



- Enterprise Validation and Review Practice Groups
  - Optimise Design Reviews example
- PTC Creo View updates
- A look ahead to X-26 (December)

## Practice Groups

### *Enterprise Validation and Review*

#### MCAD VIEWING AND COLLABORATION

- Mechanical Data Visualization and Collaboration

#### DOCUMENT VIEWING AND COLLABORATION

- Synchronized Document Reviews

#### ECAD VIEWING AND COLLABORATION

- Electronic Data Sharing and Collaboration

#### DESIGN VALIDATION AND REVIEW

- **Optimize Design Reviews**
- Accelerate Interference Checking and Analysis
- Enhance Design Checking

#### DIGITAL MOCKUP

- Automate Interference Checking and Management
- Validate Product Designs with Interactive Animations

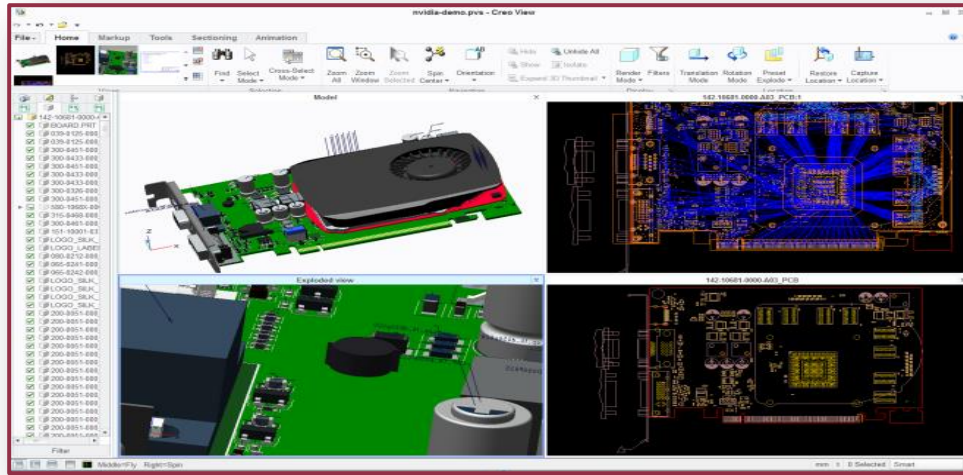
#### MODEL BASED ENTERPRISE

- Publish, Check and Visualize Model Based Designs

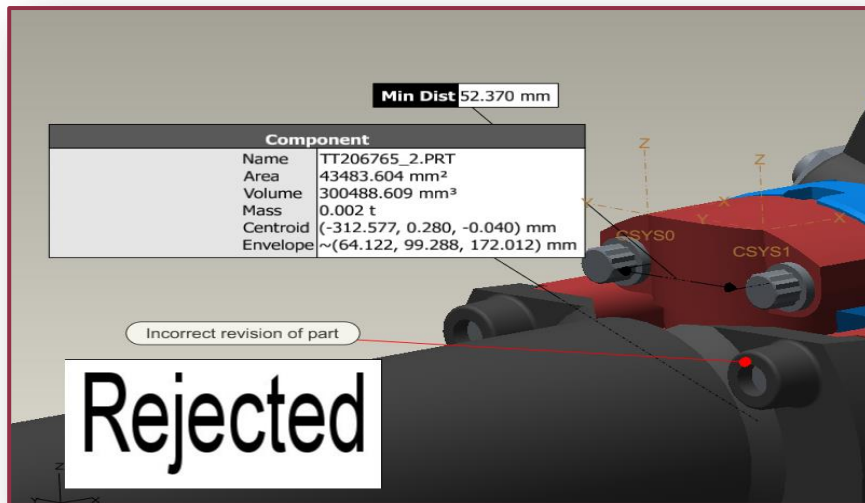
#### CUSTOM VISUAL APPLICATIONS

- Create custom applications
- Re-purpose and optimize product data
- Protect product IP





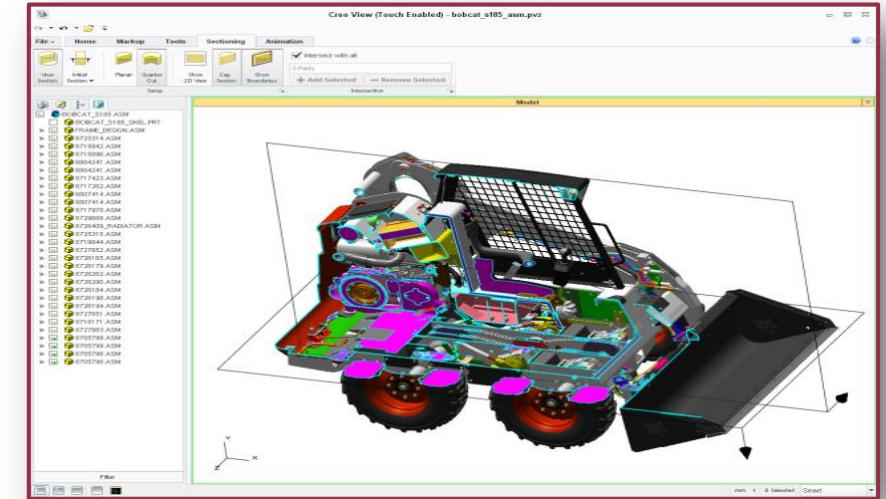
Review mechanical and electronic product data together



Capture design decisions in context



Review designs in stereo 3D



Interrogate product data : measure, cut sections, mass properties



Run visual reports to understand status

See and hear PTC Creo View and PTC Windchill Visualization in Action at [PACCAR](http://www.ptc.com/webcast/improve-design-reviews)

- [www.ptc.com/webcast/improve-design-reviews](http://www.ptc.com/webcast/improve-design-reviews) (registration required)

PTC & PACCAR  
“Improving Design Reviews”



Bill Ryan, Vehicle Packaging Engineer, PACCAR

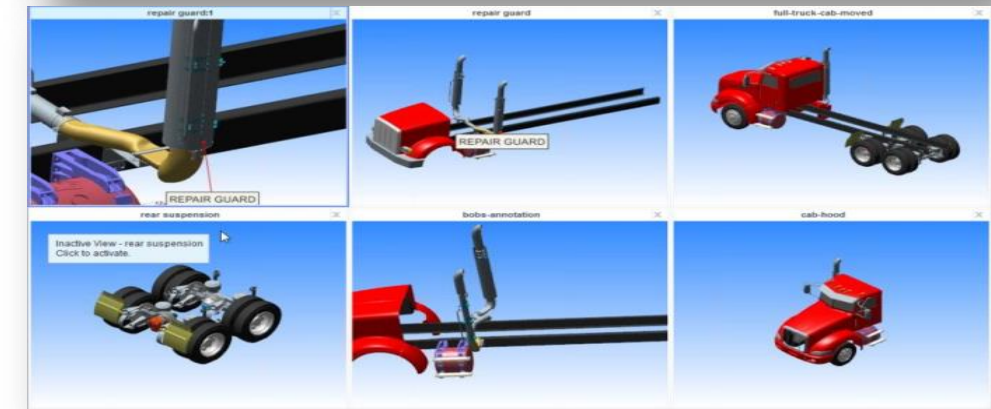
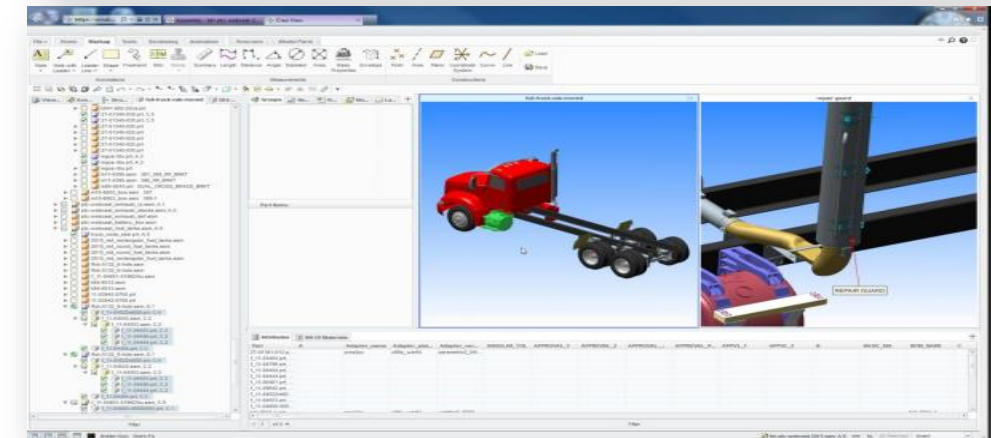

