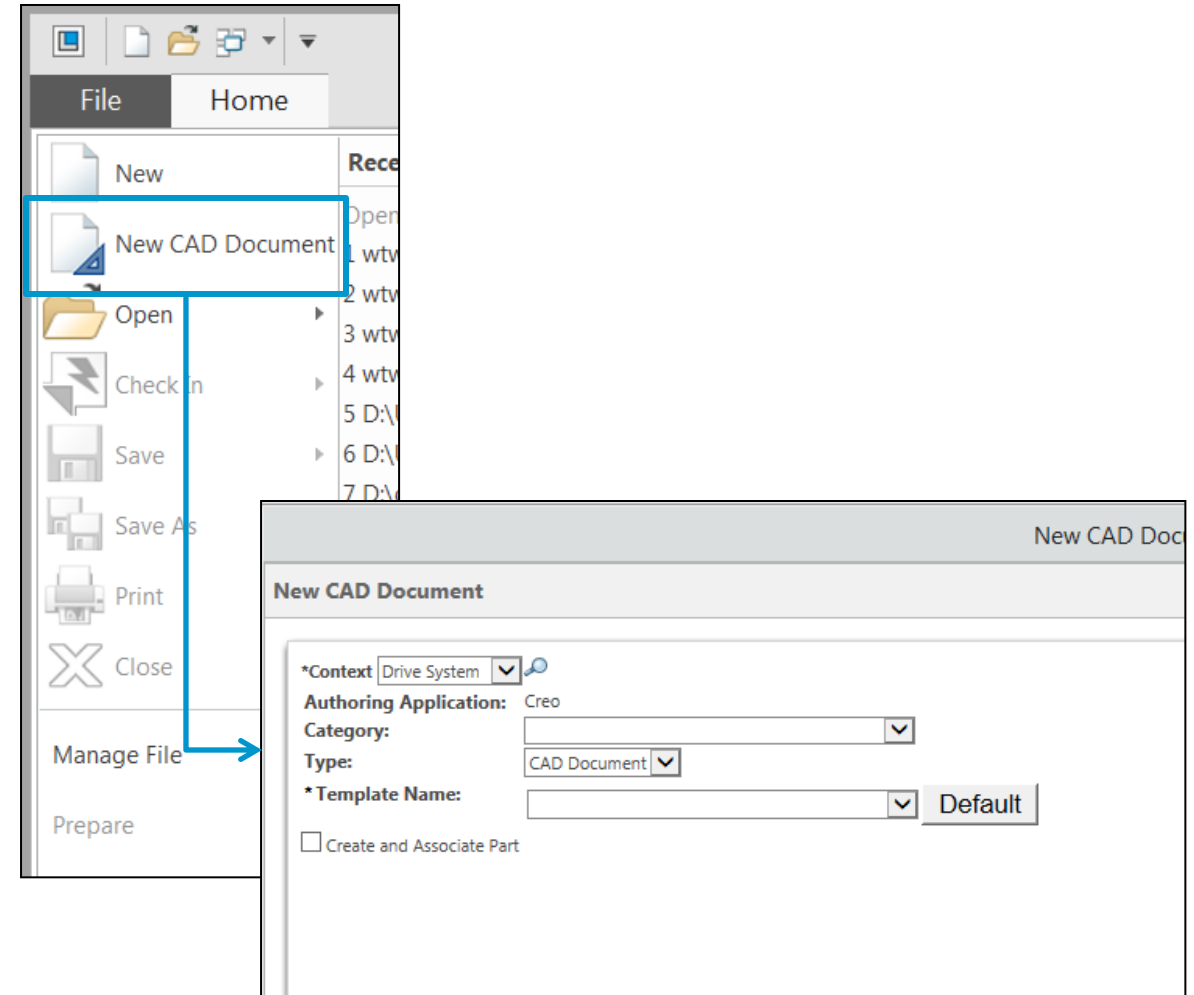
- Additive Manufacturing Support



- Support for new Tray Assembly
 - Open in/Open With support from Windchill Info pages
 - Recognition and proper categorization of this assembly subtype
 - Exclude Owner/Image auto-association to part by default

CREATE NEW CAD DOCUMENT FROM CREO 4.0

- New CAD Document available from Creo File menu
 - Launches Windchill New CAD Document UI
 - In-line with other WWGM integrated Cad tools
- Benefits:
 - Allows for Type selection
 - Provides the ability to enter Windchill attribute information
 - Follow Naming & Numbering rules defined by Windchill
 - Enables concurrent Part creation
- Planned for Creo 4.0 & 11.0 M020



A decorative graphic in the top-left corner consisting of several overlapping, pointed shapes in shades of blue, yellow, and red.

