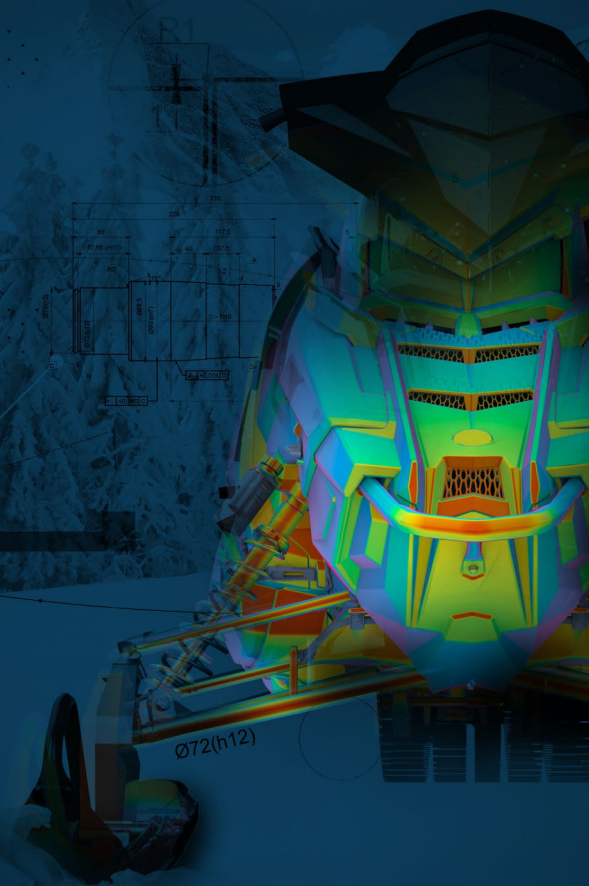





CREO PARAMETRIC CAPABILITY COMPARISON




 These tables highlight the primary product capabilities delivered in Creo Parametric 6.0 compared with Creo Parametric 5.0 and Creo Parametric 4.0

Creo Parametric Versions	4.0	5.0	6.0
User Experience			
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		.	.
Automatic saving of model tree setting			.
Enhanced model tree visibility			.
Mini-toolbar & Right Mouse Button within feature definition			.
Modernized feature dashboard with integrated help pages			.
Modernized charting tools			.

Creo Parametric Versions	4.0	5.0	6.0
Graphics			
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			.
Utilize Render Studio when outputting Mechanism and animation movies (requires Render Studio)			.
New emissive appearance (requires Render Studio)			.

Creo Parametric Versions	4.0	5.0	6.0
Sketcher			
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	4.0	5.0	6.0
Part Modeling			
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		.	.
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		.	.
New Project option for datum point creation			.
Created helical trajectory curve within Helical Sweep			.
Enhanced Mini-Toolbar support in Freestyle			.
Ability to suppress Freestyle shapes within Freestyle tree			.

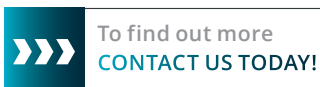
Creo Parametric Versions	4.0	5.0	6.0
Assembly			
Notification center improvements	.	.	.
Intelligent assembly mirror to simplify part reuse	.	.	.
Ability to store multiple color variations of a design using appearance states	.	.	.
Ability to create solid weld geometry	.	.	.
Ability to publish models to view as an Augmented Reality experience	.	.	.
Mechanism - Detailed diagnostics and resolution suggestions during Mechanism failures		.	.

Creo Parametric Versions	4.0	5.0	6.0
Sheetmetal			
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		.	.
Flat and flange wall enhancements			.
Enhanced workflows and interface for Merge Walls			.

Creo Parametric Versions	4.0	5.0	6.0
Detailing			
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)	.	.	.
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	4.0	5.0	6.0
3D Annotations			
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		.	.
Modernization of Notes workflow and interface for Notes			.
Enhanced parent/child behavior for annotations			.
Propagate all annotations during data sharing feature creation			.

Creo Parametric Versions	4.0	5.0	6.0
Data Exchange (included in Creo Parametric)			
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		•	•
3MF export to include appearances			•



Please visit the [PTC support page](#) for the most up-to-date platform support and system requirements.



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