Summary PTC®

**CHALLENGES:**

1. Package Reviews
2. Data Management
3. Collaboration

**BENEFITS:**

- Full Reviews
- Unified Workflow
- Better Decisions



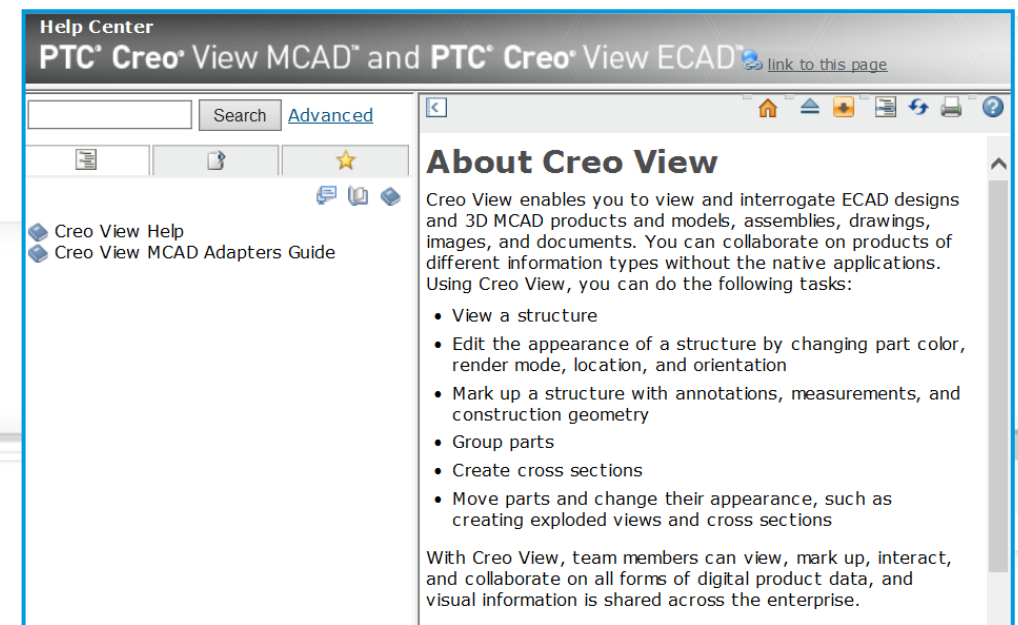
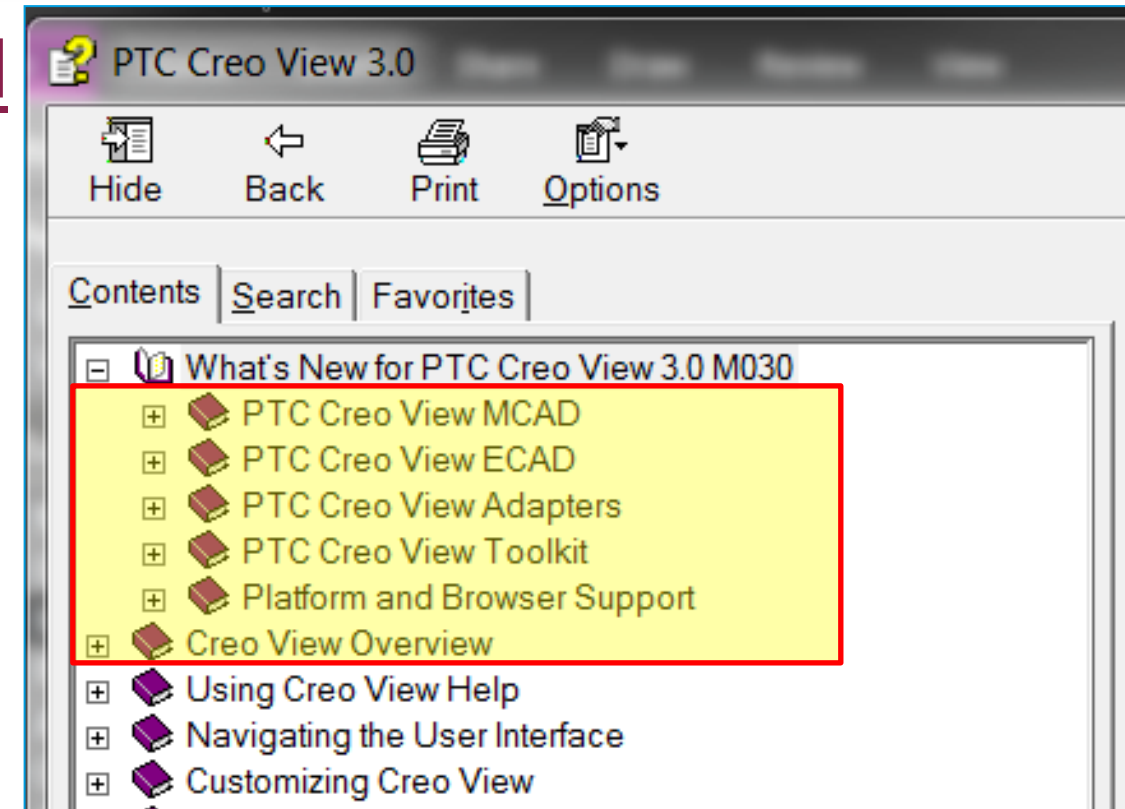
# PTC Creo View 3.0 MCAD M031

New feature summary



Embedded in PTC Creo View and also on [Support.PTC.COM](http://Support.PTC.COM)

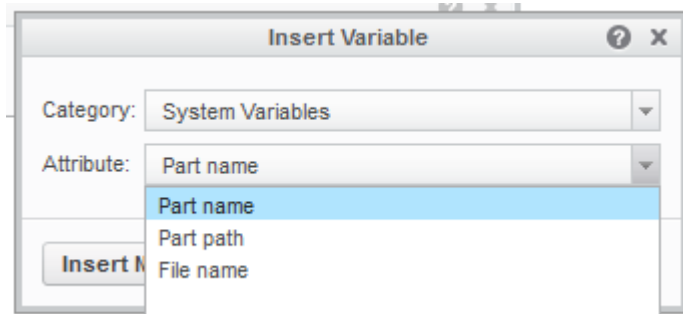
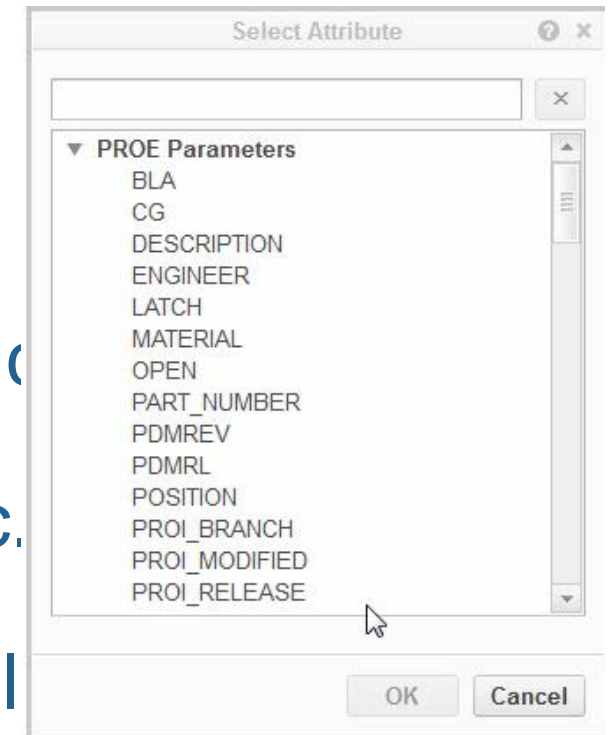
- For PTC Creo View 3.0 M031 you can find additional what’s new topics in the Help, specific to the M030 release:
- From the product, open the Help. Click the Contents tab and then browse through What’s New for PTC Creo View 3.0 M030 topics located at the beginning of the table of contents
- From [PTC.com](http://PTC.com), open the PTC Creo View 3.0 M031 [Help Center](#) and in the table of contents, click PTC Creo View Help. Browse through What’s New for PTC Creo View 3.0 M030 topics located at the beginning of the table of contents



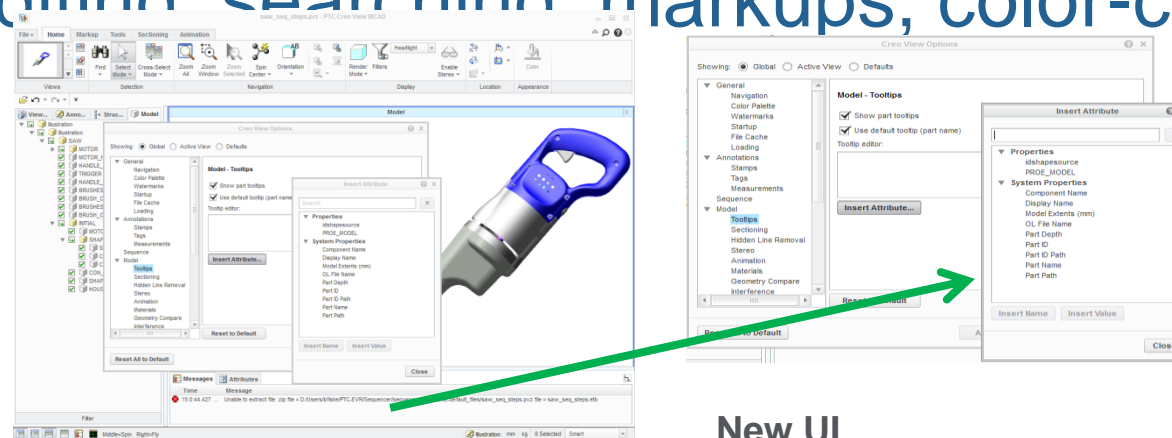
# Standardised and Improved Access to Product Properties in PTC Creo View

The use of part and system attributes is enhanced in several areas with a theme of improved consistency and flexibility throughout the PTC Creo View user interface.

- Auto-completion of potential matches
- Order by category and the ability to collapse and expand property categories
- Access to internal properties such as hierarchy level, part size, etc.
- Paste lists of search values from external sources (Microsoft Excel)
- Consistent across tooltip editing, searching markups, color-coded reporting, etc.



Existing UI



New UI



Search Product Structure

Query Name:

Search Against: all parts

Do query of type:  Part  Attribute  Spatial

Attribute:  Choose...

Category: All Choose...

Comparison: String

Condition: Matches

Value:  More...

Attribute:  Choose...

Ignore Case

Show Multi-line

Reset Save As Group Apply OK Cancel

Select Attribute Categories

- PROE Component Parame...
- PROE Parameters
- PVS File Properties
- System Properties

Select All Clear All

OK Cancel

**Simple selection of categories**

- CAD, WTPart, EPMDoc, system

Comparison: String

Condition: String  
Numeric  
Existence

Condition: Exists

Exists  
Does not Exist

## NEW: Search for existence of attributes

- Find parts with missing attributes easily
- Filter search for parts that only contain certain attributes

Includes performance improvements ~2-15X faster

Search Product Structure

Query Name:  Manage Queries...

Search Against: all parts

Do query of type:  Part  Attribute  Spatial

Attribute: Component Name Choose...

Category: System Properties Choose...

Comparison: String

Condition: Matches

Value: <Multiple Values> More...

Attribute:  Choose...

Ignore Case

Show Multi-line

Reset Save As Group Apply OK Cancel

Multiple Search Values

Add multiple search values, one per line

- 035-0012-000\_1.PRT
- 035-0020-000\_1.PRT
- 035-0023-000\_1.PRT
- 035-0024-000\_1.PRT
- 035-0055-000\_1.PRT
- 035-0107-000\_1.PRT
- 036-0012-000\_1.PRT
- 036-0013-000\_1.PRT
- 036-0014-000\_1.PRT
- 036-0016-000\_1.PRT
- 036-0018-000\_1.PRT
- 036-0023-000\_1.PRT
- 036-0027-000\_1.PRT
- 036-0043-000\_1.PRT
- 036-0051-000\_1.PRT
- 039-0008-000\_1.PRT
- 039-0011-000\_1.PRT
- 039-0125-000\_1.PRT

OK Cancel

List of search parameters  
• Pasted from Excel, etc.