AGENDA

- What's New with Creo support
- CAD to eBOM productivity gains

REMOVING THE FRICTION



Decorative geometric shapes in green, blue, yellow, and pink are scattered across the slide, primarily on the left side. A large green triangle points downwards from the top left. A blue triangle points upwards from the bottom left. A yellow triangle points downwards from the bottom left. A pink triangle points upwards from the bottom left.

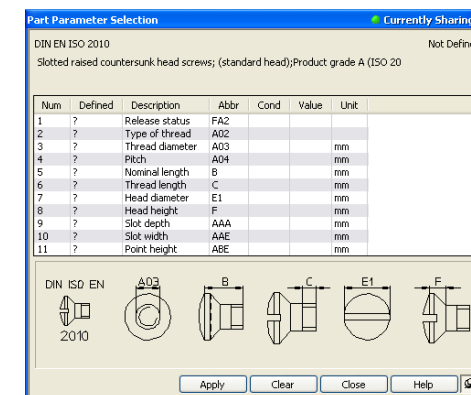
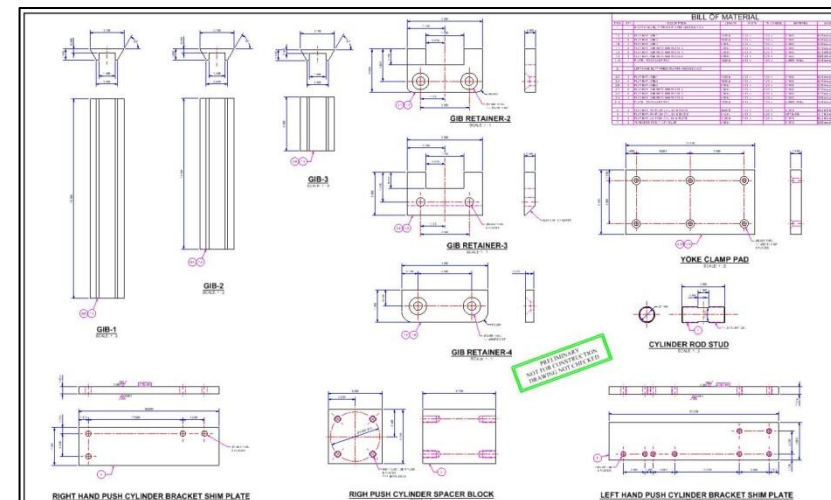
REMOVING THE FRICTION

- Naming & Numbering Synchronization
- Association Improvements
- Build Flexibility

IMPROVED SUPPORT FOR NAMING AND NUMBERING SYNCHRONIZATION BETWEEN CAD DOCUMENTS AND PARTS



1. It is easier to make the 2D drawing correctly reflect the Part numbers, when CAD Documents have the same base number.
2. When files are sent to suppliers, OEMs or collaborators, this is often done outside of Windchill; therefore the file names are used to show relationships to the Part.
3. Some companies and government agencies require files created by suppliers to have this policy.
4. Reuse is easier if CAD Documents can be found using the Part number since search in CAD specific tools only searches on file name.



CREATE NEW – “TOP DOWN”



Changes to New Part and CAD Document Number generation

New Part

Set Attributes Set Security Labels New CAD Document

Product: MAN_PDM_1

* Type: Part

* Create as End Item: No

Part Attributes

Number: (Generated)

* Name: GetServerAssigned.Generated Number

* Assembly Mode: Separable

* Source: Make

View: Design

* Default Trace Code: Untraced

* Default Unit: each

* Gathering Part: No

* Phantom Manufacturing Part: No

* Life Cycle Template: (Generated)

Team Template: (Generated)

* Location: Autoselect Folder (/MAN_PDM_1) Select Folder (/MAN_PDM_1)

Create CAD Document

Keep checked out after checkin

New Part takes the generated number

New Part

Set Attributes Set Security Labels New CAD Document

* Context: MAN_PDM_1

Authoring Application: Creo

Category: CAD Part

Type: CAD Document

* Template Name: ptc_wm_mmns_part... Default

Attributes

Number: Same As Part Number (Generated)

