

PTC® Live Global

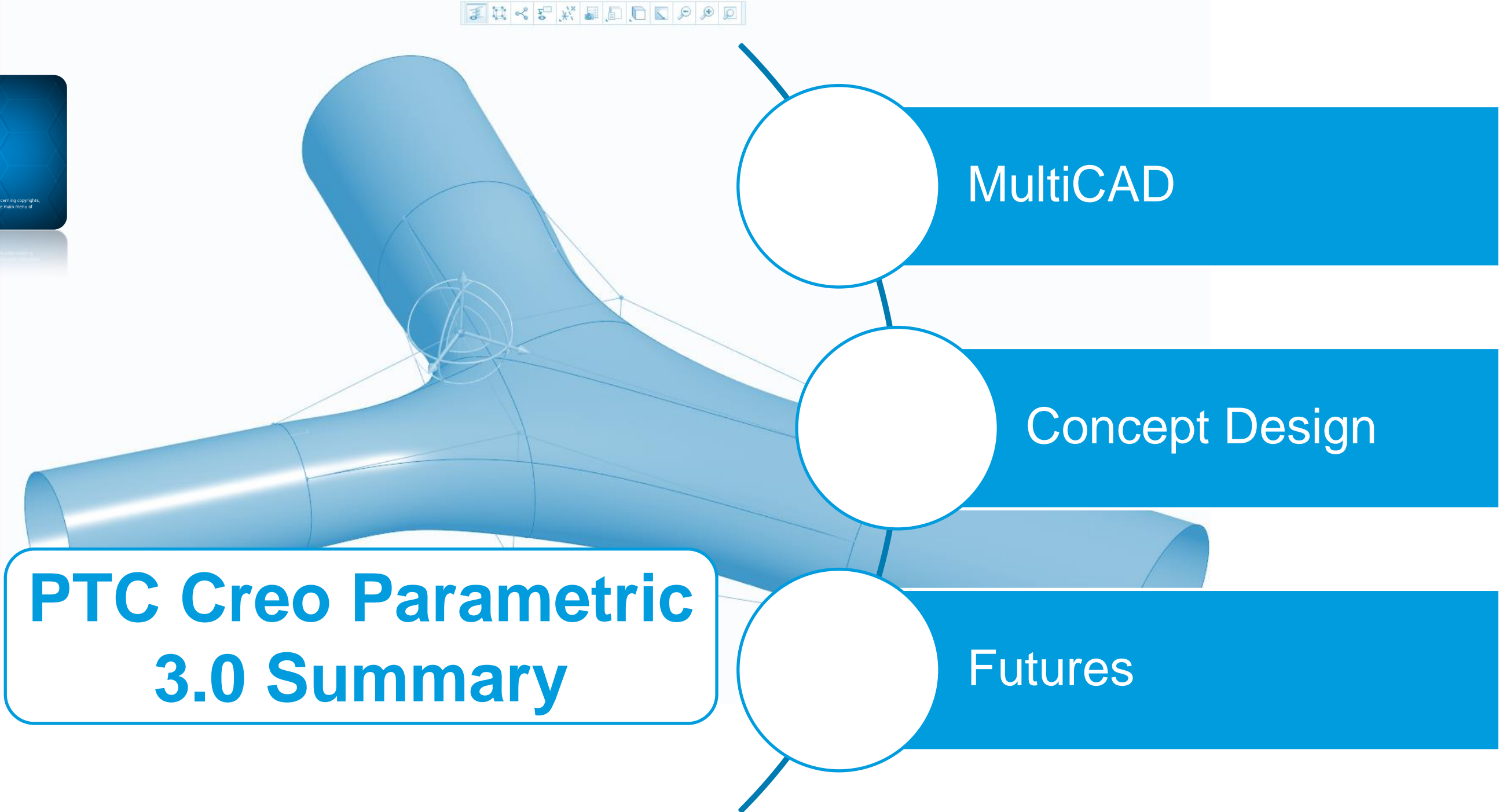
PTC 206 PTC Creo Parametric 3.0 and Beyond