**Enhanced Search tool enables insertion of external search parameters, such as a list of properties from an external source:**

Enables for searching for multiple values (implicitly “OR-ed”)

General usability enhancements

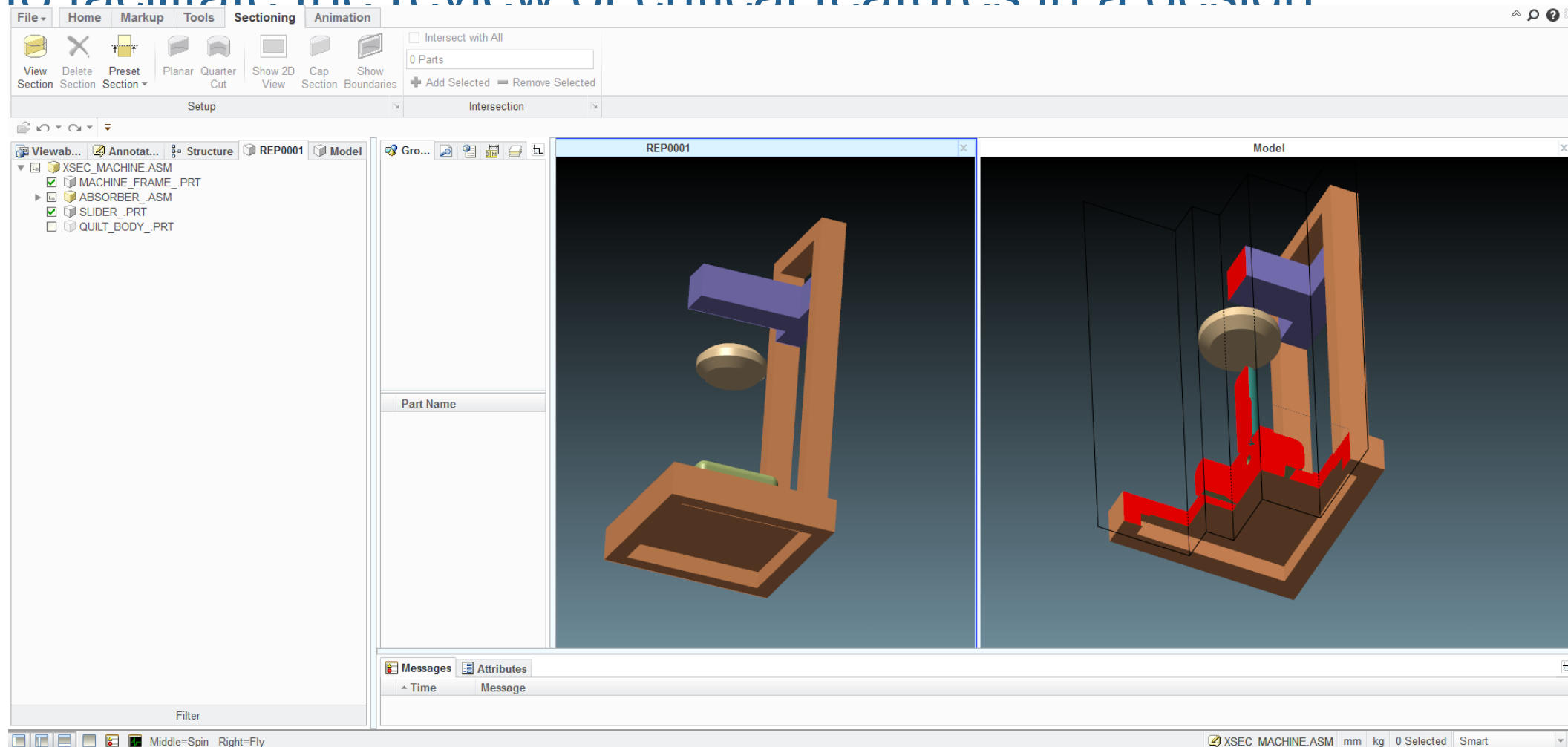
Supports color-coded reporting

Includes performance improvements ~2-15X faster

The display of offset cross sections defined in PTC Creo Parametric data sets is supported.

User Interface Location: Click Sectioning ► Preset Section

- Offset cross sections are a valuable tool for designers to communicate design intent or to facilitate the review of critical features in a design





Markups are parametric such that you can embed tags for properties into annotations and they are then associated with whichever part they are connected to.

- **User Interface Location:** In the Note dialog box click **Insert Attribute** or click **References**.

- You can preserve the order of the display of markup annotations when saving and loading.

- Attributes and attribute categories appear as a tree

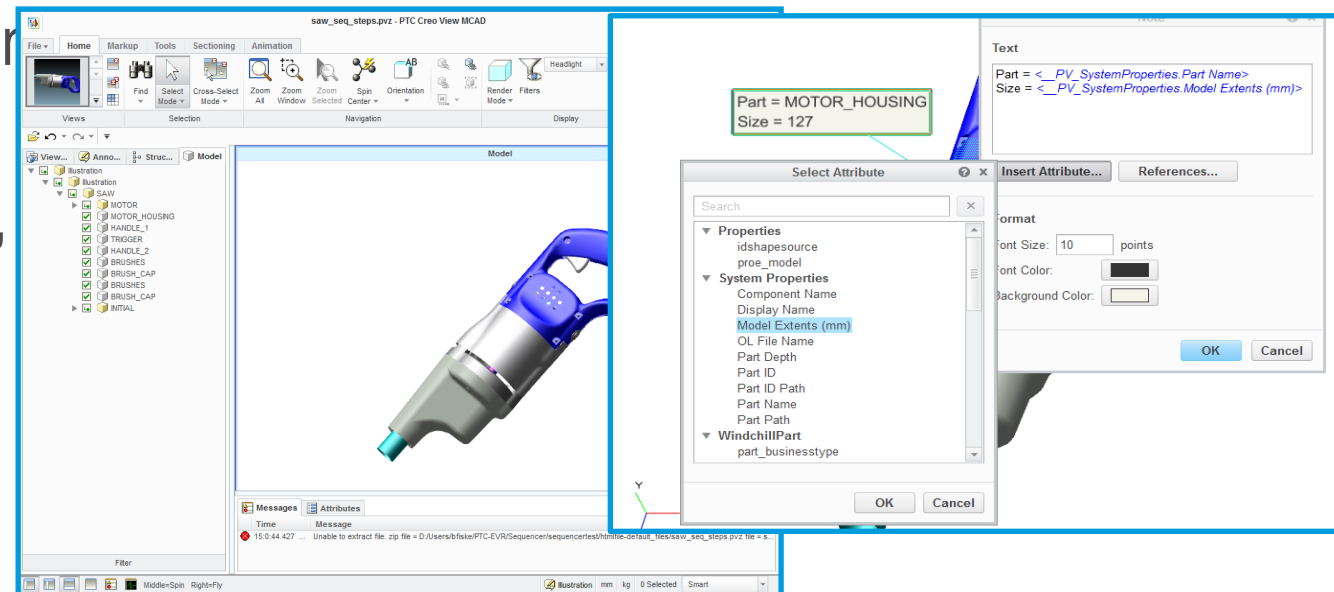
- You can search for attributes in the dialog box

- You can add one or more attributes in the Note dialog box

- The References dialog box lists references which are connected to the note directly or indirectly

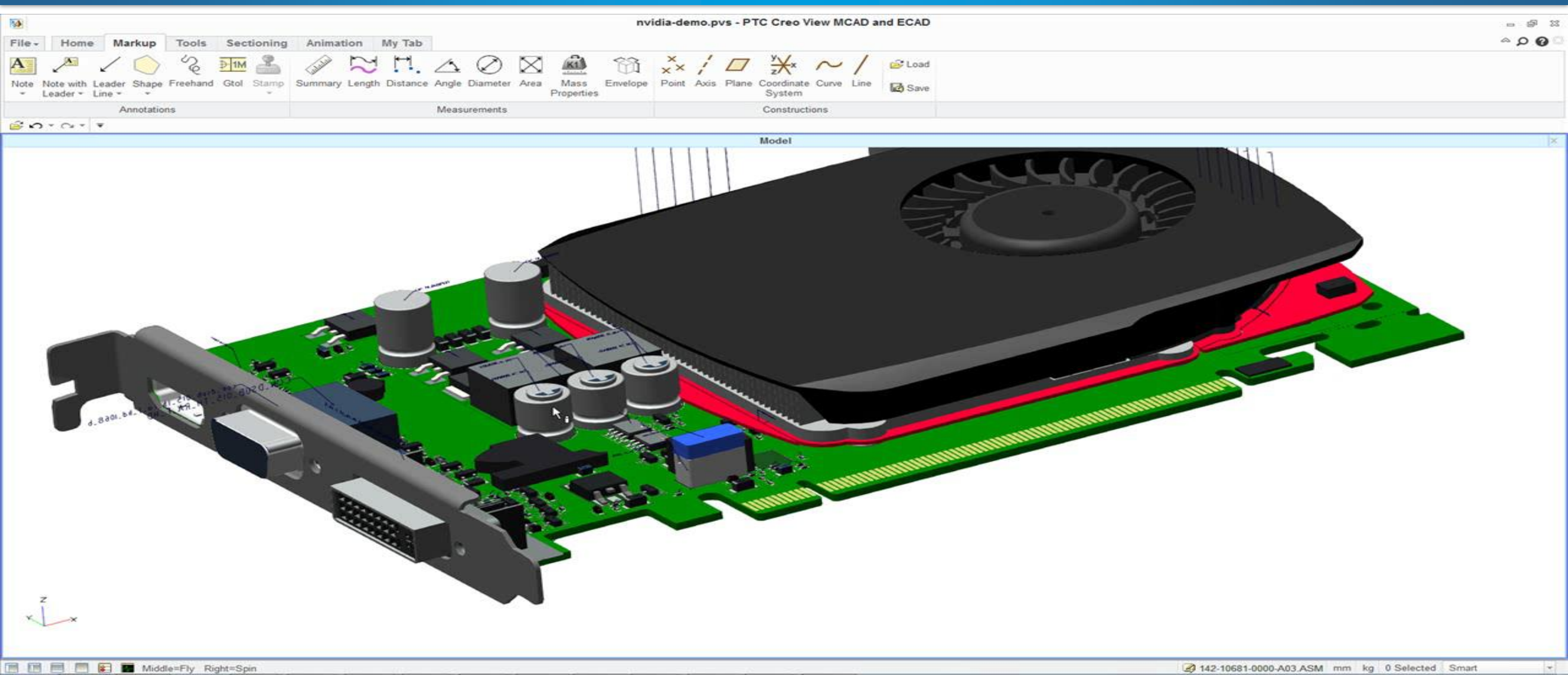
- An error message appears in the Messages pane if a component is not selected

