creo®

CAPABILITY COMPARISON

CREO 4.0 – 8.0

ptc

Creo is a 3D CAD solution that helps you accelerate product innovation so you can build better products faster. Easy-to-learn Creo seamlessly takes you from the earliest phases of product design to manufacturing and beyond. You can combine powerful, proven functionality with new technologies such as generative design, augmented reality, real-time simulation, additive manufacturing and the IoT, to iterate faster, reduce costs and improve product quality. The world of product development moves quickly, and only Creo delivers the transformative tools you need to build competitive advantage and gain market share.

These tables highlight the primary product capabilities delivered in Creo 8.0 compared with Creo 7.0, 6.0, 5.0, and 4.0.

Creo Versions	4.0	5.0	6.0	7.0	8.0
User Experience					
Help content indexed on Google® and searchable via web	•	•	•	•	•
Automatic window activation	•	•	•	•	•
User configured RMB commands supporting individual setups	•	•	•	•	•
Geometry-based selection providing intelligent context-sensitive mini vtoolbar, 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			•	•	•
Improved material assignment workflow via the Model Tree and right mouse button command				•	•
Enhanced dashboard layouts to increase ease of learning					•
Ability to detach and position multiple dashboard option panels					•
New Quilts node in the model, listing individual quilts in the model tree					•
Design items group listing bodies & quilts at the top of the model tree					•
Create custom groups in the design item node containing quilts and bodies					•
View Design items tree side by side with Model tree					•

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User Experience - 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		•	•	•	•
Utilize Render Studio when outputting Mechanism and animation movies (requires Render Studio)			•	•	•
Custom floor orientation for rendering scene				•	•
Transparency display control for boundary (BREP) and mixed (facets) geometry in the view tab				•	•
Show datum planes as semi-transparent 3D objects to better understand datum plane position in 3D space					•
Global transparency control for bodies & quilts					•

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User Experience - 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		•	•	•	•
Multibody support for data sharing features, component operations and analysis tools				•	•
Inseparable assemblies for purchase parts - create a single file for a multi-component assembly					•
Checkpoints can be merged in a design exploration session					•

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User Experience - 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 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		•	•	•	•
New Project option for datum point creation			•	•	•
Created helical trajectory curve within Volume Helical Sweep			•	•	•
Enhanced Mini-Toolbar support in Freestyle			•	•	•
Ability to suppress Freestyle shapes within Freestyle tree			•	•	•

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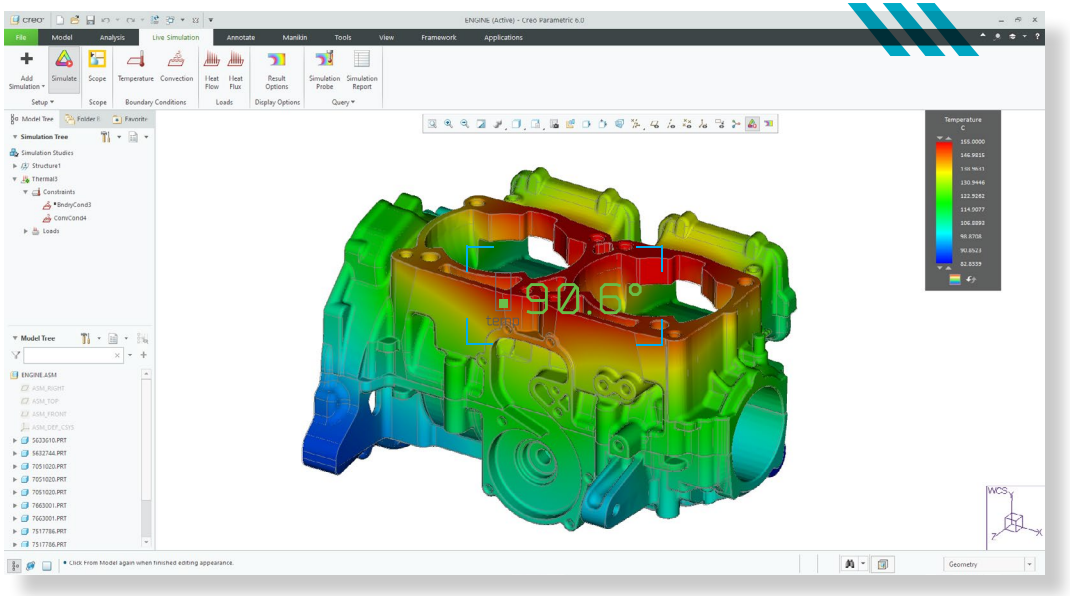
User Experience - PART MODELING (continued)					
Multibody concept for flexible part design methodologies				•	•
Multi-material models				•	•
Draft already drafted faces				•	•
Freestyle - New edit mode allowing users to snap selected control mesh vertices onto selected triangulation objects				•	•
Showing a snapshot preview of the quilt/body geometry at point in time after regeneration of the selected feature					•
Copy Quilt/Body geometry at selected point in regeneration sequence of the design model					•
Create multiple holes in one feature based upon sketch location					•
Create a straight drilled hole in combination with the tapped, tapered hole section					•
Lightweight holes available for all hole types					•
Option to Replace references of quilts recursively					•
Create a shortest distance curve on surface between 2 points					•



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User Experience - 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)		•	•	•	•
Improved graphical display of constraint icons in situations when they overlap sketched geometry			•	•	•
Improved design intent visualization (constraints and dimension references)				•	•
Easily mirror about any straight sketch entity				•	•
Enhanced control over dimension appearance in sketcher					•
Modernized dimension dragger. Easier to see, easier to grab & drag					•

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User Experience - SHEETMETAL					
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		•	•	•	•
Flat and flange wall enhancements		•	•	•	•
Enhanced workflows and interface for Merge Walls			•	•	•
Design Sheetmetal geometry in context of regular geometry (multibody)				•	•
Create multiple flat walls in one feature					•

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User Experience - 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	•	•	•	•	•
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		•	•	•	•
Streamlined and modernized sketching tools in drawings					•
Create construction draft entities in drawings					•
Create 2D views from selected draft entities					•

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User Experience - 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		•	•	•	•
Modernization of Notes workflow and interface for Notes			•	•	•
Enhanced parent/child behavior for annotations			•	•	•
Propagate all annotations during data sharing feature creation			•	•	•
Additional Indicator options in Geometric Tolerance (GTOL) creation interface				•	•
Use intelligent surface collection methods in standalone annotation					•
Streamlined placement and editing workflows for symbols					•
Interactive gallery for symbols					•
Modernized and streamlined symbol customization					•
Semantic reference support for symbols					•

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User Experience - DATA EXCHANGE (included with Creo)					
Open CATIA®, NX™, and SOLIDWORKS files (maintain data natively)	•	•	•	•	•
Import CATIA, NX, SOLIDWORKS Autodesk Inventor, and Solid Edge files	•	•	•	•	•
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		•	•	•	•
3MF export to include appearances			•	•	•
Import and Export multibody parts				•	•
Solid weld geometry exported to STEP, JT or Neutral as separate bodies					•

Please visit the PTC support page for the most up-to-date platform support and system requirements.

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