Paul Sagar

Director of PTC Creo Product Management



Agenda



**PTC Creo Parametric
3.0 Summary**

MultiCAD

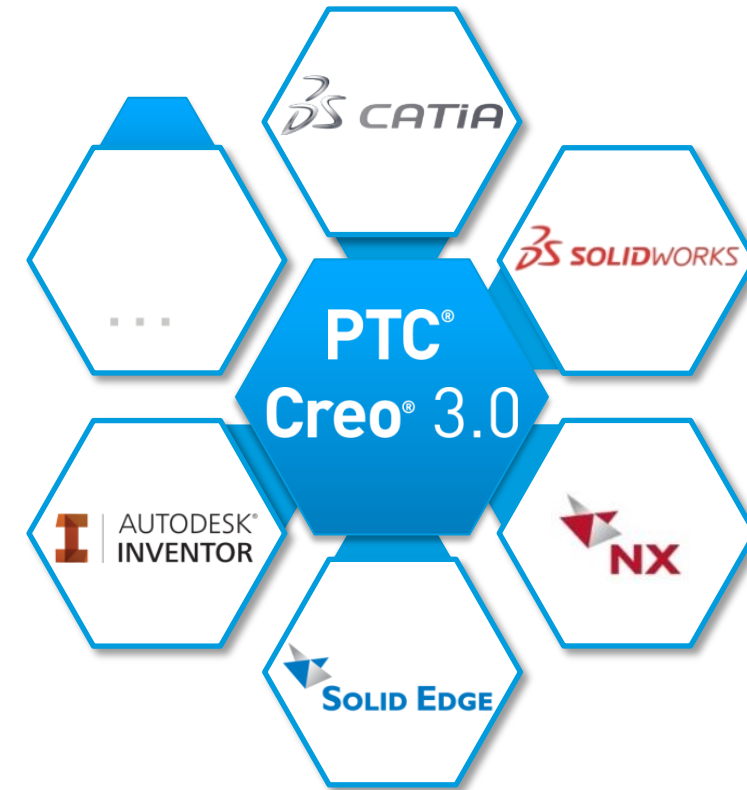
Concept Design

Futures

MultiCAD

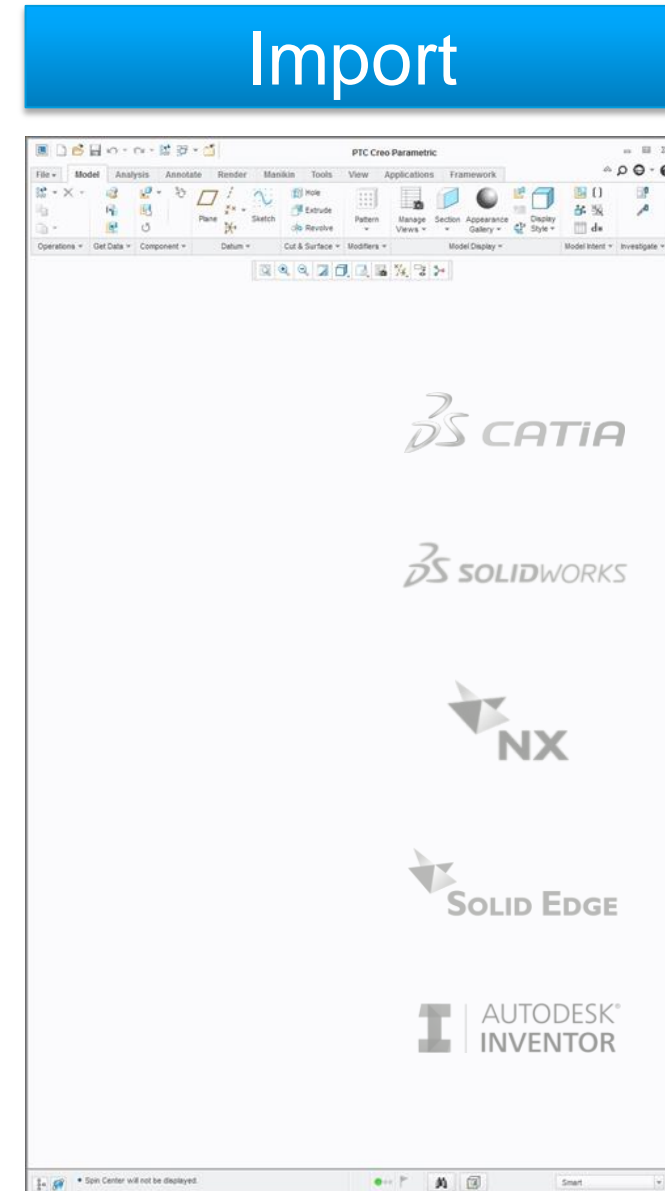
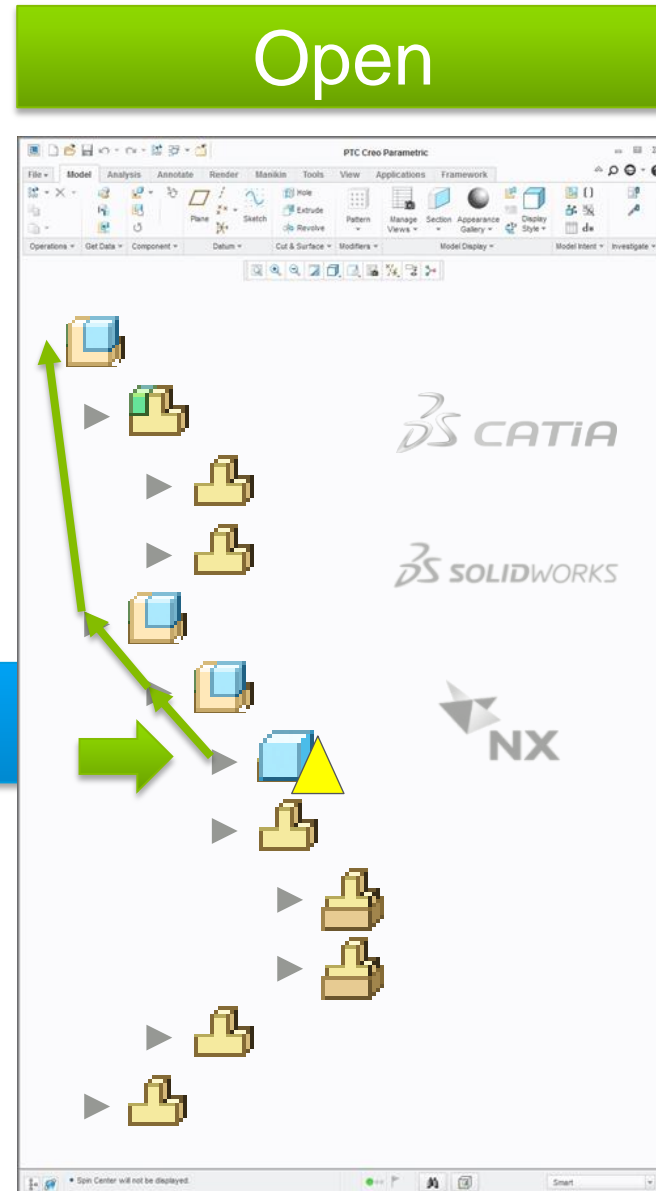
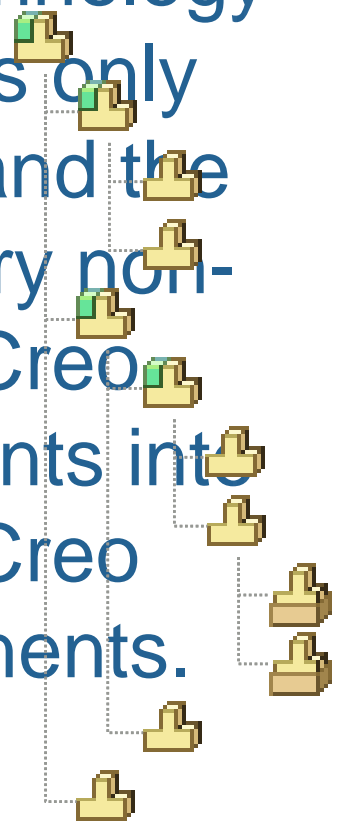
Enabling Optimal Multi-CAD Consolidation

- **Import common 3D CAD formats**
 - SolidWorks, CATIA, NX, SolidEdge, Inventor formats
 - No need for non PTC Creo software
 - **New** PTC Creo objects created
- **Open key 3D CAD formats**
 - SolidWorks, CATIA, NX formats
 - No need for non PTC Creo software
 - **No new** PTC Creo objects created
 - Convert to PTC Creo format when needed
- **Automatically update new versions of non-PTC Creo data within your designs**
- **Save As key 3D CAD formats**
 - SolidWorks, CATIA, NX formats