- Notes, measurement labels, and dimension lines can be saved in the current order of display



list or does not reference a component. The markup can be saved in the

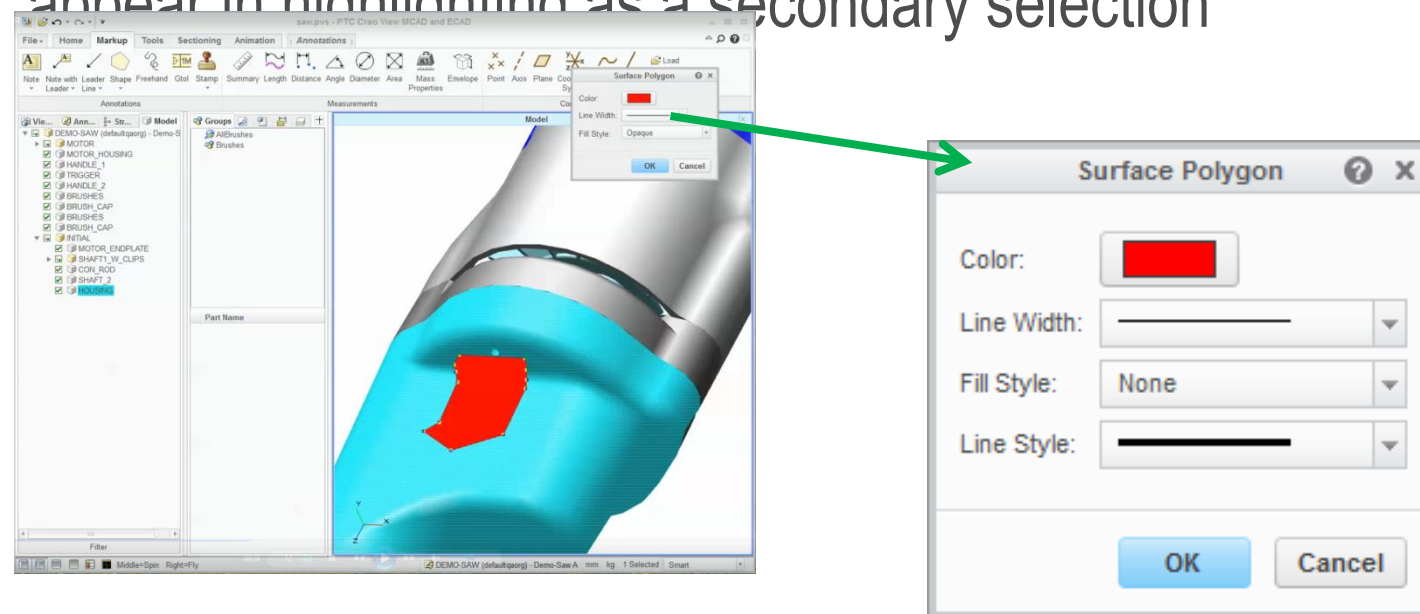
# Parametric Markup Example Video



Markups can be attached directly or indirectly to part surfaces. Markups are associated with the part to which they are connected

- You can edit markups for 3D shapes and attach them to surfaces on parts

- Add surface polygon markup directly on one or more component surface geometries
- Establish connectivity for the shape, such as rectangle and ellipse markup, by placing the center point of the shape onto the underlying component
- Connect a leader line to a note, surface polygon and component
- Select any markup to see the underlying component and the directly or indirectly connecting objects appear in highlighting as a secondary selection

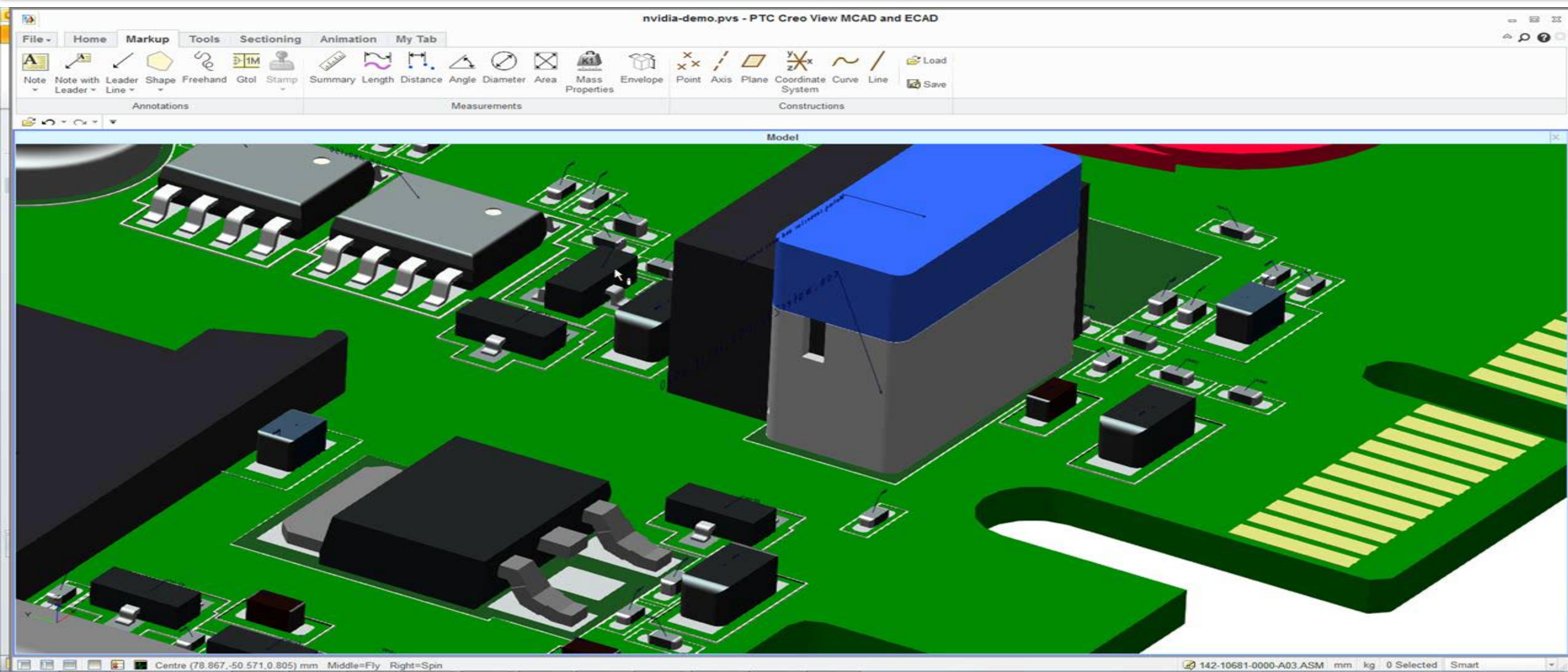


## Draw polygons directly onto model surface

- Reduces ambiguity and improves clarity of intent
- Fill : opaque, transparent, none

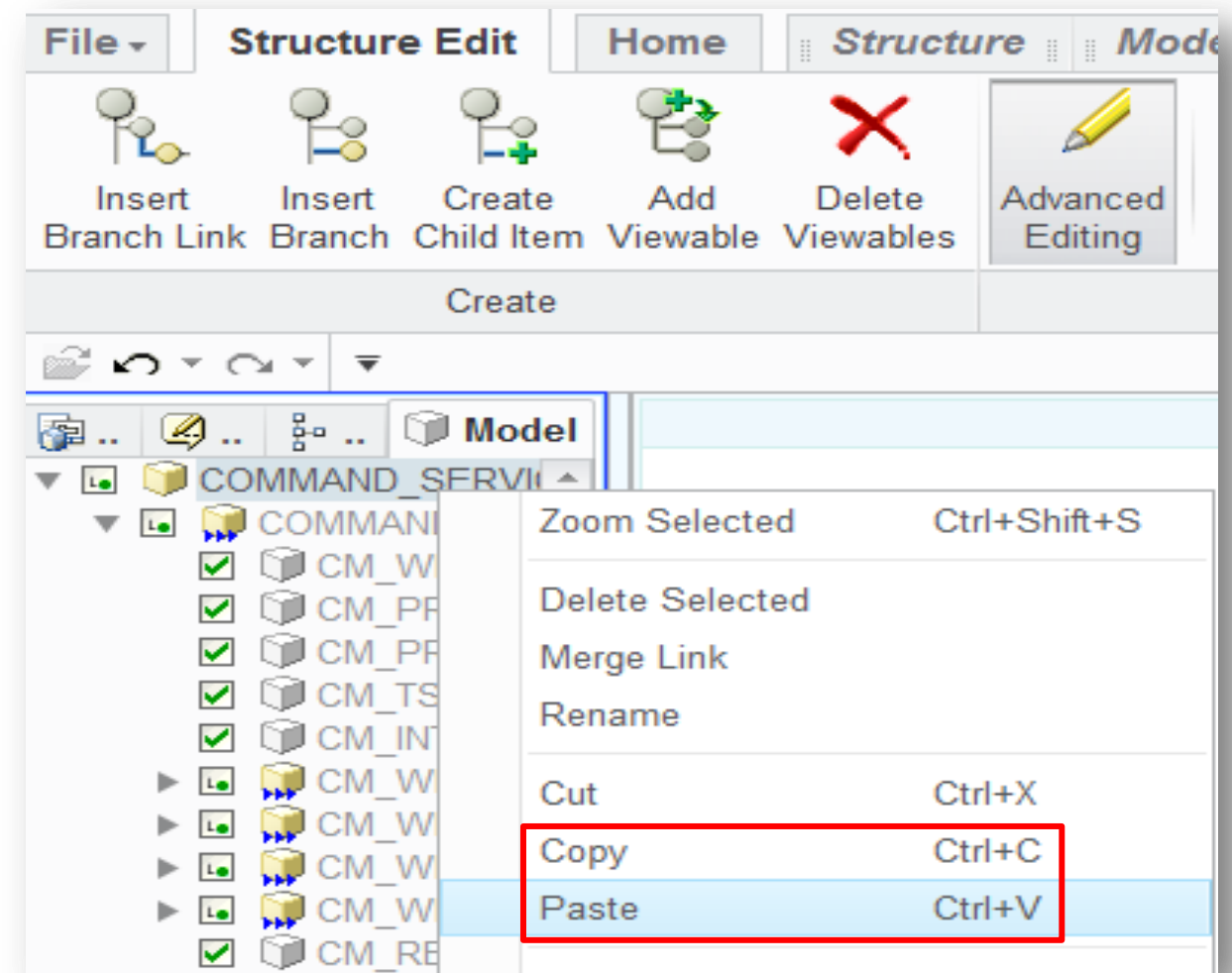


# Attaching Markups to Part Surfaces



You can edit structure data by copying and pasting

- User Interface Location:
- Click Structure ► Edit Structure and then in the Structure tab, click Advanced Editing
- Copy/Paste Tree Editing:
  - In Structure Edit mode (PTC Creo View MCAD)
  - Streamlines what-if assembly ops
  - Quickly generate different configs
  - E.g. copy one, paste multiple





You can use two hands on large format, touch-sensitive display devices.

