

# Capability Comparison of Creo Parametric 4.0

Creo Parametric offers powerful, reliable, yet easy-to-use modeling tools that accelerate the product design process. The software lets you design parts and assemblies, create manufacturing drawings, perform analysis, create renderings and animations, and optimize productivity across a full range of other mechanical design tasks. Creo Parametric will help you design higher quality products faster and allow you to communicate more efficiently with manufacturing and your suppliers.

This table highlights the primary product capabilities delivered in Creo Parametric 4.0 compared with Creo Parametric 3.0 and Creo Parametric 2.0

Creo Parametric Versions	2.0	3.0	4.0
<b>User Experience</b>			
Streamlined, familiar ribbon interface	•	•	•
Optimized, consistent user interface and workflows across sketching, part modeling, assembly modeling, drawing, 3D annotations, and a 3D dragger for moving components as desired	•	•	•
Embedded command finder for quickly locating capabilities in Pro/ENGINEER and SOLIDWORKS®	•	•	•
Real-time dynamic feature editing	•	•	•
Tutorials and help topics with the integrated PTC Learning Connector™, all available in context as needed	•	•	•
200+ free tutorials, available online through the PTC Learning Connector	•	•	•
Ribbon, hotkeys, and environment settings for configuring experience to individual preferences	•	•	•
Install Assistant to streamline license acquisition, license installation, product acquisition, and installation, including “Typical user” defaults	•	•	•
Dedicated “Getting Started” experience displaying e-learning and tutorials when product is started	•	•	•
Help content indexed on Google® and searchable via web		•	•
Automatic window activation		•	•
User configured RMB commands supporting individual setups		•	•
Notification Center to provide single access point to quickly track, sort, and fix common modeling issues		•	•
Geometry based selection providing intelligent context sensitive mini-toolbar, reducing mouse travel and increasing productivity			•
Box selection pervasive throughout the product			•
Fully customizable Mini-toolbar & Right Mouse Button			•
Ability to customize shortcut commands			•

Creo Parametric Versions	2.0	3.0	4.0
<b>Graphics</b>			
Full screen anti-aliasing option for improved display quality	•	•	•
Enhanced graphic performance and realistic materials out-of-the box		•	•
Easily switch to a full screen graphics mode reducing clutter			•
Appearance state definition to control different color combinations for the models			•

Creo Parametric Versions	2.0	3.0	4.0
<b>Sketcher</b>			
Intelligent right-click mouse mapping for fast menu picking	•	•	•
Improved action/object-based dimensioning for sketching	•	•	•
Center rectangle, Fillet with construction lines, and Construction mode options in Sketching mode	•	•	•
Sketching mode accessible by selecting an existing sketch, planar face, or datum	•	•	•
Section orientation tool	•	•	•
Context-sensitive ALT key options for adding references to sketches	•	•	•
Dimension draggers for isolating and changing individual dimensions within sketching when previewing features	•	•	•
Snap to existing geometry			•
Clearer display of dimensions/constraints			•
Clip geometry by sketch plane for improved visibility			•
Ability to programmatically drive sketched font			•