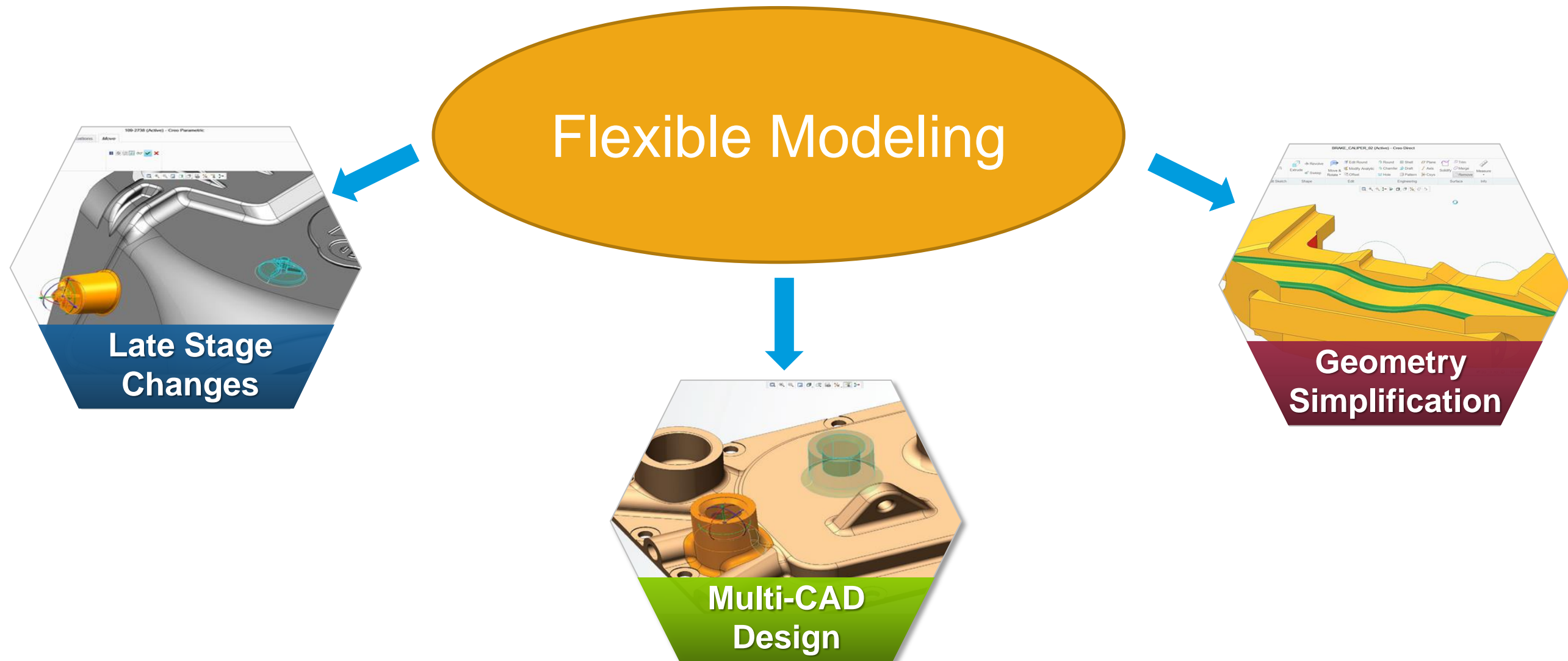
Save time and money with an effective convert as needed workflow

The PTC Creo Unite Technology converts only on-demand the necessary non-PTC Creo components into PTC Creo components.