- You can perform multi-touch gestures such as pan, rotate, and zoom in and out by using two fingers on one hand or by using two hands
- If using two hands, multiple fingers on the same hand are treated as a single touch point



Touch-enabled product navigation with Creo View



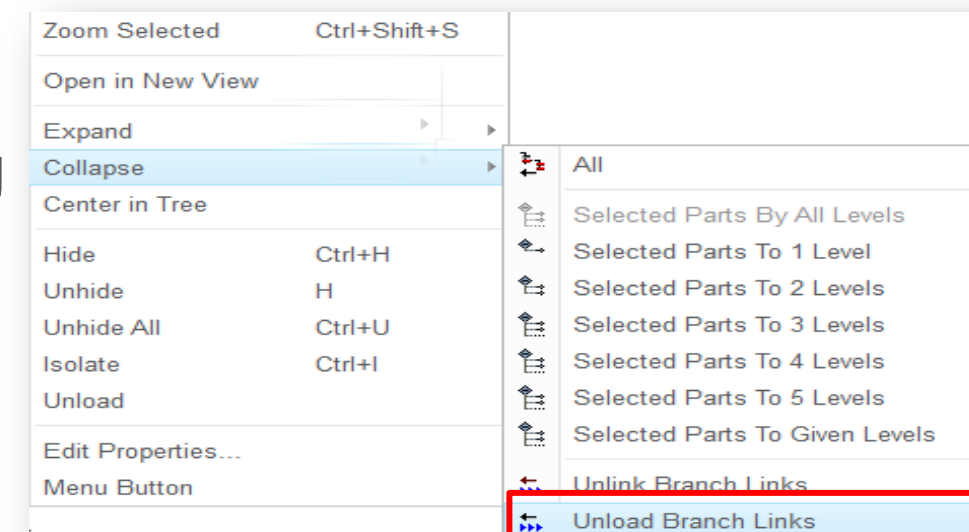
You can unload and reload branch links from within the Structure Tree view

When performing common operations on the navigation tree, unloading and reloading branch links can significantly improve the flexibility and performance of working with larger assemblies and structures

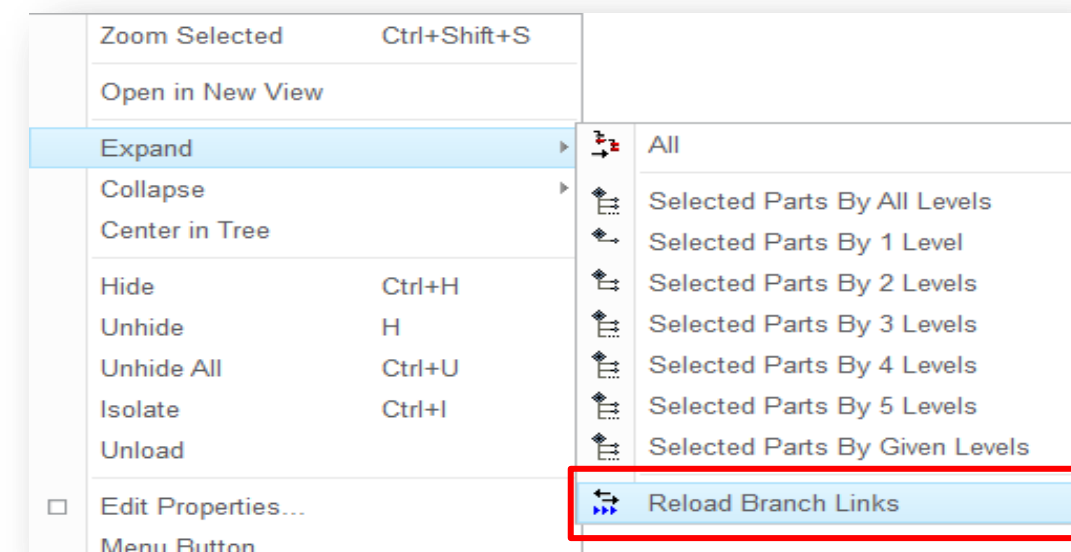
• **User Interface Locations:**

- Click Structure ► Collapse All ► Unlink Branch Links
- Click Structure ► Collapse All ► Unload Branch Links
- Click Structure ► Expand All ► Reload Branch Links
- Right-click in the Structure Tree and click Collapse or Expand

- Removes need to reload entire data if something changes (e.g. republishing)
- Supports scenarios of publishing updated sub-assemblies, etc.
- Reload preserves view state (tree expansion, branch link expansion, geometry state)
- Allows users to update the geometry in an open session without re starting and re loading massive assemblies
- Updates the required *sub-assemblies* of the structure, rather than having to perform a complete reload



double click on component text = Link



click on triangle = *Expand*

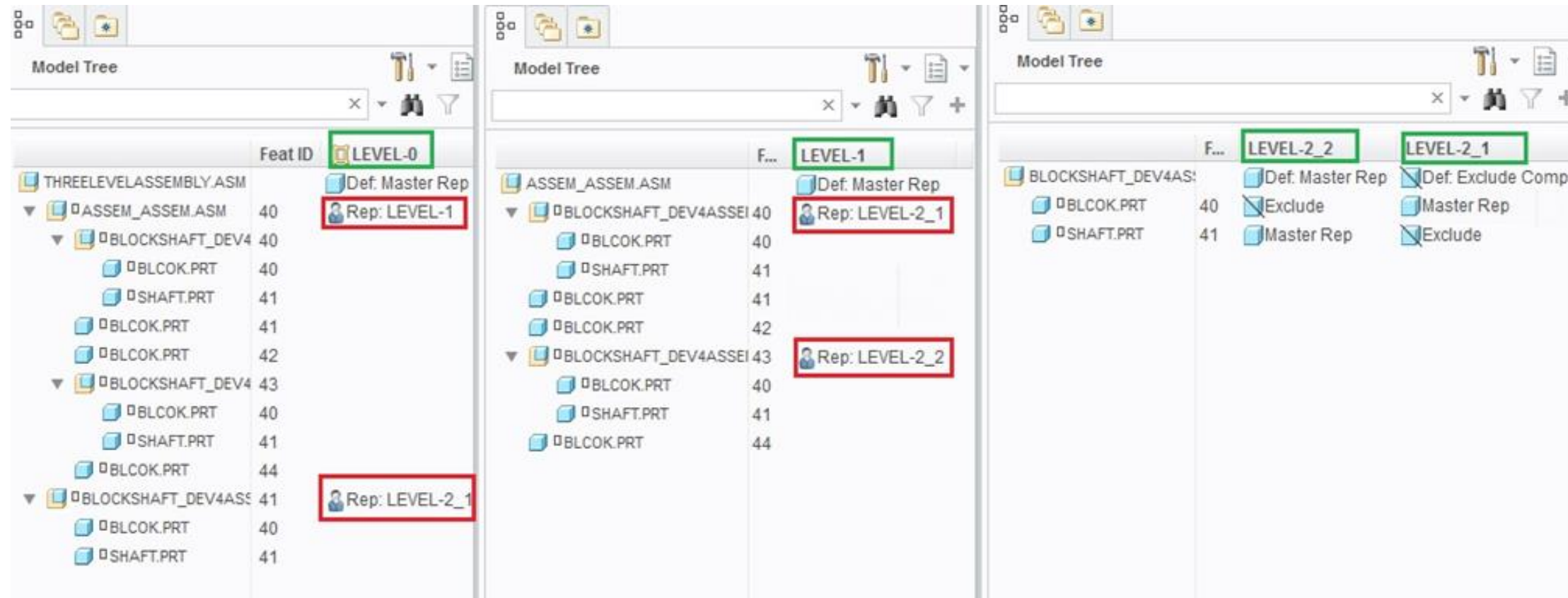
# PTC Creo View 3.0 M030 Publishing Adapter Updates

The PTC Creo Parametric Adapter for PTC Creo Parametric supports new functionality introduced in PTC Creo Parametric 3.0:

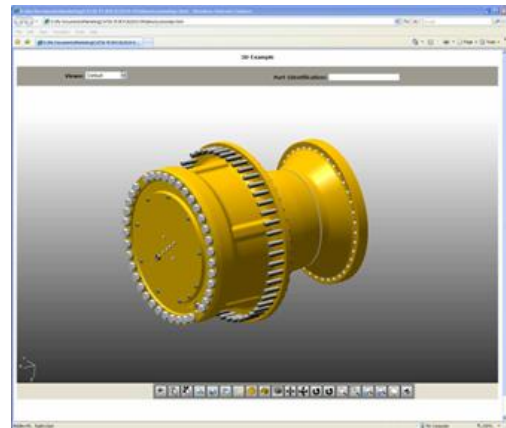
- **Publishing of Assigned Mass Properties**
- **Publishing of GTOL enhancements**
  - Additional Material conditions
  - New All Over and Datum Translation modifier symbols
  - Text Below, Text on Right & Text Justification for Text Above and Text Below
- **Publishing Multi-CAD assemblies with PTC Creo Unite technology**
  - Support for command line and interactive publishing of PTC Creo Unite parts and assemblies
  - See article [CS188975](#) for information about the current caveats of PTC Creo Unite technology working with PTC Creo View and PTC Windchill PDMLink 10.2
- **Publishing of hierarchical simplified reps**
  - User defined reps which contain substitution is currently not supported and such reps will not be converted



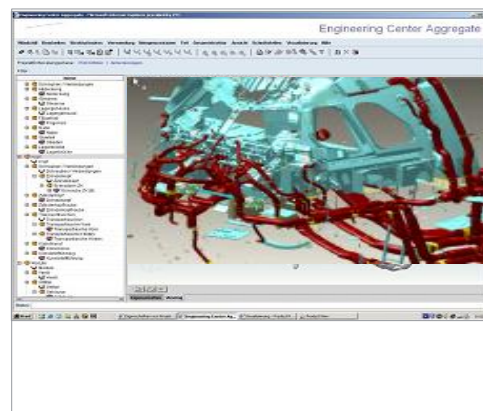
## Publishing example



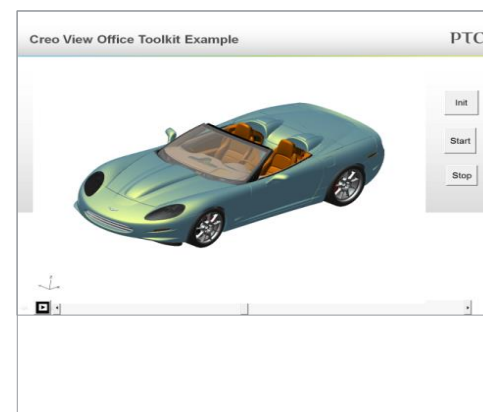
- Top Assembly is **THREELEVELASSEMBLY.asm**
  - Has a Simplified Representation **Level-0**
    - which refers to representation **Level-1** of **ASSEM\_ASSEM.asm**
    - and **Level2\_1** of **BLOCKSHAFT\_DEV4.asm**
  - **ASSEM\_ASSEM** has a representation **Level-1** which refers to **Level-2\_1** and **Level-2\_2** of