Creo Parametric Versions	2.0	3.0	4.0
<b>Part Modeling</b>			
Improved and intelligent workflows for Helical Sweep, Curve from equation, Curve from cross-section, and Curve through points	.	.	.
Tapered extrude option	.	.	.
More intuitive Parallel and Rotational Blend tools	.	.	.
Track changes feature available when loading models created in Creo Parametric that were modified in Creo Direct* (review, accept/reject, preview, and inspect)	.	.	.
Dynamic definition of cross sections via draggers or on-the-fly selection of planes	.	.	.
Cross sections accessible directly from the Model Tree	.	.	.
Real-time precise preview of resulting feature	.	.	.
Automatic regeneration of feature when making real-time changes to sketches	.	.	.
Multilevel subdivisional modeling in Freestyle for more control to the surface mesh, enabling finer detailing without altering the existing shape	.	.	.
Direct loading of data from Creo Sketch for creating new models	.	.	.
Ability to drive freeform geometry parametrically in Freestyle by aligning edges of Freestyle geometry with external geometry including: positional, tangent, or normal constraints		.	.
Chordal round option		.	.
Define round transitions using circular, conic, and C2 continuous cross sections		.	.
Ability to un-trim a surface or quilt		.	.
Connection analysis tool to analyze position, tangency, and curvature continuity of curve and surface connections		.	.
3D thickness check tool to analyze mold geometry		.	.
Draft analysis enhancements to make results easier to interpret		.	.
Redesigned reroute functionality		.	.
Easily position holes at any specified angle			.
Import/export freestyle control mesh			.
Support for multiple objects and enhanced splitting of the control mesh in freestyle			.
Enhanced capabilities and functionality for Sketch based feature			.
Ability to create a midplane			.
Maintain analytic geometry for warp features			.
Ability to create solid weld geometry			.
Simplified material assignment and out-of-the box standard materials			.
3D Printing – direct connection to Stratasys & 3D Systems 3D Printers as well as iMaterialize online print Bureau			.
3D Printing – Lattice feature (requires Creo Additive Manufacturing Extension). Ability to generate and optimize lattice structures enabling the designer to design in desired engineering responses			.

Creo Parametric Versions	2.0	3.0	4.0
<b>Assembly</b>			
Streamlined new component placement constraints	.	.	.
Fast loading of assemblies as lightweight graphics representation	.	.	.
Lightweight graphics representation for quickly restructuring, reordering, and renaming components in assembly designs	.	.	.
Additional constraint types (normal constraints for surfaces, distance, angle offset, and normal constraints for points and lines)	.	.	.
Enhances performance and user experience in "Chooser" tool		.	.
Simplified regeneration status and		.	.
Notification center improvements			.
Intelligent assembly mirror to simplify part reuse			.
Ability to store multiple color variations of a design using appearance states			.
Ability to create solid weld geometry			.

Creo Parametric Versions	2.0	3.0	4.0
<b>Sheetmetal</b>			
Dynamic real-time editing and preview of sheet metal parts, corner reliefs, etc.	.	.	.
Streamlined workflows in common tools such as bend and wall creation	.	.	.
Flat pattern preview and feature including highlighting of overlapping geometry	.	.	.
Form feature creation (with optional rounds and taper) or a partial piercing from a sketch	.	.	.
Sheet metal walls from two intersecting walls with a bend between them	.	.	.
Automatic creation of a flat pattern family table instance from a sheetmetal model	.	.	.
User interface and workflow for Die form		.	.
Flatten geometry attached to forms		.	.
Bend tool enhancements including the ability to bend multiple planes, bend line relief placements, and create multiple bend reliefs		.	.
Enhanced workflows and interface for twist wall creation			.
New capabilities for edge bend and edge treatment options			.
Ability to perform direct modeling based operations to sheetmetal parts, whether native Creo designs or imported geometry			.

Creo Parametric Versions	2.0	3.0	4.0
<b>Detailing</b>			
Optimized drawing tables workflow focused on most common user interactions	•	•	•
Enhanced workflow for creating and inserting tables	•	•	•
Tables Gallery for previews of predefined tables		•	•
Properties dialog for tables and BOM balloon regions		•	•
Text wrapping in table cells		•	•
Extended controls and setting for BOM balloons, including type and reference text		•	•
Dynamic repositioning of dimensions including snapping, free placement and locking dimension lines		•	•
New note and dimension creation user interface and format tab		•	•
New comprehensive text symbol palette and True-Type text fonts to support ASME and ISO standards			•
New Geometric Tolerance (GTOL) creation interface and workflow including syntax checking to ensure compliance with GD&T standards			•
New Datum Feature Symbol creation interface and workflow including Syntax checking of to ensure compliance with GD&T standards			•
New Datum Target creation interface and workflow including Syntax checking to ensure compliance with GD&T standards			•
Intelligent built-in standard target areas for Datum Targets (point, circle, rectangle)			•
Enhanced dimension creation and editing user interface and workflow			•
Quickly and easily add raster images into drawing without using Microsoft Windows OLE			•
Replace the model of a drawing view with a related model (family table, simplified rep, inheritance/merge) while preserving view settings and annotations			•
Support for non-linear cross hatching patterns using industry standard pattern file format (*.pat)			•

