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Top Enhancements in PTC Creo® Parametric™ 3.0
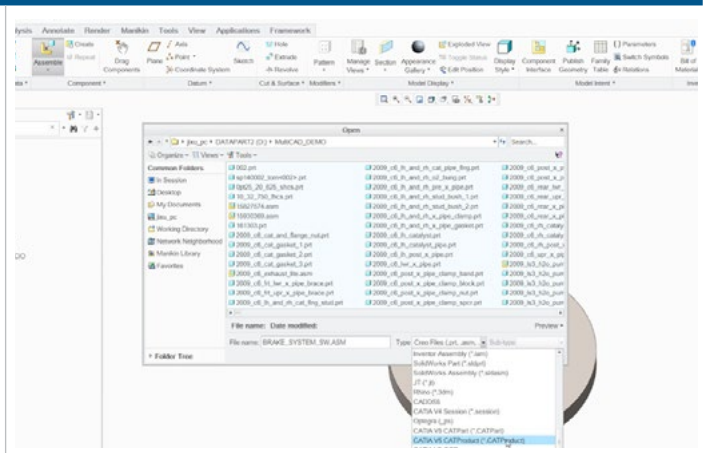
The Essential 3D Parametric CAD Solution

PTC Creo Parametric, PTC's 3D parametric modeling system, provides the broadest range of powerful yet flexible 3D CAD capabilities to help you address your most pressing design challenges. It uses proven technologies from Pro/ENGINEER®, plus hundreds of new capabilities to unlock design productivity. Here are some of the top enhancements in PTC Creo Parametric 3.0.

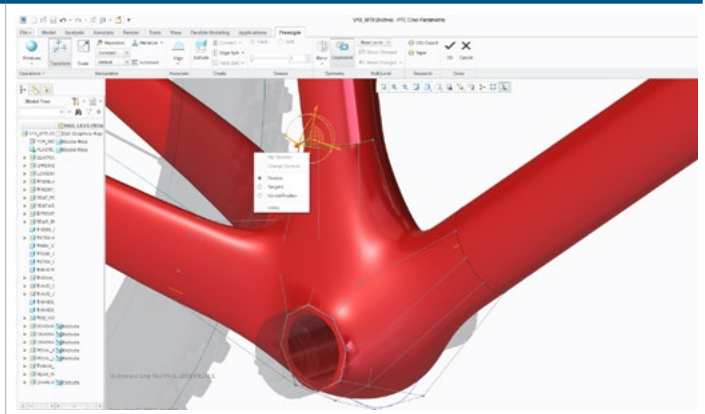## Unparalleled multi-CAD data handling capabilities

As well as being able to import neutral file formats (e.g., STEP, IGES, DXF, etc.) directly into PTC Creo Parametric, users can now both import and open CATIA®, SolidWorks®, and Siemens® NX™ files without the need for a separate translator or access to the native authoring software.

With the addition of collaboration extensions, users can also collaborate with non-native PTC Creo software and receive automatic updates of geometry changes from CATIA®, SolidWorks™, and Siemens NX™ inside PTC Creo Parametric.

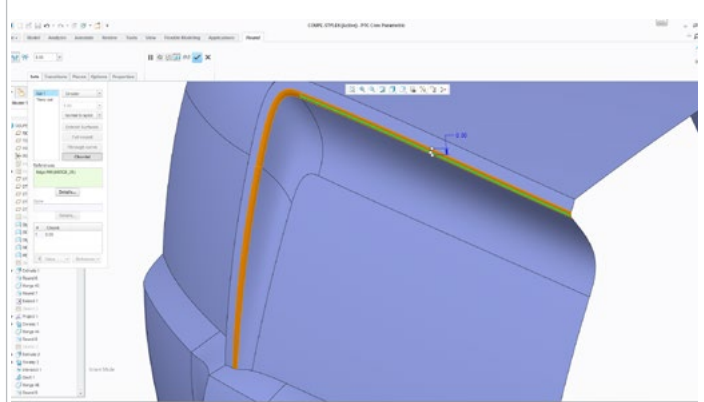## Drive Freestyle geometry parametrically