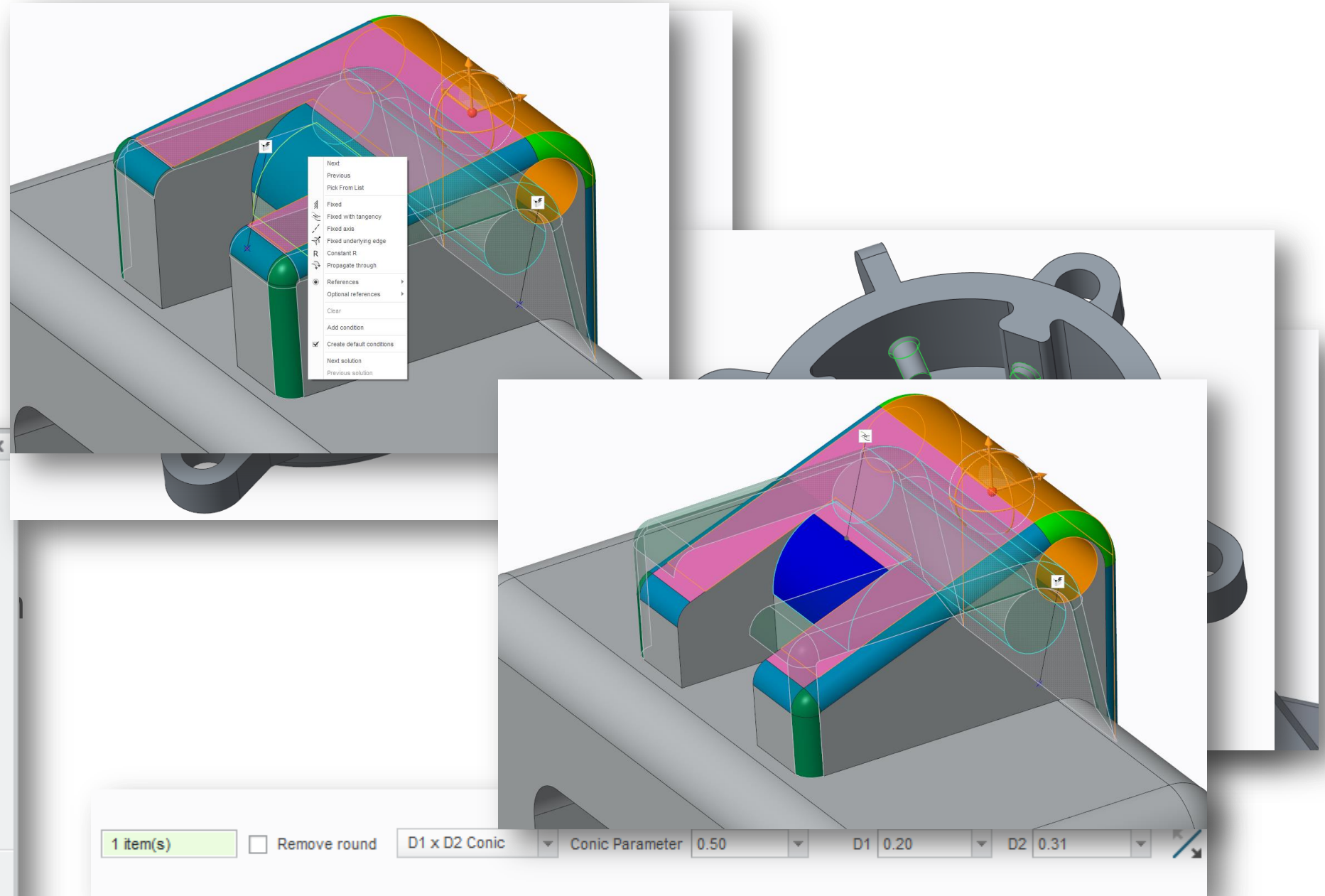
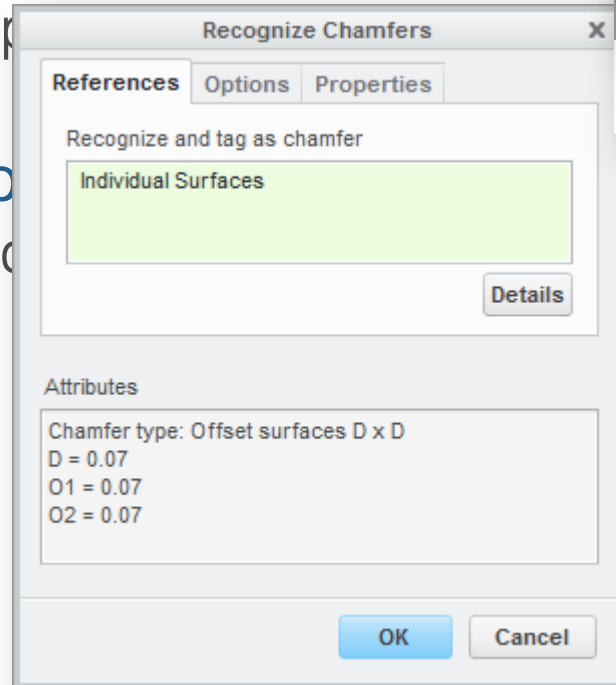
What is Flexible Modeling?

- A set of direct modeling capabilities for facilitating quick and easy changes to CAD models in the PTC Creo Parametric environment



What's New

- **Rounds and Chamfers**
 - Support for non-circular rounds
 - Chamfer handling and editing
 - Round/chamfer recognition feature
- **Patterns**
 - **New** Flex Pattern tool
 - Pattern properties
- **Tangency Properties**
 - Add conditions