Creo Parametric Versions	2.0	3.0	4.0
<b>3D Annotations</b>			
Streamlined tools for creating and manipulating annotation content	.	.	.
Tools to create and manage combination states	.	.	.
Tools to create 3D annotations based on existing model dimensions	.	.	.
3D Detail options to control cosmetic appearance of annotations on models	.	.	.
3D Detail options to control appearance of annotations per 3D standards (ASME Y14.41-2003, ISO 16792, etc.)	.	.	.
Options to dynamically reposition dimensions in 3D drawings just like dynamic dimension repositioning in 2D drawings	.	.	.
Dynamic dragging of dimensions in 3D with intelligent snapping to center of witness lines and adjacent dimensions	.	.	.
Selection of endpoints of edges as references for dimension annotations in 3D	.	.	.
Selection of edges as edge references for dimensions if they are normal to the annotation plane	.	.	.
Symbols in 3D notes support model based definition		.	.
Print and Print Preview User Interface		.	.
New comprehensive text symbol palette and True-Type text fonts to support ASME and ISO standards			.
New Geometric Tolerance (GTOL) creation interface and workflow including syntax checking and semantic references to ensure compliance with GD&T standards			.
Datum reference frame object integrated into GTOL allows specification of datum reference frame coordinate system to ensure compliance with GD&T standards			.
New Datum Feature Symbol creation interface and workflow including Syntax checking and semantic references to ensure compliance with GD&T standards			.
New Datum Target creation interface and workflow including Syntax checking and semantic references to ensure compliance with GD&T standards			.
Intelligent built-in standard target areas for Datum Targets (point, circle, rectangle)			.
Support for movable Datum Target symbol to ensure compliance with GD&T standards			.
Enhanced dimension creation and editing user interface and workflow including support for semantic references of dimensions to ensure compliance with GD&T standards			.
Enhanced selection and dynamic movement of all annotations			.
Support for multiple appearances (color and texture) in the model that can be associated with combination states			.
Control visibility of annotations and supplemental geometry either by direct assignment to combination state or by using layers			.
Options to control publishing of combination states to Creo View and set the default combination state to be opened in Creo View			.
Print models with multiple combination states as a multi-page output – each combination state on a separate page			.

Creo Parametric Versions	2.0	3.0	4.0
<b>Data Exchange (included in Creo Parametric)</b>			
AutoCAD® 2012 drawing import*	•	•	•
Non-geometric data import from Creo Elements/ Direct®, including PMI across assemblies and mapping of docuplanes	•	•	•
Open CATIA®, NX®, and SOLIDWORKS files (maintain data natively)		•	•
Import CATIA, NX, SOLIDWORKS AutoDesk Inventor, and Solid Edge		•	•
Update and Export CATIA, NX, and SOLIDWORKS files**		•	•
Support for current STEP AP242 including defined Product Manufacturing Information (PMI)			•
JT support for cross-sections, exploded states and additional annotation types			•
Opening native Creo Elements/Direct models in Creo			•
Unified import/export profiles for non-Creo formats			•
Validation tool to compare key characteristics of native and converted data			•

\*Installation of relevant libraries required

\*\*Requires PTC Creo Collaboration Extension

For more information, visit: [PTC.com/cad/creo/parametric](http://PTC.com/cad/creo/parametric) or contact a PTC sales representative.

© 2016, PTC Inc. (PTC). All rights reserved. Information described herein is furnished for informational use only, is subject to change without notice, and should not be taken as a guarantee, commitment, condition or offer by PTC. PTC, the PTC logo, Product & Service Advantage, Creo, Elements/Direct, Windchill, Mathcad and all other PTC product names and logos are trademarks or registered trademarks of PTC and/or its subsidiaries in the United States and other countries. All other product or company names are property of their respective owners. The timing of any product release, including any features or functionality, is subject to change at PTC's discretion.

J7837–CapabilityComparisonofCreoParametric4.0–EN–1016