With the introduction of a new Align capability inside Freestyle, PTC Creo Parametric users can now create and drive their freeform, stylized designs parametrically. Users have the ability to connect their Freestyle geometry to other external geometry with positional, tangent, or normal conditions. Any changes made to this external geometry will automatically update the Freestyle geometry during regeneration, maintaining the appropriate connection. This allows users to more effectively combine freeform, organic geometry with dimension based design intent.
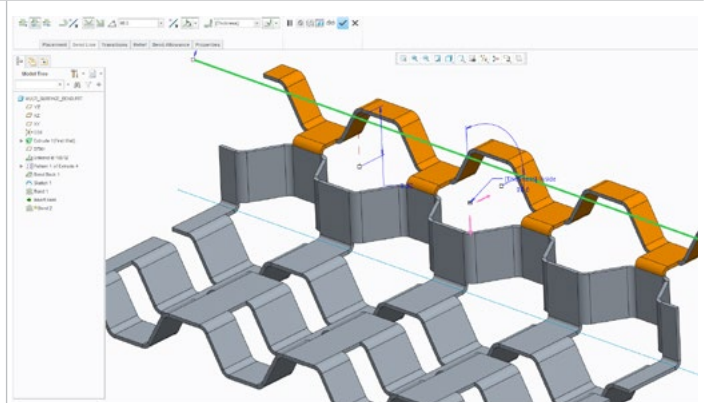
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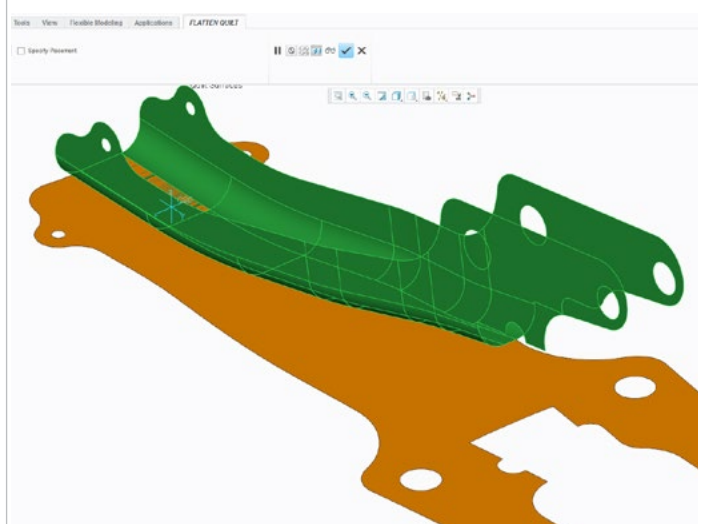## New and improved modeling capabilities

Chordal/constant width rounds: A new round option has been added in PTC Creo Parametric allowing users to create chordal rounds. In addition to specifying constant width, users can define round geometry using circular, conic, and C2 continuous cross-sections. Chordal rounds can eliminate the tedious process of having to construct round geometry using surfacing techniques.

Sheetmetal productivity: Rapidly create sheetmetal geometry using new functionality to automatically apply bend relief, create sheetmetal rips, bend coplanar geometry, and simplify die form and flat pattern creation.

Enhanced surface tools: Improve the quality of surface models using improved Boundary Blend functionality to create and optimize surface geometry and boundary connections. The ability to untrim surfaces and flatten complex freeform surfaces provides additional benefits when defining and working with surface geometry.
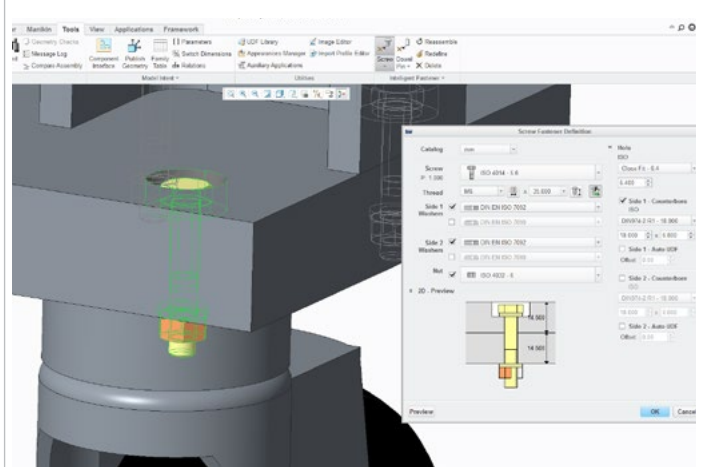
Advanced support for patterns: Geometry pattern capability has been extended in PTC Creo Parametric to allow users to pattern geometry across an area of topology change. This significantly improves the usage of geometry pattern and significantly improves overall productivity. In addition users can now reference pattern features that have been placed on any pattern instance, rather than needing to always place the features on the references of the pattern leader.