* Name: GetServerAssigned.Generated Number.prt

* File Name: Same As Number

* Revision: A

Description:

* Location: Autoselect Folder (/MAN_PDM_1) Select Folder (/MAN_PDM_1)

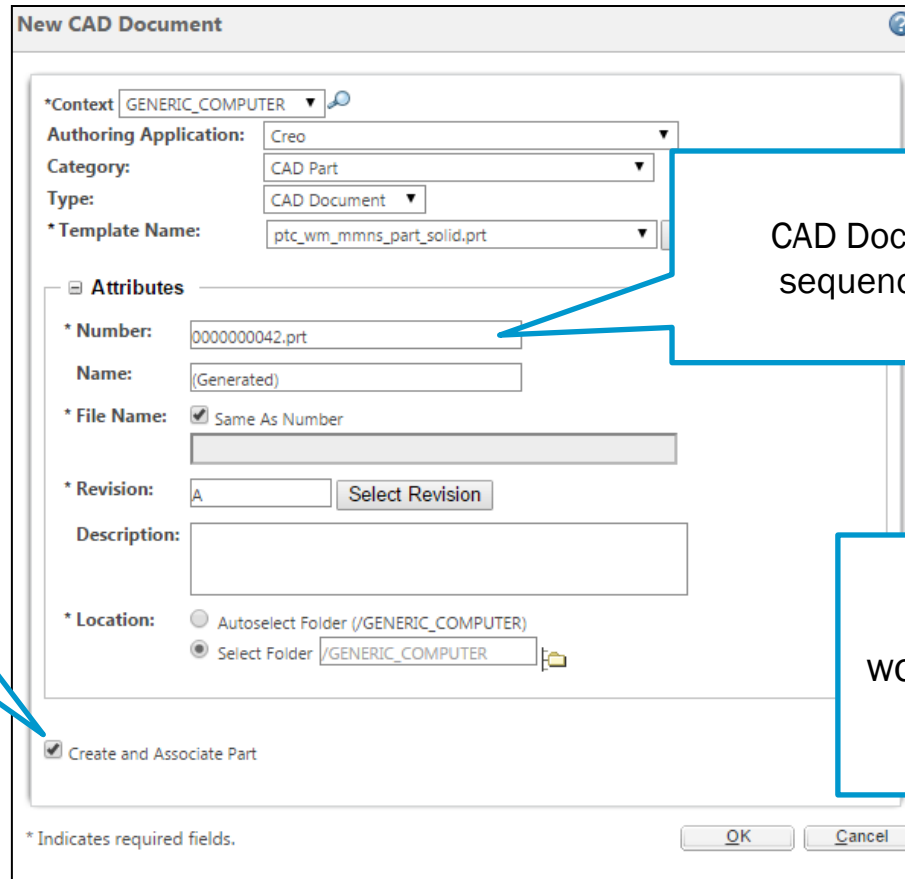
Preference controlled with Windchill 11.0 M010!

CAD Document reuses Part Number + appends extension!

Note: In 10.2, CAD Docs created with this method would have a different number without extension.

CREATE NEW – “BOTTOM UP”

Changes to New CAD Document and Auto Associate and Create Parts



New CAD Document

*Context: GENERIC_COMPUTER

Authoring Application: Creo

Category: CAD Part

Type: CAD Document

*Template Name: ptc_wm_mmns_part_solid.prt

Attributes

* Number: 0000000042.prt

Name: (Generated)

* File Name: Same As Number

* Revision: A

Description:

* Location: Autoselect Folder (/GENERIC_COMPUTER) Select Folder (/GENERIC_COMPUTER)

Create and Associate Part

* Indicates required fields.

Part will use same base number as CAD Document, even if configured to generate number by default

CAD Document takes number from sequence and appends extension

Note: In 10.2, CAD Docs created with this method would not have extension and the Parts would have the **next** number in the sequence.