Lets take a look

Concept Design

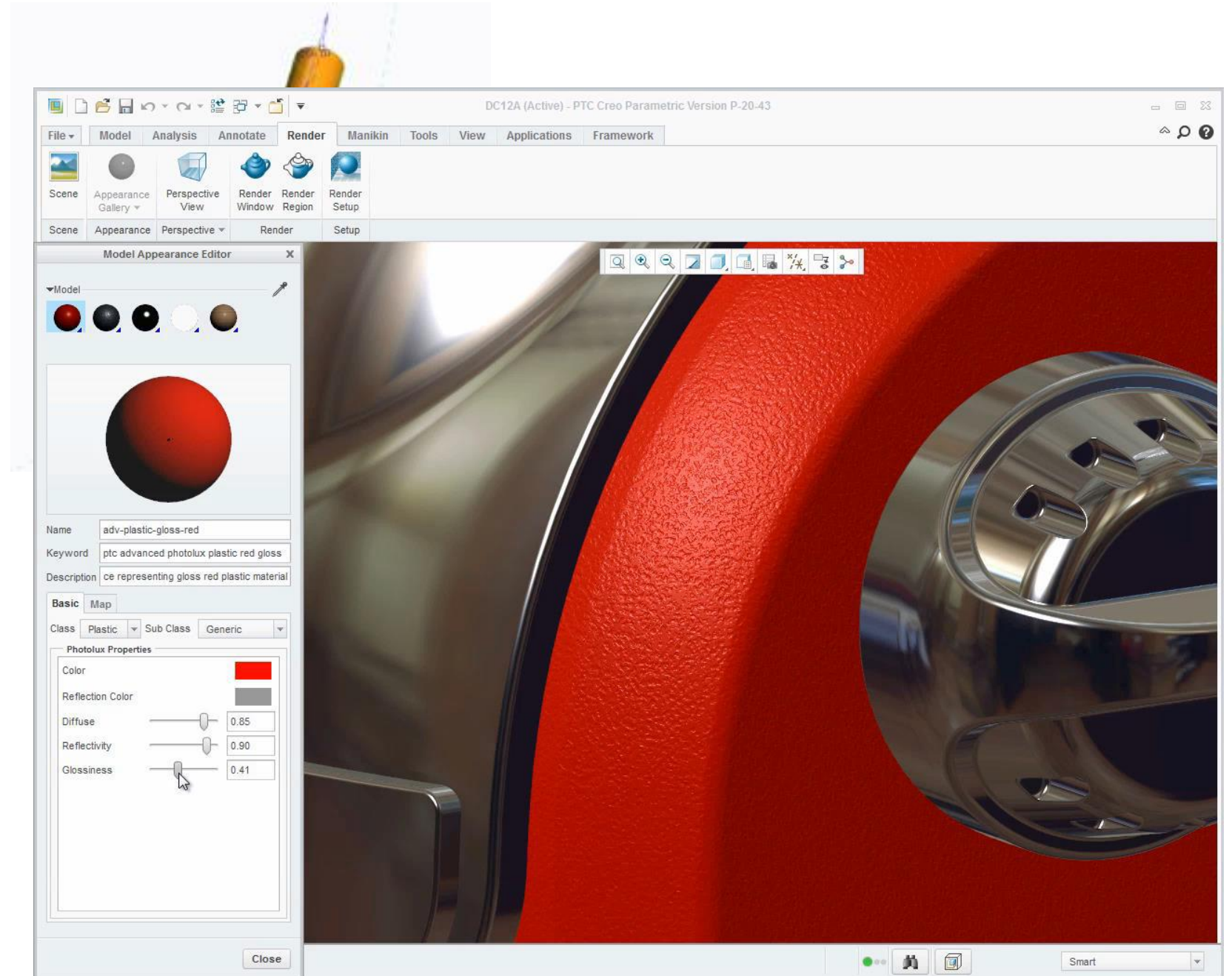
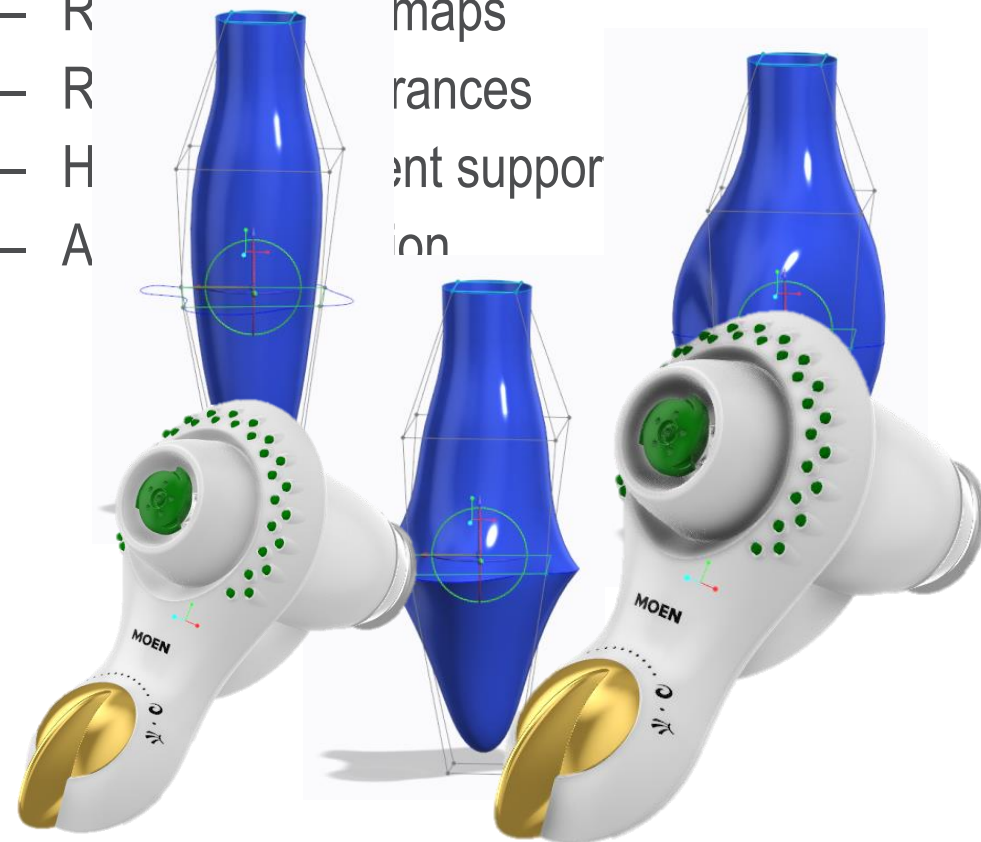
Surfacing and Graphics