Extensive hardware libraries

PTC Creo Parametric includes a rich and extensive library of fasteners with a simple and intuitive user interface to speed up the process of adding standard hardware to your assembly design. In addition to assembling the hardware, the system will automatically create the appropriate clearance holes in the parts for you as well, where appropriate.
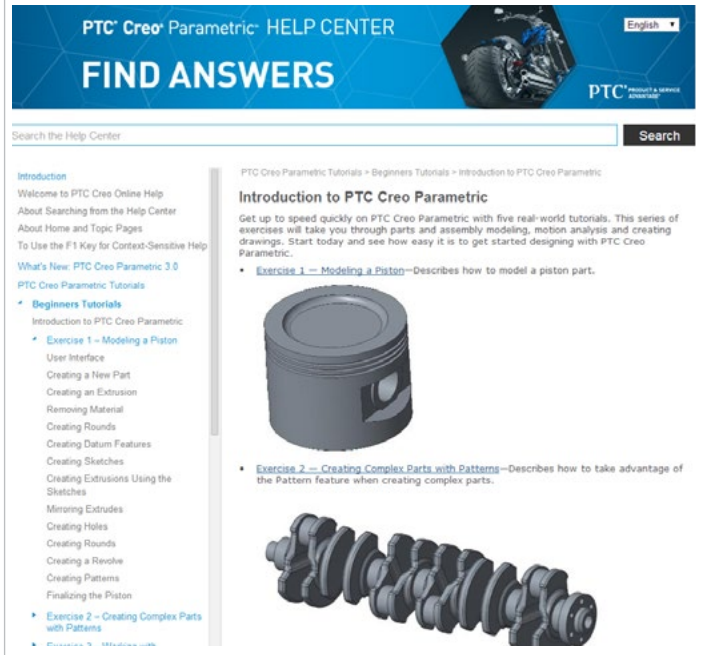
The supplier library catalog has also been extended inside PTC Creo Parametric to provide a single, localized interface with simple "google" style search for off the shelf components. These components can then be simply and easily brought into your PTC Creo assembly.

Integrated learning tools

PTC Creo Parametric ensures that new users are able to learn quickly, and experienced users are able to achieve higher levels of productivity. When starting the software, users will now see a dedicated window displaying the new content in the release. New getting started tutorials and getting started guides have also been added to the release allowing new users to quickly get up to speed with the basics.

PTC Creo Parametric also features a new and significantly improved help system including Google® indexing. Help is now fully searchable on the web offering faster access to relevant information when combined with the PTC Learning Connector.
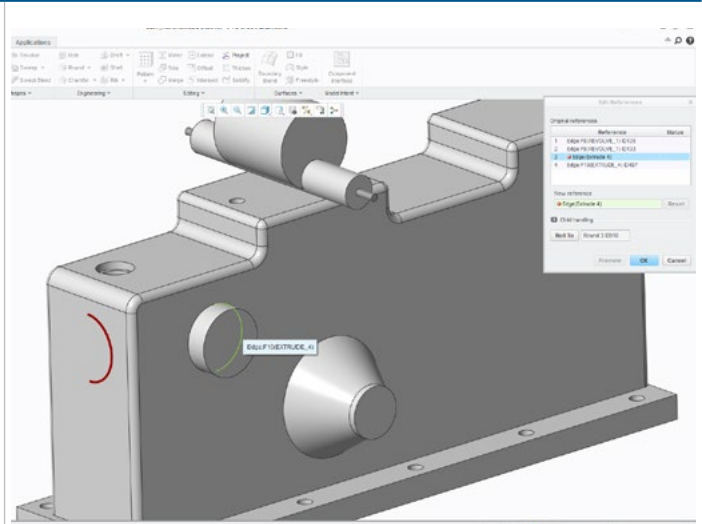
Intuitive reference editing

