Capability Comparison of Creo Parametric 5.0

Creo is a 3D CAD solution that helps you build better products faster by accelerating product innovation, reusing the best of your design and replacing assumptions with facts. Go from the earliest phases of product design to a smart, connected product with Creo. Add augmented reality to allow everyone to visualize your design. In the fast-changing world of the Industrial IoT, no other company can get you to substantial value as quickly and effectively as PTC.

This table highlights the primary product capabilities delivered in Creo Parametric 5.0 compared with Creo Parametric 4.0 and Creo Parametric 3.0

Creo Parametric Versions	3.0	4.0	5.0
User Experience			
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		•	
Additional Commands for Showing and Hiding; Show only & Show all except			•
Modernized interaction handles			•
Modernized, intuitive, flexible model tree search in part & assembly modes			•
Automatic display of common filters in the Model Tree by default			•
Enhanced simple search in the model tree to dynamically list objects as typing a name			•

Creo Parametric Versions	3.0	4.0	5.0
Graphics			
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		•	•
Design in perspective			•
Modernized ModelCHECK report making it easier for user to identify issues in the data and resolve them			•

Creo Parametric Versions	3.0	4.0	5.0
Sketcher			
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		•	•
Dimension preview while dragging and dimension glyphs (indicating the dimension type)			•

Creo Parametric Versions	3.0	4.0	5.0
Part Modeling			
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		•	•
Volume Helical Sweep capability to create accurate geometry for grinding wheel and screw conveyor use cases			•
Faster redefinition of Feature Mirror			•
Sketch Region support allows re-use of sketches for several features			•
Easily apply drafts to design models containing rounds and chamfers			•

Part Modeling - continued			
Freestyle – Slice Freestyle shape by designated datum plane			•
Freestyle – Preview the objects before importing them into Freestyle.			•
Freestyle - Toggle between standard and box modes to rapidly design your freestyle surfaces			•
Freestyle – Utilize reference snapping when using Add Edge in Freestyle			•
Freestyle - Use **Align Curvature** to align shapes without losing the curvature continuity			•
Freestyle - Use the **Align** command to align edges to external curves or edges with G0, G1, G2, or G3 connections			•

Creo Parametric Versions	3.0	4.0	5.0
Assembly			
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		•	•
Ability to publish models to view as an Augmented Reality experience		•	•
Mechanism - Detailed diagnostics and resolution suggestions during Mechanism failures			•

Creo Parametric Versions	3.0	4.0	5.0
Sheetmetal			
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		•	•
New Types of Corner Reliefs, Normal and Square added			•
Additional control to Corner Relief orientation added			•
Improved flatten representation of sheet metal parts			•
Conversion is improved, by additional control to get unified sheet metal thickness			•

Creo Parametric Versions	3.0	4.0	5.0
Detailing			
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		•	•

Detailing - continued			
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)		•	•
Mini Toolbars for 2D Drawings			•
Improved Undo and Redo Support in Detailed Drawings			•
Improved Large Assembly Performance in Detailed Drawings through HLR multi-threading			•

Creo Parametric Versions	3.0	4.0	5.0
3D Annotations			
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		•	•
Mini Toolbars for 3D Annotations			•
Improved Undo and Redo Support in Model–Based Definition			•
Improved Failure Notifications for 3D Annotations			•

Creo Parametric Versions	3.0	4.0	5.0
Data Exchange (included in Creo Parametric)			
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		•	•
Transferring Configurations from Creo Elements/Direct to Explode States			•
Improved associative drawing Import from Creo Elements/Direct to support views containing Configuration information			•
3MF export			•
Open INVENTOR files (maintain data natively)			•
Easily select and define new import profiles for all required formats			•
Updated profile settings for Creo View Export to control model display settings			•
Updated Import Validation Tool improvements making it easier to identify and resolve issues			•
Support offset cross-sections in Step format			•

*Installation of relevant libraries required
**Requires PTC Creo Collaboration Extension

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

J10774–CapabilityComparisonofCreoParametric5.0–EN–0218