

CAPABILITY COMPARISON

CREO 2.0 – 7.0



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. And with cloud-based augmented reality in each seat of Creo, you can collaborate with anyone, instantly at any step in the product development process. In the fast-changing world of the Industrial IoT, no other company can get you to substantial value as quickly and effectively as PTC.



These tables highlight the primary product capabilities delivered in Creo 7.0 compared with Creo 6.0, 5.0, 4.0, 3.0, and 2.0.

Creo Versions	2.0	3.0	4.0	5.0	6.0	7.0
User Experience						
Simplified installation, setup, and customization for acquisition of license and updates	•	•	•	•	•	•
Improved measure tool focused on single summary command, improved usability, and cleaner results display	•	•	•	•	•	•
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						•





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User Experience - GRAPHICS						
Faster graphics performance taking greater advantage of the available graphics card	•	•	•	•	•	•
Order-independent transparency to improve performance and display quality	•	•	•	•	•	•
Smoother display of edges and more realistic display leveraging anti-aliasing	•	•	•	•	•	•
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						•

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User Experience - ASSEMBLY						
Enhances performance and user experience in "Chooser" tool		•	•	•	•	•
Simplified regeneration status		•	•	•	•	•
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						•



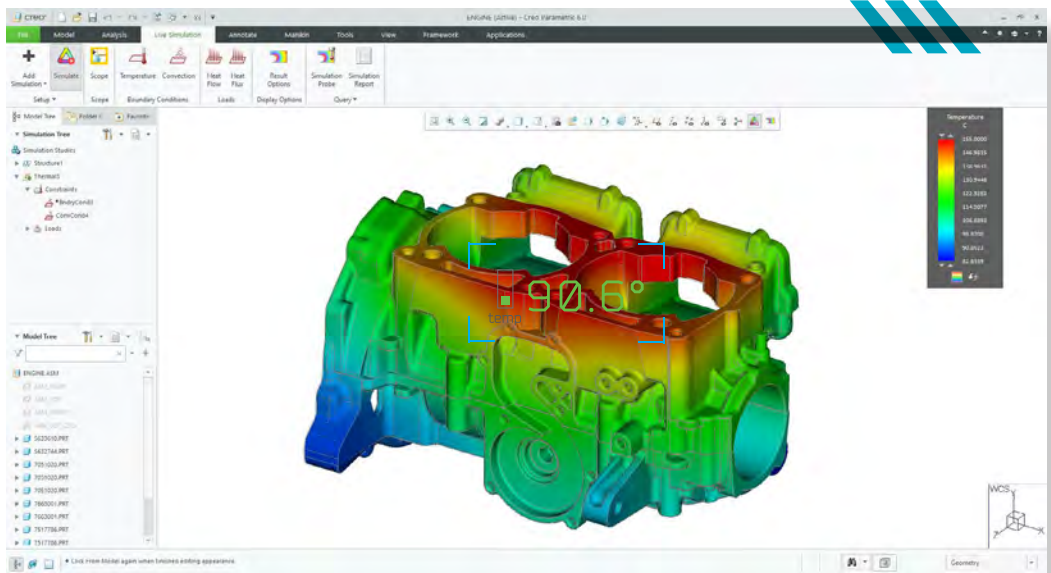
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User Experience - PART MODELING						
Simplified and streamlined workflows for creating parallel and rotational blends
New collapse command to consolidate Direct editing operations
Dynamics creation and editing of cross sections
Dynamically view cross section in separate 2D windows
Streamlined workflows for cross hatching placement
New Freestyle design capability leveraging subdivisional surface modeling for rapid freeform surface creation
Knot analysis tool to more easily understand surface and curve flow
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
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



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User Experience - PART MODELING (continued)						
Freestyle – Utilize reference snapping when using Add Edge in Freestyle				•	•	•
Freestyle – Use the Align command to align edges to external curves or edges with G0, G1, G2 connections				•	•	•
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					•	•
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						•





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User Experience - 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)				•	•	•
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						•

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User Experience - SHEETMETAL						
Ability to add a bend when joining two walls	•	•	•	•	•	•
Create sketch-based form features and partial piercings	•	•	•	•	•	•
Highlight overlapping geometry in the flat pattern preview window	•	•	•	•	•	•
Single click creation of flat pattern family table instance	•	•	•	•	•	•
Ability to add offset when creating a flat wall feature	•	•	•	•	•	•
Consistent corner relief placement options	•	•	•	•	•	•
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				•	•	•
Flat and flange wall enhancements				•	•	•
Enhanced workflows and interface for Merge Walls					•	•
Design Sheetmetal geometry in context of regular geometry (multibody)					•	•



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User Experience - DETAILING						
Dynamic dimension placement and movement
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				.	.	.



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User Experience - 3D ANNOTATIONS						
Dynamic dimension placement and movement
Edges that are perpendicular to the annotation plane can be selected as dimension references
Endpoints of edges can be selected as dimension references
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						.



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User Experience - DATA EXCHANGE (included with Creo)						
SOLIDWORKS license no longer needed for import
Support reading and writing of NX 7 data
Improved support for reading JT XT-Brep data
Transfer non geometric information from Creo Elements/Direct
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

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

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