- **Freestyle expansion**

- Drive Freestyle geometry parametrically
 - Align control mesh to external edges/curves
- Join faces or edges to change topology

- **Work in an immersive environment**

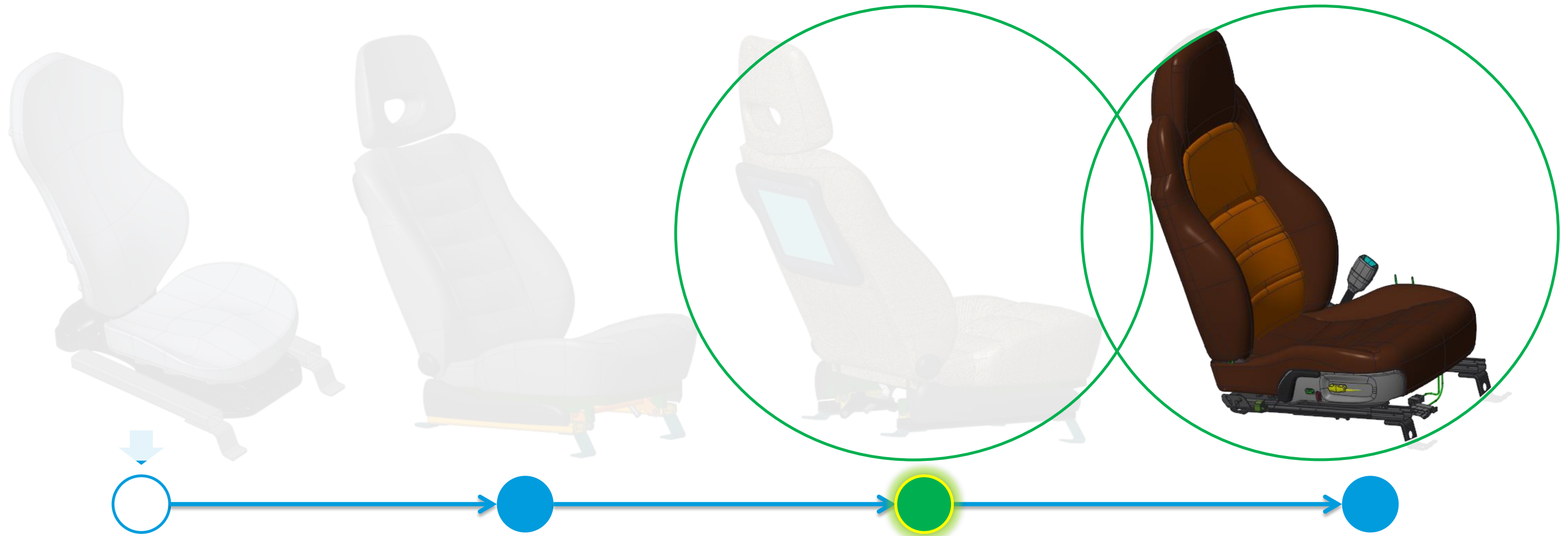
- R maps
- R rances
- H ent support
- A on



What do you do when you...

- **Try out ideas** for next version of the product?
- About to perform **significant changes** to a **complex design**?
- **Evaluate different concepts** for a product design?
- **Incorporate** new ideas into the “official” design?