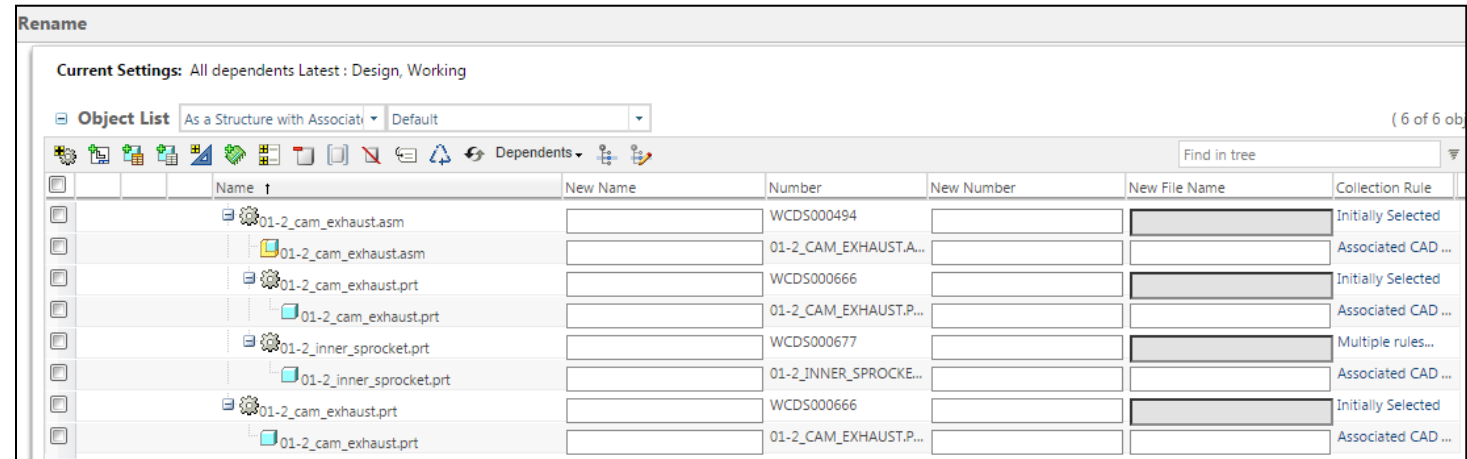
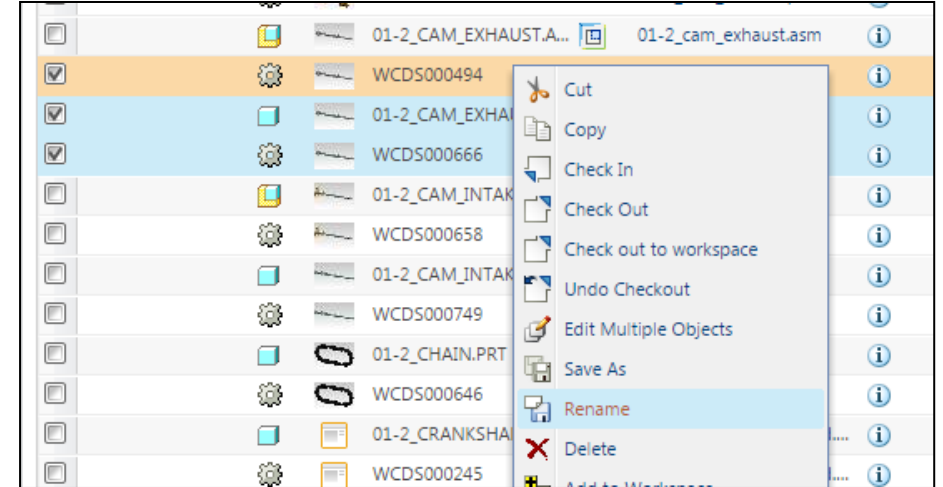
RENAME IMPROVEMENTS



- Common Use Cases requiring rename
 - DoD contractors often get the final Part number after CAD Docs and Parts have been created and need to rename to number provided by government.
 - CAD Documents could have been created with temporary numbers/names and then want inherit their related Part's number
- Customer Challenges
 - Need to rename groups of related and unrelated objects at the same time
 - Desire to keep naming and numbering conventions the same between Parts and their related CAD documents

RENAME IMPROVEMENTS

- Multi-object select from Folder and Search results tables
 - Includes support for combinations of Parts and CAD documents
- Collection Support
 - Leveraging workspace Rename paradigm from Common Space
 - Further collect related CAD documents and Parts



RENAME - SETTING NEW NAME



Set New Name

New Number:

- Unchanged
- Autogenerate
- Generate document number based on related Part
- Change

From: To:

New Name:

- Unchanged
- Same as number
- Change

From: To:

New File Name:

- Unchanged
- Same as number
- Change

From: To:

OK Cancel

1. Parts can now take a new number from the sequence
 - Past releases did not support number generation during rename, could only support manual entry
 - In 11.0 – capable of pulling from sequence
 2. CAD Documents can have same base number as Part
 - 1st set Part numbers in rename
 - 2nd select CAD documents and “generate from related part” option
 - CAD Documents number generated by applying the Regular Expression preference to find the string to replace
- Similar Name & Number synchronization during Save As operation planned for Windchill 11.0 M020!