Edit References functionality has been significantly overhauled in PTC Creo Parametric. Not only has a new, intuitive interface been added, but users can now also see and choose child features to re-route simultaneously, significantly improving the process of changing references. In addition to edit references, Replace References has also been enhanced with a similar interface displaying all the children of the replacing references and the choice to replace the children or not.
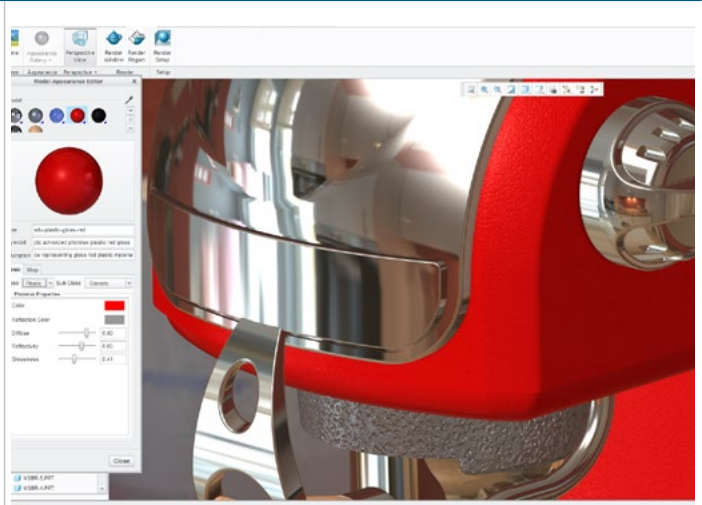
In addition to the new capabilities in edit and replace references, PTC Creo Parametric will now graphically display the old location of missing references. This allows users to more easily understand the old design intent and correct problems when they occur.
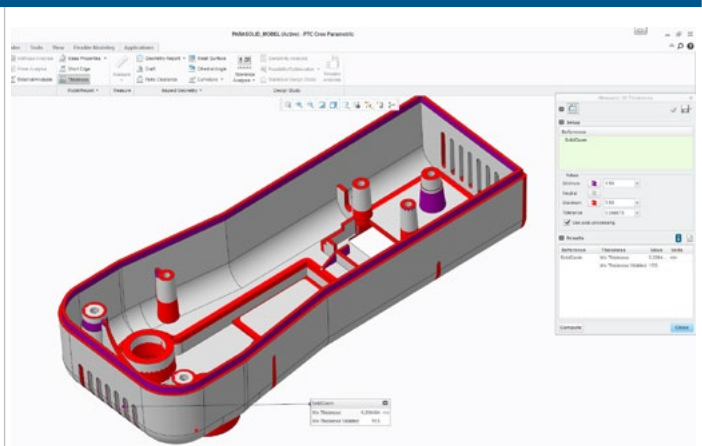

Enhanced graphic performance and quality

PTC Creo Parametric continues to see significant improvement in graphics performance and capability. Shade with reflections has been significantly improved to support realistic materials with access to different material classes that will each be displayed appropriately, such as metal, glass, paint, and plastic. In addition, bump maps are now realistically displayed in the graphics areas. Full support has been added for HDR images that can now show realistic reflections interactively on the geometry, offering control over the intensity, location, and size of the HDR image.

All these changes allow users to work in an immersive environment and see a representation of how the product will look in the real world.