PTC Creo View Web  
Toolkit  
for embedding in  
web pages



PTC Creo View Java  
Toolkit  
Building custom Java  
apps

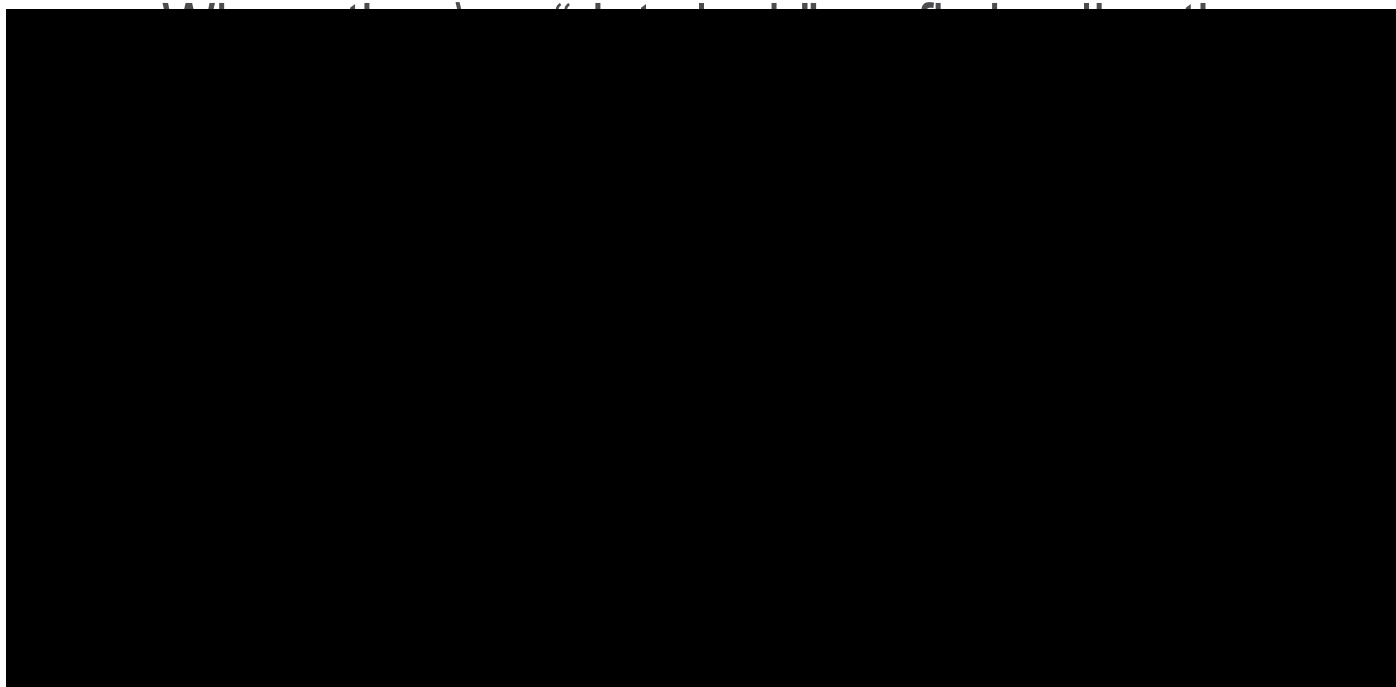


PTC Creo View Office  
Toolkit  
Embed in Office  
Documents

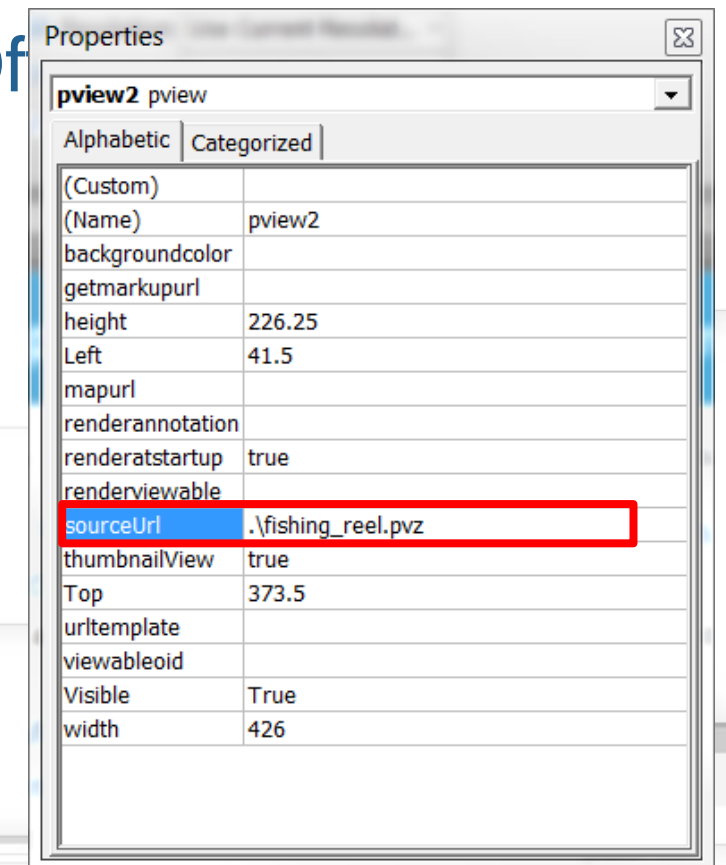
# PTC Creo View 3.0 M030 Toolkits Web, Java, MS Office

Simplifies usability and portability of embedding Creo View data in Microsoft Office formatted files

- In previous versions of the PTC Creo View Office Toolkit, users had to embed the explicit path to the PTC Creo View file(s) to be used
  - Example: `C:\ptc\creo_view_api\demodata\Fishing_Reel\fishing_reel.pvz`
- As of PTC Creo View 3.0 M030, users of the PTC Creo View Office Toolkit can embed data files relative to the location of the Microsoft Office document
  - Example: `.\fishing_reel.pvz`

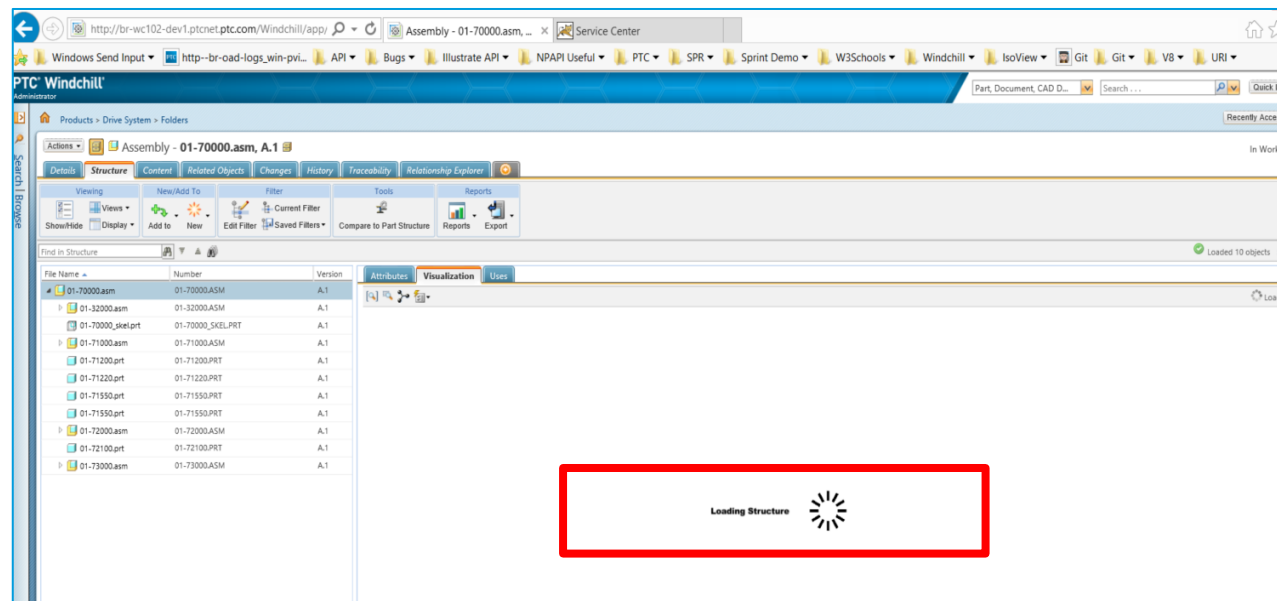


ent folder

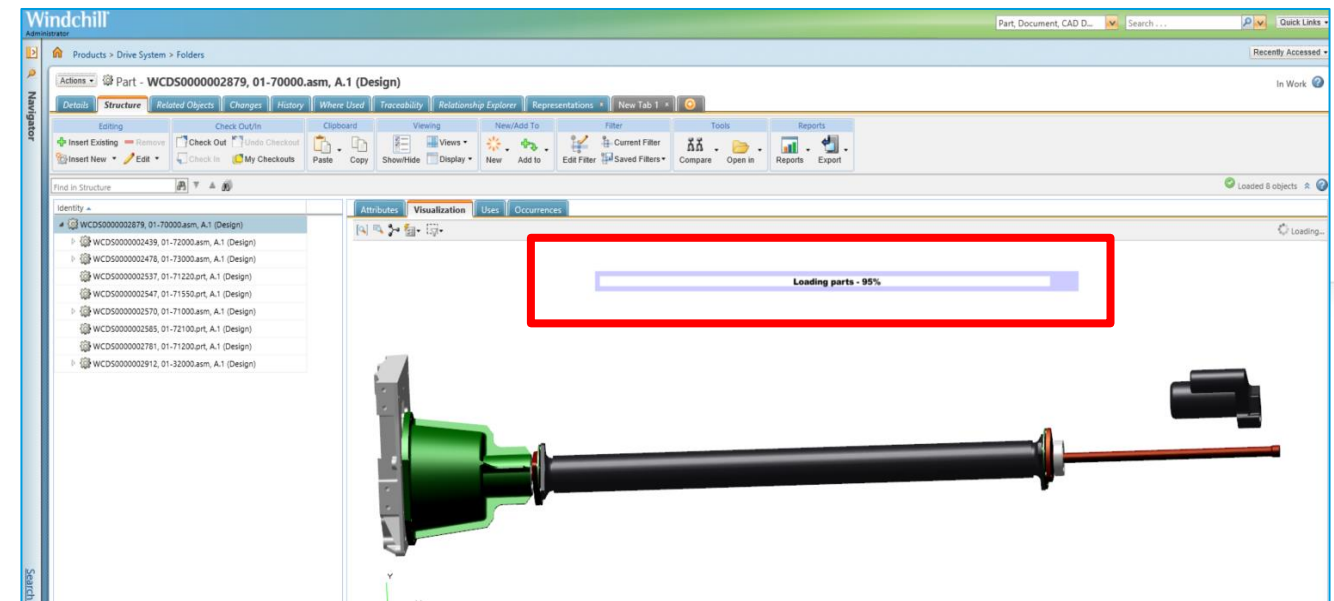


Updated means of indicating progress towards loading PTC Creo View data

- A new progress bar has been added to the PTC Creo View Web Toolkit to show that the structure is being loaded and the graphics window is loading
- Progress/Busy indicator is turned on and it's appearance controlled by **server\_prefs**