CREATE PART – FLEXIBLE COMPONENTS

CAD models in different states for the same Part (position and/or geometry)

Top.asm

- (pos= -1,0,0)→
 - (pos=0,0,0)→
 - (pos=0,0,0)→
 - (pos=0,0,0)→
- (pos= 1,0,0)→
 - (pos=0,0,0)→
 - (pos=0,1,0)→
 - (pos=0,0,0)→
- (pos= 0,0,0)→
- (pos= 0,5,0,10°)→
- (pos= -1,0,0)→
- (pos= 1,0,0)→

Lin_actuator.asm

Cyl_out.prt
cyl_in.prt
hose.prt

Lin_actuator_01.asm

Cyl_out.prt
cyl_in.prt
hose_01.prt

Plate.prt

Plate.prt

Cable1.prt

Cable1_01.prt



CREATE PART – FLEXIBLE COMPONENTS



Automate the process of linking related CAD models to the same Part

- Associate CAD documents to Part using a common base number

LIN_ACTUATOR.ASM -



owner



- LIN_ACTUATOR

LIN_ACTUATOR_01.ASM -



image

Name	Value	Description
Operation		Category for Operations preferences
Auto Associate		Preferences for associate, disassociate and auto-associate operations
Allow Association of Model Items by Model Item Sub-types	CUSTOM.LIBRARY	Lists Model Item sub-types
Allow Association of Model Items by Model Item Types	COMPONENT	Lists Model Item types for
Auto Associate Find and Create Part Using Base Number	No	Use a regular expression
Auto Associate Naming Parameter		Identifies the soft attribut
Auto Associate Numbering Parameter		Identifies the soft attribut
Auto Associate Truncate Name File Extension	No	Truncates the file extensio
Auto Associate Truncate Number File Extension	No	Truncates the file extensio
Auto Relate Drawings To Parts	Owner Only	Drawings that are derived
Build Part After Associate	No	When the user associates
Create Alternate Link On Check In	Yes	Allows a link of the next a

Note: Also need to have Create alternate link on check in set to Yes.

Operation		Category for Operations preferences
Auto Associate		Preferences for associate, disassociate and auto-associate operations
CheckIn Operation		Category for CheckIn preferences
Edit Attributes		Workgroup Manager preferences for Edit Attributes operation
General		General Windchill Workgroup Manager preferences
Base Number Regular Expression	[*\\ \\]*	Regular expression applied to an object's Number to find its base number without any suffixes or extensions.
Custom Modeled Part Class	wt.part.WTPart	Specifies a fully qualified class name for a custom modeled part if there are any.

CREATE PART – FLEXIBLE COMPONENTS

Auto Associate

Workspace: jentest_gc

Object List

Insert Edit

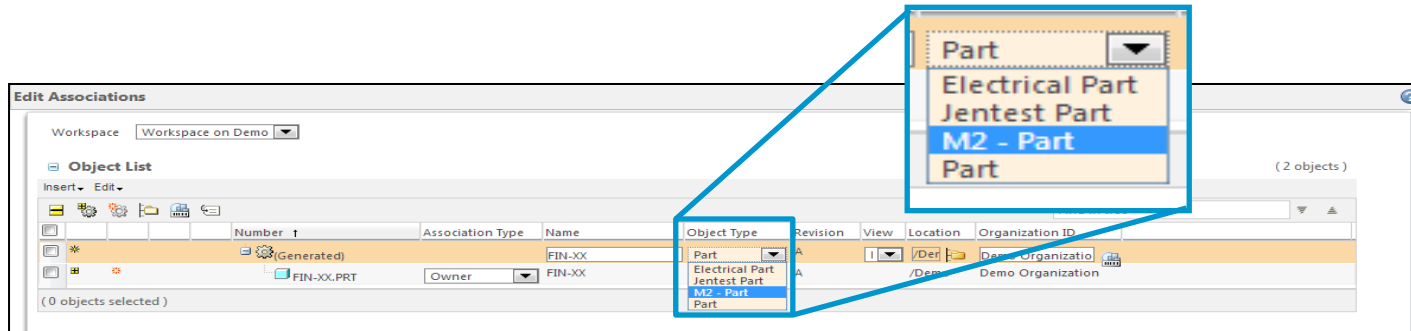
		Number ↑	Association Type	Name	Object Type	Revision	View
<input type="checkbox"/>		0000000044		0000000044.prt	Part	A	Design
<input type="checkbox"/>		0000000044_FLEX001.PRT	Image	0000000044_flex001.prt	CAD Document	A	
<input type="checkbox"/>		0000000044_FLEX002.PRT	Image	0000000044_flex002.prt	CAD Document	A	
<input type="checkbox"/>		0000000044.PRT	Owner	0000000044.prt	CAD Document	A	