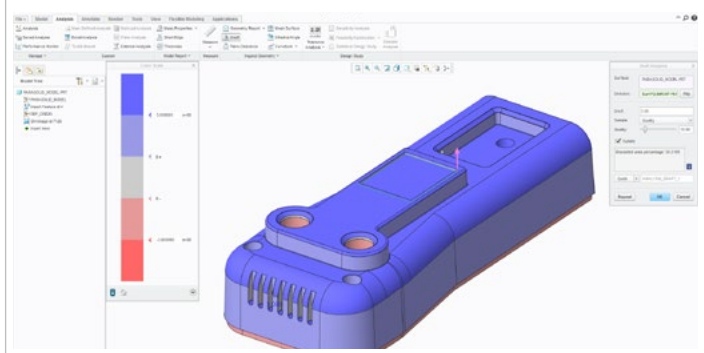## Powerful analysis and diagnostic tools

3D thickness checking: It is now possible to perform a 3D thickness check. The geometry is color coded based upon minimum and maximum allowable thickness specified by the user. The minimum thickness location will be displayed directly on the model. This allows designers to quickly determine the feasibility of the design before waiting for the design to be transferred to a Mold Design expert.
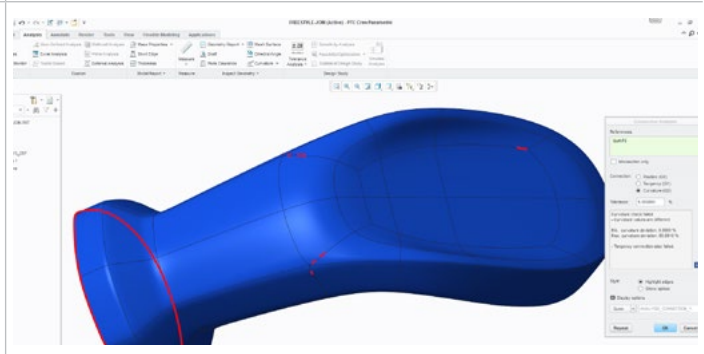
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Powerful analysis and diagnostic tools *(continued)*

Enhanced draft analysis display: Improvements have been made to the display of a draft analysis making the results easier to understand and interpret. Users will now see the lighting effect on the model as well as full support for seeing the model with edges displayed. In addition, users can now control the number of visible colors, with the ability to set this down to a minimum of three colors.
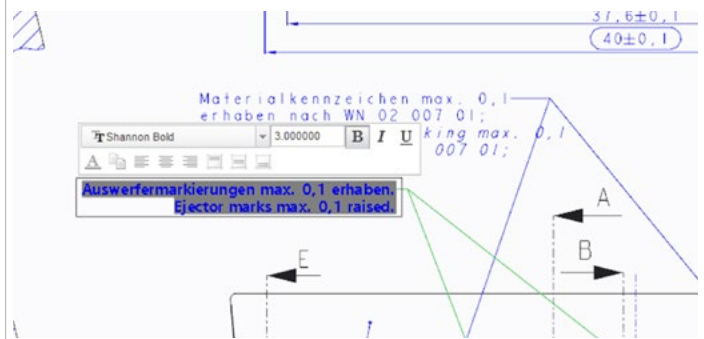

New connection analysis tools: PTC Creo Parametric introduces a new connection analysis tool to analyze position, tangency, and curvature continuity of curve and surface connections. Rather than select individual surface boundaries and curves, the user is able to quickly analyze the entire model to highlight connections that do not meet the design criteria.
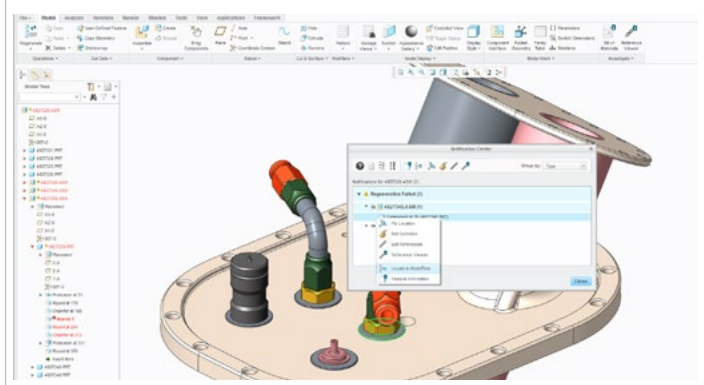

Enhanced 2D/3D detailing UI

The notes creation UI and dimension creation UI have been overhauled to provide WYSIWYG interaction following Microsoft® design standards. During definition, users are able to preview the placement and positioning of notes and dimensions. The functionality eliminates subsequent operations that are typically required to edit placement and display. Additional RMB shortcut menus and the removal of menu manager reduce mouse travel and increase user productivity.