- Snapshot entire session content
- Capture incremental changes

- Instantly toggle checkpoints
- Accept changes or revert risk-free

Lets take a look

Customer Ideas site

When item inside a group has failed, show the group as failed.
44 Voted on 46 times. You have not voted OPEN FOR VOTING
Created on Nov 13, 2013 10:57 AM by Justin Moore - Last Modified: Nov 13, 2013 10:57 AM

When a feature inside of a group has failed the group itself does not indicate it contains a failed feature. You have to expand the group to see if the failed feature is inside. If the group that contains the failed feature would also show as a failed feature it would make navigating the model tree much easier when fixing failed features.

Categories: Parametric Tags: none (add)

Add Edges to Draft Check Display
23 Voted on 23 times. You have not voted OPEN FOR VOTING
Created on May 10, 2013 6:13 AM by Doug Schaefer - Last Modified: May 10, 2013 6:14 AM

When performing a draft check, the model turns two color showing clearly where the break in draft is. However, all shading and edge information is not displayed, so the contour of the model is very difficult to read. Adding the edges to the draft check display would solve this.

Here are two images, one of the shaded model with edges and one of the model in draft check. You can see how adding the edges would improve things.

Show whether an Inherited part is up to date or not in the model tree.
14 Voted on 14 times. You have not voted OPEN FOR VOTING
Created on Oct 29, 2013 7:46 AM by Justin Moore - Last Modified: Oct 29, 2013 8:10 AM

When inheriting a part it is displayed in the model tree. It would be nice if you could look at your model tree and see if inherited parts were up to date with the version that is in your model tree (this is current functionality in creo elements / direct modeling and model manager). Currently you have to look in your workspace and see which version is in the workspace and then update the inheritance. There is no indicator as to whether you have "updated the inheritance" or need to.

Categories: Parametric Tags: none (add)

Creo 2.0 import support SolidWorks 2013
27 Voted on 27 times. You have not voted OPEN FOR VOTING
Created on May 24, 2013 6:58 AM by zhen zhen - Last Modified: May 24, 2013 6:58 AM

Refer to <https://www.ptc.com/appserver/cs/view/solution.jsp?n=CS28543>, Creo 2.0 import support SolidWorks only up to 2012, but now more and more suppliers are upgraded to 2013

Categories: Direct, Parametric, Simulate Tags (edit): solidworks, support, import, creo, 2.0, 2013

Allow draft angle to be higher then 30 degrees
25 Voted on 27 times. You have not voted OPEN FOR VOTING
Created on Jun 26, 2012 7:43 AM by Davor Granic - Last Modified: Jun 26, 2012 8:14 AM

Make drafting possible with higher value then 30 degrees. It's silly to use chamfer, sometimes to achieve this because of 30 degree limit.

Also make this possible in extrude taper option.

Categories: Parametric Tags (edit): taper, chamfer, draft, angle

Default Draft Check Should be 3 Colors
10 Voted on 10 times. You have not voted OPEN FOR VOTING
Created on Jan 24, 2014 8:55 AM by Doug Schaefer - Last Modified: Jan 24, 2014 8:55 AM

Revise the color scheme for draft check to be 3 colors. Assuming the user is checking for X degrees of draft, all surfaces should display in one of 3 colors:

- One color (Blue?) for draft greater than +X
- One color (Magenta?) for draft less than -X
- One color (Yellow?) for draft between +X and -X.

Categories: Parametric Tags (edit): draft_analysis

Orient View Normal to Axis
12 Voted on 14 times. You have not voted OPEN FOR VOTING
Created on May 4, 2012 8:54 AM by Bill Ryan - Last Modified: May 4, 2012 8:55 AM

The orient View option includes two axis options to orient vertical and horizontal. The option to view Normal to axis needs added. Reason: Verify hole alignment visually when working with brackets mounted to surfaces is one example.

The image below is from Wf5. I have confirmed with PTC technical support that the function is not available in Creo Parametric 2.0 using case 10799554.

Reorient an object "normal to" a planar surface or plane
15 Voted on 15 times. You have not voted OPEN FOR VOTING
Created on Apr 11, 2013 3:40 AM by NICOLAS BOURGEOIS - Last Modified: May 17, 2013 7:33 AM

- Need new option in "Normal To" menu to orient an object "Normal To" a selected plane or planar surface without a secondary reference.
- Or have the ability to directly orient an object "Normal To" a selected plane like it is possible in SolidWorks.

Need support for &sym(<name>) in 3D notes
14 Voted on 16 times. You have not voted OPEN FOR VOTING
Created on May 8, 2013 2:00 PM by Tim McLellan - Last Modified: May 8, 2013 2:00 PM

Please provide consistent interface and functionality for annotation across the product suite. Currently &sym(<symname>) cannot be placed in a 3D annotation.

<http://communities.ptc.com/ideas>

Futures

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