(0 objects selected)

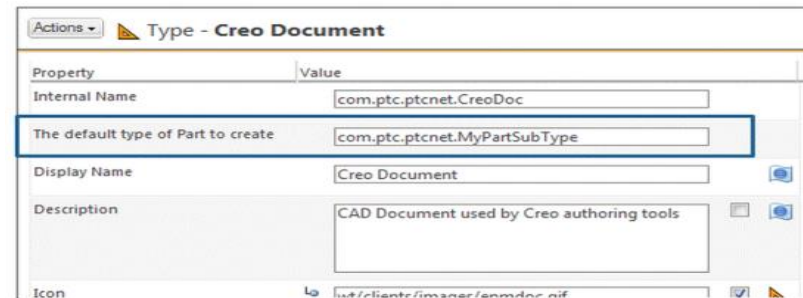
Note: In 10.2, since each of the CAD Document numbers are different, auto associate would have created 3 different Parts.

PART TYPE SELECTION DURING ASSOCIATION

- Provides greater flexibility when associating Parts and CAD Documents
 - Type selection provided in a drop-down in the “Object Type” display column in Edit Associations UI:



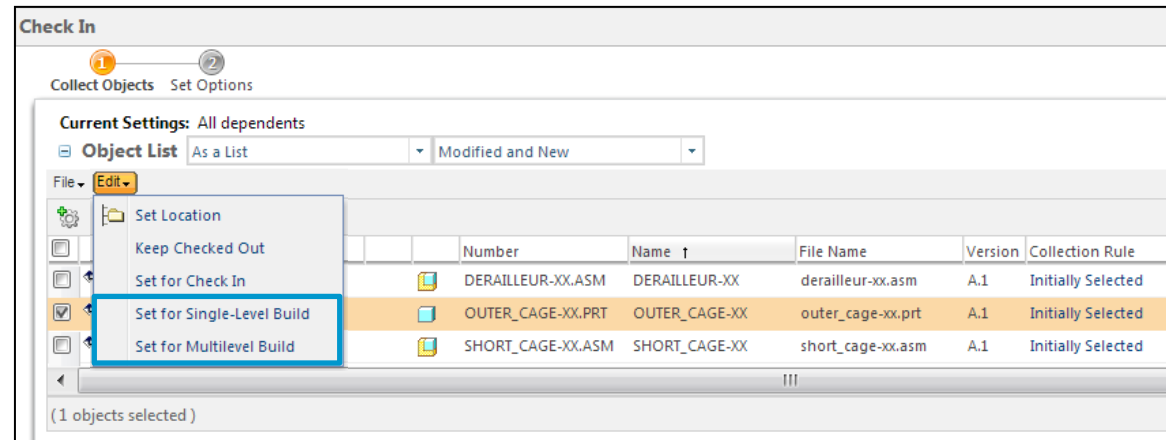
- Default Type displayed based on mapping defined by the Type Manager



- End User flexibility to override mapping and manually select the preferred/appropriate Part type

SELECTIVE BUILD ON CHECK-IN

- Allows a user to selectively choose which CAD Documents will build its related Parts during the Check-In process