Notification Center

PTC Creo Parametric introduces a new concept in the form of the Notification Center. This allows users to quickly see, interrogate, and action problems in their models. Problem items will be displayed directly in the model tree. If the model contains failed or outdated items, notifications will be displayed. Users can then open up the notification center to view and act upon them.
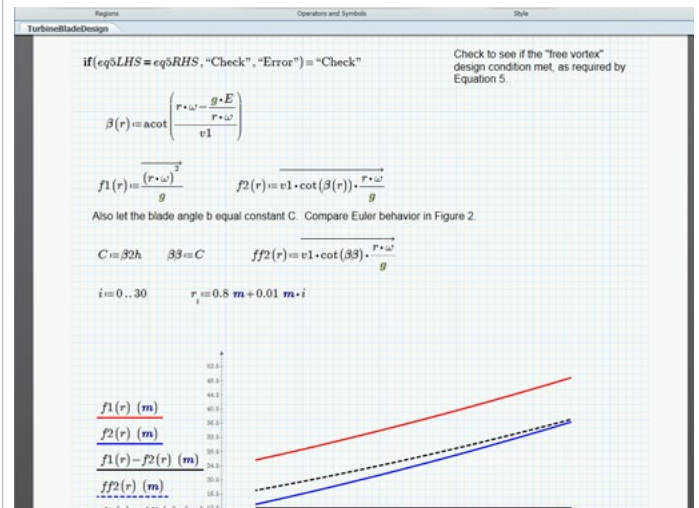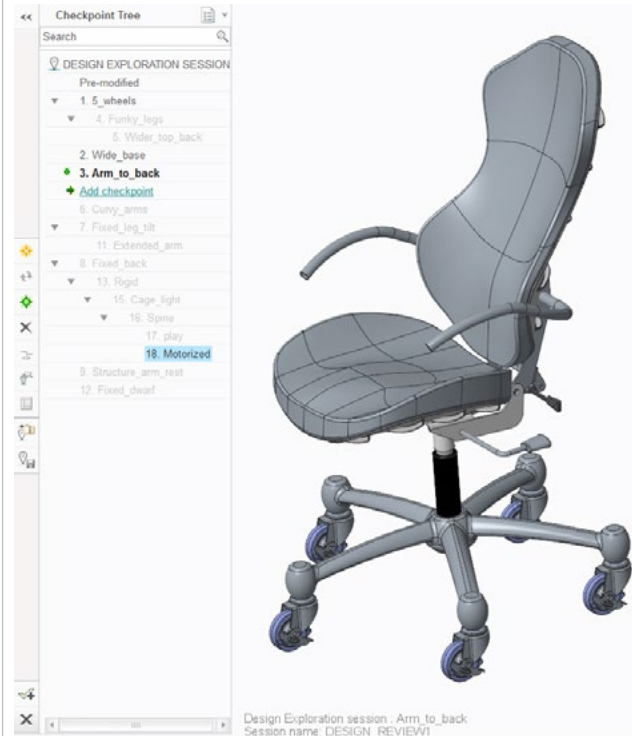
PTC Mathcad® integration

It is now possible to integrate a PTC Mathcad worksheet directly inside the PTC Creo Parametric part or assembly. Users can seamlessly read and write parameters with the worksheet.

*Available in PTC Creo 3.0 M020

And one more reason to upgrade, the PTC Creo Design Exploration Extension (DEX)

The PTC Creo Design Exploration Extension (DEX) is a dedicated tool for enabling the quick and easy investigation of alternative concept designs within the parametric environment. Using "checkpoints," DEX eliminates the need to manage multiple versions of data and allows designers to move seamlessly back and forth between design branches.

Upgrade to PTC Creo Parametric today.

For more information, visit **PTC.com/product/creo/parametric** or contact a PTC sales representative.

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