Prior to Creo View 3 M030: Spin box used for load progress



As of Creo View 3 M030: progress bar shows loading status



# PTC Creo View 3.0 M030 - Platform Updates

## Additional Platform and Browser Support

- The PTC Creo View Adapter for PTC Creo Parametric supports Microsoft Windows Server 2012 R2.
- Adobe Live Cycle ES4 transitions to Adobe Experience Forms (AEF) (*M032 – mid-June*)
  - No change in functionality, pricing, licensing – just a re branding
  - AEF PDF Generator tested and supported with Microsoft Windows Server 2012 R2
- **Early advance notice: Last version of Chrome Browser support with NPAPI**
  - The Netscape Application Programming Interface (NPAPI) is the mechanism by which the PTC Creo View application user interface can be embedded in the Google Chrome web browser.
  - From January through September 2015, there is a phased discontinuation of the NPAPI support in the Google Chrome web browser. As of September 2015, Google will discontinue support of this capability.
  - See this [NPAPI notification link](#) for a more detailed description of the reasons for, timeline, and user

# PTC Windchill Visualization New Feature Summary

## Summary of New Features/Functions

- Improved Performance of Accessing Positioning Assemblies from PTC Windchill
- Improved Branch Link Manipulation in PTC Creo View
- Show Assembly Features in Dynamic Structures
- CAD Represented Parts Capability for Dynamic Part Structures
- Interference Detection for Assembly Features & CAD Represented Parts
- Ability to Display or Not, Unplaced Components in Part Structures
- Standard Attributes available in PTC Creo View
- Provide “Open in PTC Creo View” action in PTC Windchill CAD Document Structure
- Use Specified Filter when Opening Annotations from PTC Windchill
- PTC Visualization Getting Started Guide



# Performance of Accessing Positioning Assemblies from PTC Windchill

Scaling performance for efficient distributed design re

Usage of positioning assemblies is growing rapidly

Both CATIA V5 and PTC Creo Parametric

Customers are using these from “top to bottom”

Lots of “branch link” expansion...

Communications are now optimised

Reduced “chatter” between PTC Creo View & PTC Windchill 10.2 M030

Internal tests have shown up to 3X improvement in structure access

... dependent upon network, etc.


**Creo View** \*

1<sup>st</sup> Load = 44 sec

- 13 sec for links
- 31 sec for models

2<sup>nd</sup> Load = 20 sec (*Cache Enabled*)

- 13 sec for links
- 7 sec for models



The screenshot shows the PTC Creo View software interface. On the left is a tree view of an assembly structure. The root is 'btr-master-model-all.asm, A.25'. It contains a sub-assembly 'ngp\_master\_all.pb.asm, A.17', which in turn contains several other sub-assemblies: '2010\_master\_579123sb\_box\_2d.asm, A.5', 'ngp\_cab.asm, A.10', 'ngp\_slpr.pb.asm, A.2', '579123sb\_hood.asm, A.6', 'ngp\_aero.pb.bumpers.asm, A.1', 'ngp\_mh.pb.fairing\_tender.asm, A.47', 'ngp\_aero\_sbf\_a\_front\_susp.asm, A.5', and '2010\_ngp\_mh\_isx\_exhaust.asm, A.16'. On the right is a 3D model of a truck, with different parts highlighted in various colors like yellow, red, and grey.

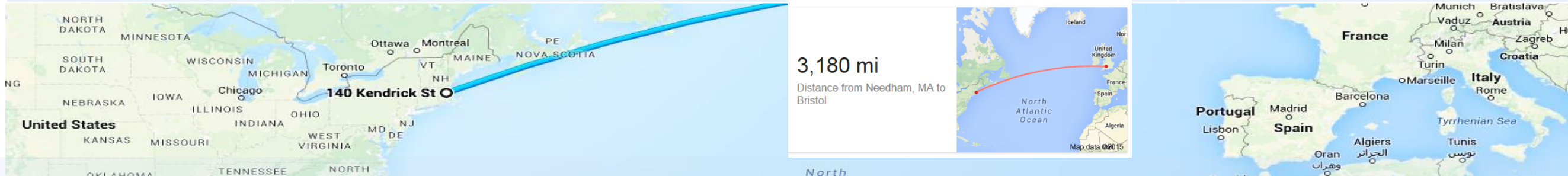
**Why this is important**

\* Data courtesy of Bill Ryan, PACCAR  
Presented at PTC Global Live 2013

# Performance of Accessing Positioning Assemblies from PTC Windchill

Optimizing for web access patterns yields significant improvement

	Local to PTC Windchill Server	Bristol / Needham	HTTP Requests
Pre change	4.16 secs	39.62 secs	1485
Post change	3.86 secs	11.89 secs	297



- **Comparison of opening a PTC Creo View assembly with PTC Windchill local vs over LAN/WAN**

- Local implies running from laptop to locally connected PTC Windchill Server instance
- Bristol/Needham (which involves a round trip via Needham to get to the server).

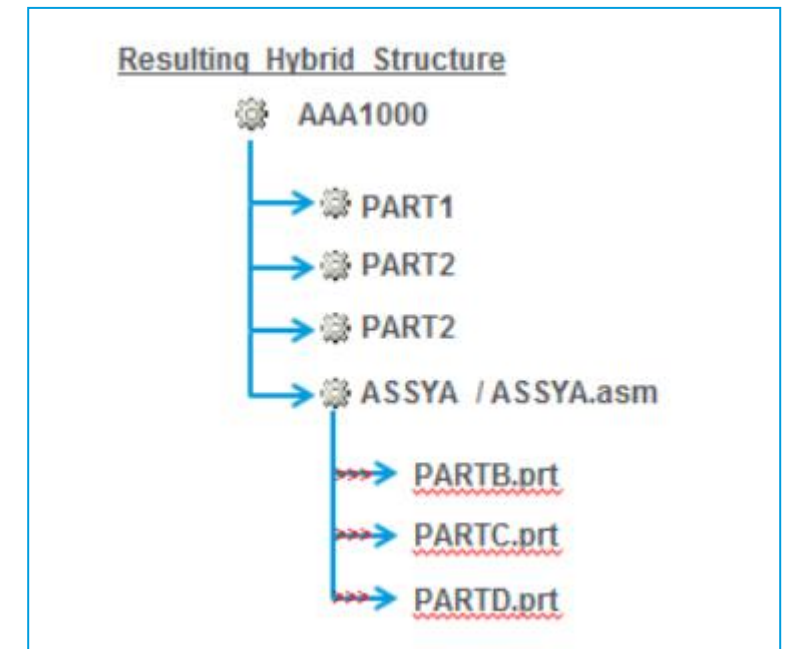
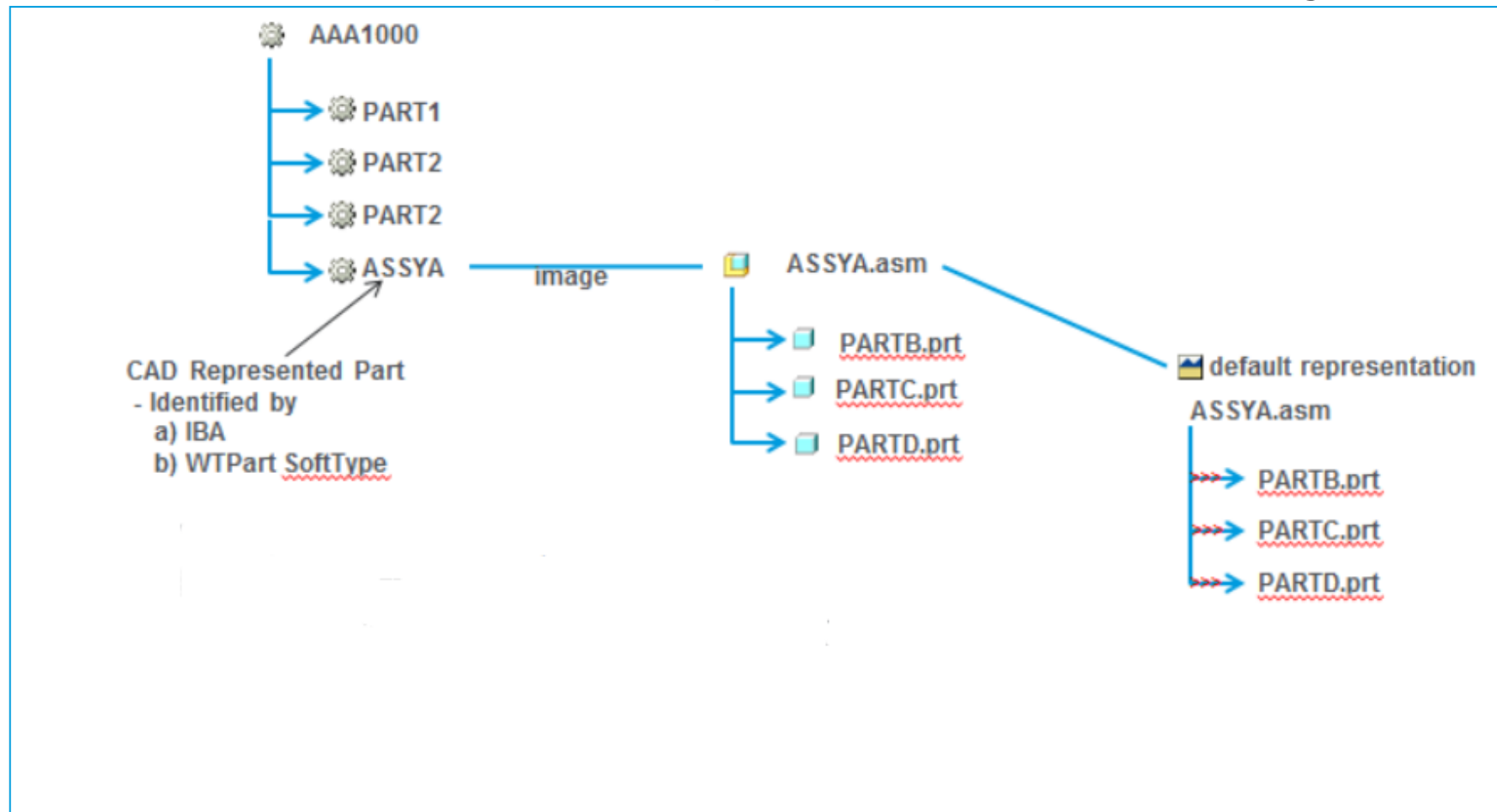
A **CAD Represented part** is a **WTPart** in the part structure that is to be represented by the CAD representation

- When displaying dynamic part structures, the 3D graphics are composed of merged representations
- For some **WTParts**, the child parts can be missing, incomplete or do not have associated graphics
  - An example would be Supplier Parts – a single **WTPart** represents the complete supplier assembly.
- Beginning with PTC Windchill 10.2 M030, CAD Represented Parts can now be displayed
  - The whole representation of the CAD Represented part is used to display the 3D graphic
    - This is the **default** representation
    - If missing, the default representation from an image associated **EPMDocument** is used, and all children merged
  - Any child **WTParts** of the CAD Represented Part in the **WTPart** structure are removed from the visualization structure

*Also supported in later models. Part representation*

A graphical view

- In this example:
  - **WTPart** “ASSYA” has an image association to a CAD Assembly that has three children
  - The CAD Assembly has a published representation.
  - When identified as a CAD Represented Part, the resulting visualization structure will include the children of



Hybrid viewable structure (as seen in Creo View & Viz tab)



Standard Attributes are an alternative way to that of IBAs for adding attributes to certain Windchill objects.