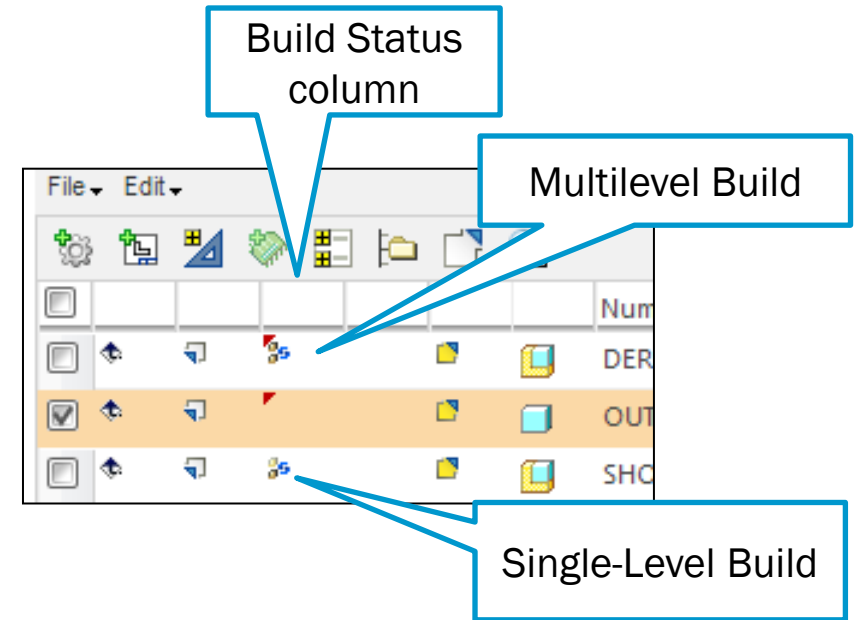


- Challenges
 - Prior release it was an All-or-Nothing proposition
 - No easy way to toggle between the two
 - Build process controlled by a preference
 - Check-in process cumbersome if specific items are not to be built

SELECTIVE BUILD ON CHECK-IN



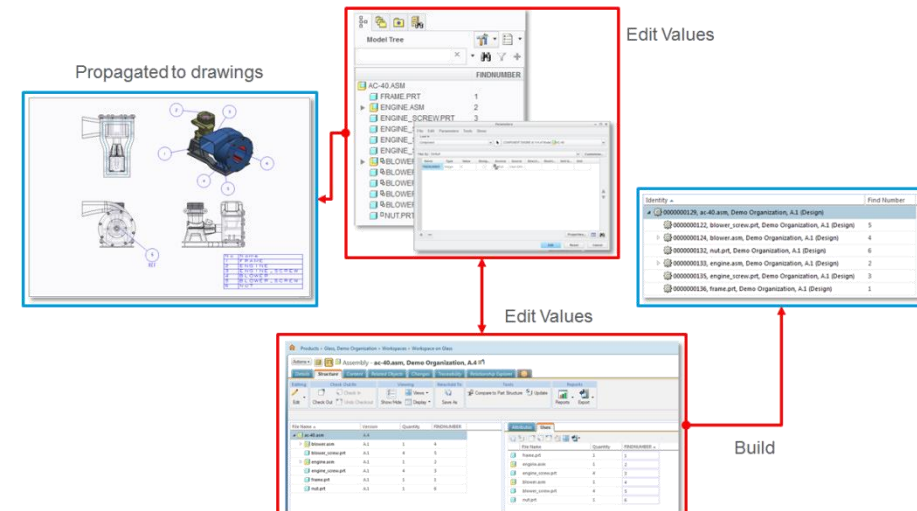
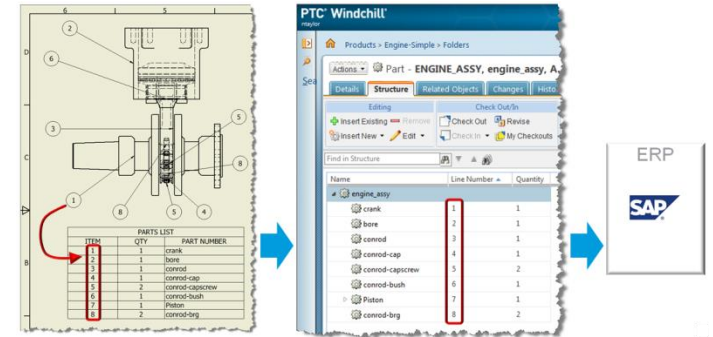
- Check-In Options:
 - Build Single Level
 - Build Multi-level (assemblies)
 - Do Not Build (blank)
- Default setting controlled by preference(s)
 - Operation > CheckIn Operation > Build After Check In (All/None) [User Preference]
 - If Admin Locks the value as “ALL” then the menus become disabled.
 - EPM Services Preference > Build Service Preferences > Build Image Association by Default (Yes/No) [Site Preference]
 - Not reflected in the UI, but preference will be honored if set to NO and will not build