Unlike IBAs Standard Attributes are stored in the (extended) table of the host object rather than being in a linked table, therefore they provide better performance.

Currently WTDocuments and WTParts support Standard Attributes. WVS also includes the necessary support for the future EPMDocument support of Standard Attributes.

- Use Cases

- Scenario #1: A user launches a dynamic representation in PTC Creo View for a given representable, the standard attributes for the representable are accessible in PTC Creo View
- Scenario #2: A user publishes a representable having a standard attribute, the representation will have the standard attribute and it's value accessible when opening in PTC Creo View

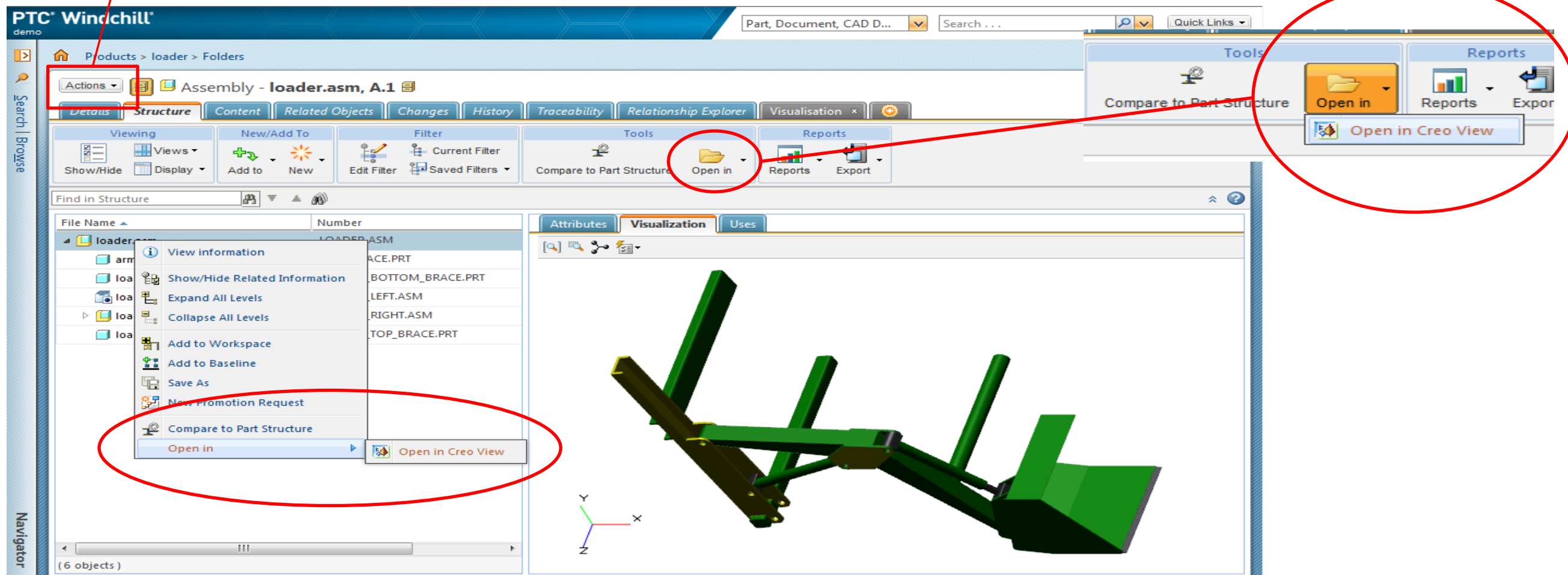
- UI Change in Creo View

- In Creo View the standard attributes will be displayed alongside other attributes on the “Attributes” tab appended with a:
  - For **WTPart** = `part_sa_{attribute name}`
  - For **WTDocument** = `doc_sa_{attribute name}`
  - For **EPMDocuments** = `epmdoc_sa_{attribute name}`

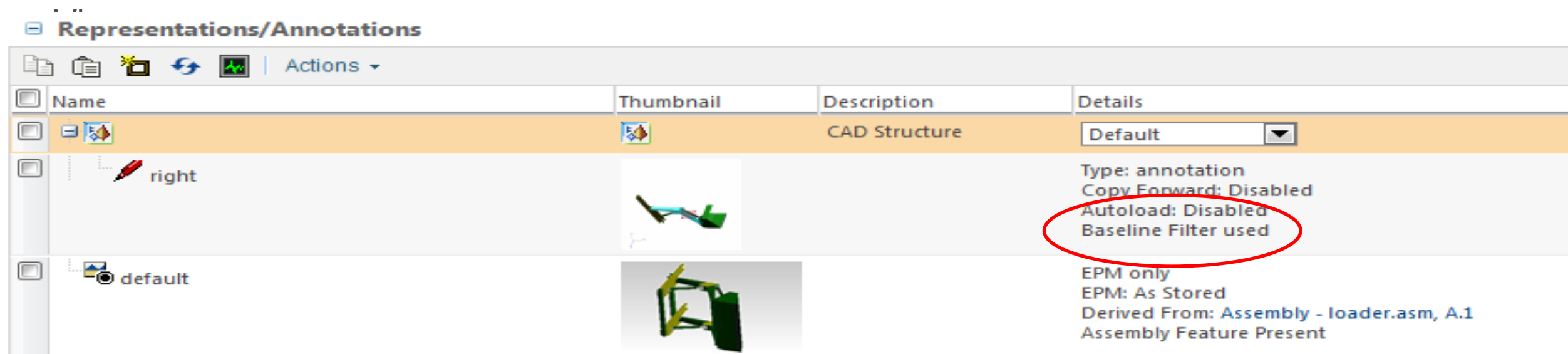
*No system changes required*

# Provide Open in PTC Creo View action in CAD Document Structure Tab

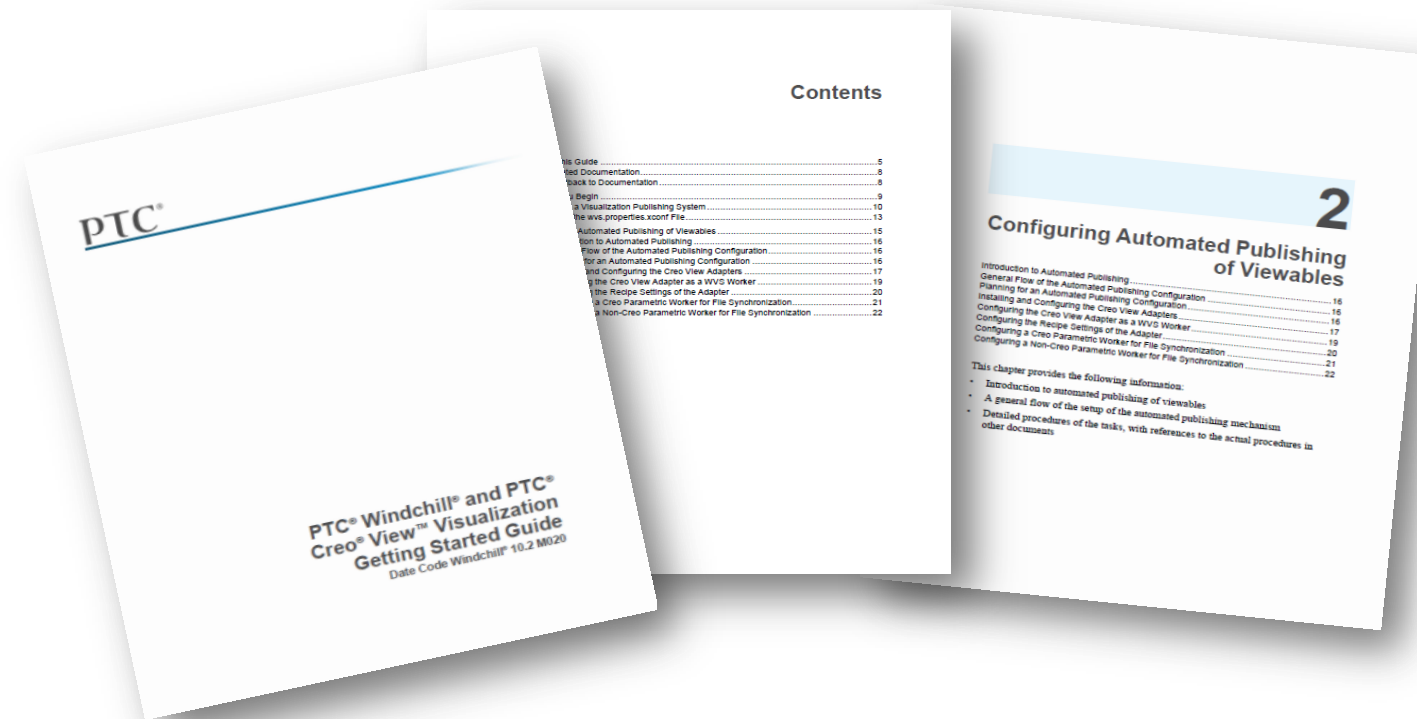
- Provide an action on the CAD Document Structure Tab which performs the same function as the “Open in PTC Creo View” action on the WTPart Structure Tab
  - Opens the Structure in PTC Creo View using the active filter being used in the Structure Tab
  - Provides the action in the right menu and the user interface ribbon



- Applies when viewing dynamic structures (Part structures or CAD Document structures)
  - When the annotation set is saved, a baseline is captured for the structure as opened in PTC Creo View, and that baseline is saved and associated to the annotation.
  - When that annotation set is opened in PTC Creo View from Windchill the saved baseline is applied to the structure, rather than the “default” filter being used.
  - If the user wants to see the annotation set with a different filter, the structure can be opened into PTC Creo View with the required filter from the structure row, and then the annotation set opened while in PTC Creo



## Improving the PTC Windchill Visualization Administrator User Experience



**Provides a step-by-step guide to getting started with PTC Windchill Visualization**

PTC Windchill and PTC Creo View Visualization Getting Started Guide

[Link to document](#)





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