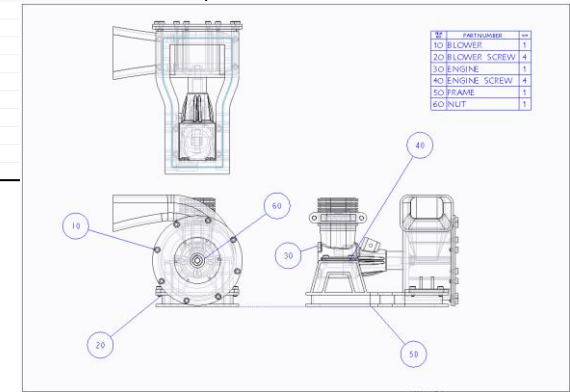
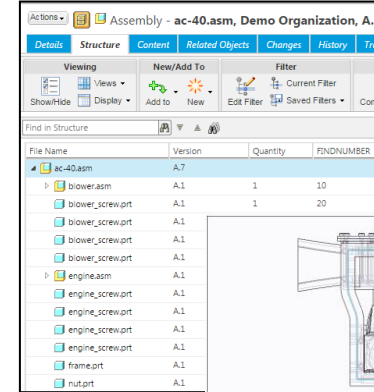
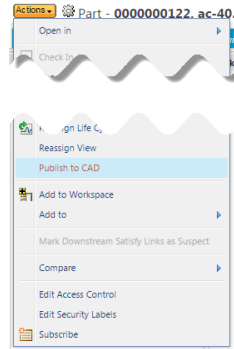
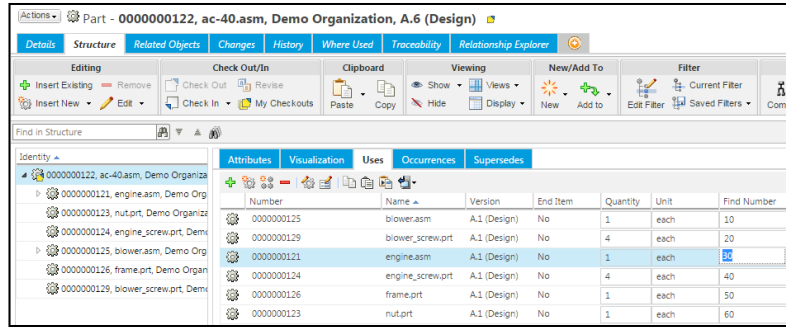
FIND/LINE NUMBER – TOP DOWN SUPPORT



- Synchronizing Find/Line numbers between the eBOM and its CAD drawing
- “Bottom-up” capabilities introduced in 10.2 M010
 - Map designated component parameter of assembly to attribute defined on EPM Memberlink
 - Set preference to designate MemberLink attribute to populate Line/Find numbers on BOM
 - Assign component parameter for display in drawing tables and balloon numbers
- Challenge
 - Difficult to input and maintain
 - Must enter values in CAD assembly for all occurrences of the same component individually
 - Changes must be driven from the assembly or assembly CAD structure (build process)



FIND/LINE NUMBER – TOP DOWN SUPPORT



1. Edit Find/Line numbers on the BOM
 2. Propagate changes down to CAD structure (and assembly)
 3. Changes synchronized with drawing during retrieval
- Benefit
 - Product Structure Browser more conducive to these edits
 - Industry standard rules enforced by UI
 - Automatically populates all occurrences in CAD with same values
 - Support with
 - All Creo versions
 - NX

Decorative geometric shapes in green, blue, yellow, and pink are scattered around the slide, primarily on the left side.

SUMMARY

- Keeping pace with new Creo development
- Improved flexibility, productivity and performance
- Introduction of CAD to eBOM management easier and more robust!

The image features several colorful geometric shapes, including triangles and lines in shades of blue, green, yellow, orange, pink, and purple, scattered across the background. A large, multi-colored triangular shape is prominent on the right side. The text 'LIVE WORX 16' is centered, with 'LIVE' in a thin, outlined font and 'WORX 16' in a bold, solid black font. A small 'TM' trademark symbol is positioned to the upper right of the '6'.

LIVE
WORX 16™

TAKE A FRESH LOOK AT THINGS

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